



User Guide

Geomagic Design X | November 2013

Copyright Notice

©1993-2013. 3D Systems, Inc. All rights reserved. Produced in the USA. 11/30/13

The content of this manual is furnished for informational use only, is subject to change without notice, and should not be construed as a commitment by 3D Systems, Inc. Any names, places, and/or events in this publication are not intended to correspond or relate in any way to individuals, groups or associations. Any similarity or likeness of the names, places, and/or events in this publication to those of any individual, living or dead, place, event, or that of any group or association is purely coincidental and unintentional.

No warranties of any kind are created or extended by this publication. Any products and related material disclosed in this publication have only been furnished pursuant and subject to the terms and conditions of a duly executed agreement to license the Software. Any warranties made by 3D Systems, Inc. with respect to the Software described in this publication are set forth in the License Agreement provided with the Software and printed in this publication. As more definitively stated and set forth in the License Agreement, 3D Systems, Inc. does not and will not accept any financial or other responsibility that may result from use of the Software or any accompanying material including, without limitation, any direct, indirect, special or consequential damages.

Individuals or organizations using the Software should ensure that the user of this information and/or the Software complies with the laws, rules, and regulations of the jurisdictions with respect to which it is used. This includes all applicable laws concerning the export of technology and the protection of intangible or intellectual property rights. 3D Systems, Inc. asserts its rights in and will endeavor to enforce all proprietary rights embodied in the Software and this publication including, without limitation, all copyright, patent, trademark, and trade secrets or proprietary information. The only rights given to an individual or organization purchasing the Software are those explicitly set forth in the License Agreement. Other than as explicitly allowed in the License Agreement, copying the Software or this material (including any format or language translation) is prohibited absent the prior written consent of 3D Systems, Inc.

3D Systems, 3D Systems, Inc., Geomagic, Geomagic Capture, Geomagic Design, Geomagic Design X, Geomagic Design Direct, Geomagic Studio, Geomagic Verify, Geomagic Control, Geomagic Wrap, Geomagic Freeform, Geomagic Freeform Plus, Geomagic Sculpt, Geomagic Capture for SolidWorks, Geomagic Capture for SpaceClaim, and all product logos are either registered trademarks or trademarks of 3D Systems, Inc. All other trademarks within this user guide & tutorial are the property of their respective owners and are used for identification purposes only. Other than to identify this Software and publication, individuals or organizations purchasing the software are not entitled to use 3D Systems's trademarks without 3D Systems's prior written consent.

Correspondence regarding this publication should be directed to:

3D Systems, Inc. Geomagic Solutions.

Email:

geomagic.sales.americas@3dsystems.com

geomagic.sales.emea@3dsystems.com

geomagic.sales.japan@3dsystems.com

geomagic.sales.china@3dsystems.com

geomagic.sales.apac@3dsystems.com

geomagic.sales.korea@3dsystems.com

Web: www.geomagic.com/

Table of Contents

1. INTRODUCTION	1
1.1. Welcome to Geomagic Design X	2
1.2. Key Themes of Geomagic Design X	4
1.3. Features of Geomagic Design X	9
2. GETTING STARTED	15
2.1. System Requirement	15
2.2. Installation	18
2.3. What Can Geomagic Design X Do?	20
2.4. Basic Entity Terminology	21
2.5. Introduction to the Interface	28
2.5.1. User Interface Overview	28
2.5.2. Understanding User Interface Philosophy	33
Dialog Tree Convention	33
Selection Convention	35
Mode Convention	36
Docking Bar Convention	37
2.5.3. Navigating Entities	37
2.6. Data Processing	41
2.6.1. Changing Units	41
2.6.2. Importing Scan Data	43
2.6.3. Basic Scan Data Processing Workflow	45
2.6.4. Basic Reverse Design Workflow	46
2.6.5. Exporting a Model	49
3. UNDERSTANDING SCAN DATA	50
3.1. Why 3D Scanning?	50
3.1.1. Power of 3D Scan Data	50
3.1.2. Applications of 3D Scan Data	51
3.1.3. Is 3D Scanning the Only Step in Creating a CAD-Ready File?	52
3.2. Types of 3D Scan Data	54
3.2.1. Understanding Point Clouds	55
3.2.2. Understanding Mesh	58
3.2.3. Mesh vs. Point Cloud	62
4. UNDERSTANDING REVERSE DESIGN	63
4.1. What is Reverse Design?	63

4.2. Reverse Modeling Strategy	64
4.2.1. Mesh Modeling.....	66
4.2.2. Parametric Feature Modeling.....	68
4.2.3. Fitting Surface Modeling.....	70
4.2.4. Hybrid Modeling.....	71
4.3. Powerful Tools for Reverse Modeling	72
4.3.1. Mesh Buildup Wizard(TM).....	73
4.3.2. Auto Surfacing.....	74
4.3.3. Align Wizard(TM).....	75
4.3.4. Redesign Assistant(TM).....	76
4.3.5. Auto Sketch.....	77
4.3.6. Various Modeling Wizards(TM).....	78
4.3.7. Accuracy Analyzer(TM).....	80
4.3.8. LiveTransfer(TM).....	82
5. GETTING HELP	83
6. USER INTERFACE	84
6.1. Title Bar	85
6.2. Menu Bar.....	85
6.3. Mode	86
6.4. Toolbar	87
6.5. Docking Bars.....	91
6.5.1. Tree	93
6.5.2. Display	96
6.5.3. Accuracy Analyzer	105
6.5.4. Help.....	110
6.5.5. Viewpoint.....	111
6.5.6. Properties	112
6.5.7. Color Bar.....	113
6.5.8. Support.....	116
6.5.9. Image Folder.....	117
6.5.10. Console.....	119
6.5.11. Error List	119
6.5.12. Profile View.....	120
6.6. Viewing Window.....	121
6.7. Dialog Tree	122
6.8. Monitoring.....	125
6.8.1. Status Bar	125
6.8.2. Progress Bar	125
6.8.3. Time Indicator	125
6.9. Esc Key.....	125
6.10. Mouse Control	126

6.11. Shortcut Keys.....	127
6.12. Pop-up Menus.....	129
6.13. 3D Mouse	131
6.14. Quick Modeling	132

7. MENU 135

7.1. File	135
7.1.1. File > New	135
7.1.2. File > Open	136
7.1.3. File > Save.....	137
7.1.4. File > Save As.....	138
7.1.5. File > TeamPlatform	139
7.1.6. File > Export.....	144
7.1.7. File > Publish	146
File > Publish > Point Stream.....	146
File > Publish > ICF	150
7.1.8. File > Capture Screen	153
7.1.9. File > Print.....	155
7.1.10. File > Print Preview	156
7.1.11. File > Print Setup.....	157
7.1.12. File > LiveTransfer(TM)	158
File > LiveTransfer(TM) > To SolidWorks	176
File > LiveTransfer(TM) > To Siemens NX	180
File > LiveTransfer(TM) > To Creo(Pro/E).....	182
File > LiveTransfer(TM) > To AutoCAD.....	184
File > LiveTransfer(TM) > To Inventor	186
File > LiveTransfer(TM) > To Solid Edge	188
File > LiveTransfer(TM) > Save As CATIA V4.....	191
File > LiveTransfer(TM) > Save As CATIA V5.....	193
7.1.13. File > Batch Process.....	194
File > Batch Process > Design.....	194
File > Batch Process > Run.....	197
7.1.14. File > Preferences	200
7.1.15. File > Summary Info	211
7.1.16. File > Recent Files	212
7.1.17. File > Exit.....	213
7.2. Select.....	213
7.2.1. Select > Mode	215
7.2.2. Select > All.....	218
7.2.3. Select > None	219
7.2.4. Select > Inverse	220
7.2.5. Select > Custom Region	221
7.2.6. Select > Boundary Entities	222
7.2.7. Select > Entities Around Edges/Faces	223
7.2.8. Select > Enlarge.....	224
7.2.9. Select > Shrink.....	227
7.2.10. Select > Convert Current Selection	228
7.2.11. Select > By Criteria	229
7.2.12. Select > By Color.....	232
7.2.13. Select > Exterior Poly-Face	233

7.3. Edit	235
7.3.1. Edit > Undo	236
7.3.2. Edit > Redo	236
7.3.3. Edit > Repeat Last Command	237
7.3.4. Edit > Cut	237
7.3.5. Edit > Copy	238
7.3.6. Edit > Paste	239
7.3.7. Edit > Delete	240
7.3.8. Edit > Rebuild	240
7.3.9. Edit > Rollback	241
7.3.10. Edit > Roll Forward	241
7.3.11. Edit > Roll To End	242
7.3.12. Edit > Suppress	243
7.3.13. Edit > Unsuppress	243
7.3.14. Edit > Feature	244
7.4. Insert	244
7.4.1. Insert > Import	246
7.4.2. Insert > Mesh	255
7.4.3. Insert > Region Group	258
7.4.4. Insert > Point Cloud	259
7.4.5. Insert > Ref. Geometry	261
Insert > Ref. Geometry > Plane	262
Insert > Ref. Geometry > Vector	272
Insert > Ref. Geometry > Coordinate	280
Insert > Ref. Geometry > Polyline	284
Insert > Ref. Geometry > Point	291
7.4.6. Insert > Solid	298
Insert > Solid > Extrude	299
Insert > Solid > Revolve	304
Insert > Solid > Sweep	308
Insert > Solid > Loft	312
Insert > Solid > Offset	317
Insert > Solid > Thicken	318
Insert > Solid > Hollow	320
Insert > Solid > Cut	323
Insert > Solid > Boolean	325
7.4.7. Insert > Surface	327
Insert > Surface > Mesh Fit	329
Insert > Surface > Boundary Fit	335
Insert > Surface > Auto Surfacing	342
Insert > Surface > Extrude	351
Insert > Surface > Revolve	354
Insert > Surface > Sweep	357
Insert > Surface > Loft	361
Insert > Surface > Offset	366
Insert > Surface > Fill Face	369
Insert > Surface > Extend	372
Insert > Surface > Trim	375
Insert > Surface > Untrim	378
Insert > Surface > Trim & Merge	380
Insert > Surface > Sew	381
Insert > Surface > Reverse Normal	385
7.4.8. Insert > Modeling Wizard	387
Insert > Modeling Wizard > Solid Primitives	387

	Insert > Modeling Wizard > Surface Primitives.....	391
	Insert > Modeling Wizard > Extrusion Wizard.....	395
	Insert > Modeling Wizard > Revolution Wizard.....	399
	Insert > Modeling Wizard > Sweep Wizard.....	403
	Insert > Modeling Wizard > Loft Wizard.....	407
	Insert > Modeling Wizard > Pipe Wizard.....	410
7.4.9.	Insert > Modeling Feature.....	415
	Insert > Modeling Feature > Fillet.....	416
	Insert > Modeling Feature > Chamfer.....	423
	Insert > Modeling Feature > Draft.....	425
	Insert > Modeling Feature > Emboss.....	428
	Insert > Modeling Feature > Linear Pattern.....	432
	Insert > Modeling Feature > Circular Pattern.....	434
	Insert > Modeling Feature > Curve Pattern.....	437
	Insert > Modeling Feature > Mirror.....	440
	Insert > Modeling Feature > Helix.....	443
	Insert > Modeling Feature > Spiral.....	447
	Insert > Modeling Feature > Move Face.....	451
	Insert > Modeling Feature > Delete Face.....	452
	Insert > Modeling Feature > Replace Face.....	454
	Insert > Modeling Feature > Split Face.....	457
	Insert > Modeling Feature > Transform Body.....	459
	Insert > Modeling Feature > Duplicate Body.....	464
	Insert > Modeling Feature > Remove Body.....	465
7.4.10.	Insert > CAD Correct.....	467
	Insert > CAD Correct > Refit.....	467
7.4.11.	Insert > Mesh Sketch.....	472
7.4.12.	Insert > Sketch.....	479
7.4.13.	Insert > 3D Mesh Sketch.....	481
7.4.14.	Insert > 3D Sketch.....	483
7.4.15.	Insert > Scanner Direct Control.....	484
	Insert > Scanner Direct Control > LiveScan(TM).....	485
	Insert > Scanner Direct Control > Geomagic Capture.....	502
	Insert > Scanner Direct Control > Creafom Portable 3D Scanners.....	508
	Insert > Scanner Direct Control > Z Corporation ZScanner 700.....	510
	Insert > Scanner Direct Control > Konica Minolta VIVID.....	512
	Insert > Scanner Direct Control > Breuckmann.....	517
	Insert > Scanner Direct Control > KODEN OCM-A (32 bit only).....	519
	Insert > Scanner Direct Control > Roland LPX (32 bit only).....	520
	Insert > Scanner Direct Control > Konica Minolta RANGE5/RANGE7 (64 bit only).....	522
	Insert > Scanner Direct Control > 3D Digital Scanner (32 bit only).....	523
	Insert > Scanner Direct Control > Shining3D.....	524
7.5.	Tools.....	526
7.5.1.	Tools > Scan Tools.....	527
	Tools > Scan Tools > Mesh Buildup Wizard.....	528
	Tools > Scan Tools > Triangulate.....	540
	Tools > Scan Tools > Align Between Scan Data.....	548
	Tools > Scan Tools > Merge.....	554
	Tools > Scan Tools > Combine.....	558
	Tools > Scan Tools > Decimate Meshes.....	560
	Tools > Scan Tools > Normal Information Manager.....	563
	Tools > Scan Tools > Transform Scan Data.....	565
	Tools > Scan Tools > Average Meshes.....	569

	Tools > Scan Tools > Boolean	573
7.5.2.	Tools > Mesh Tools.....	576
	Tools > Mesh Tools > Find Defect	578
	Tools > Mesh Tools > Healing Wizard	581
	Tools > Mesh Tools > Fix Normal.....	584
	Tools > Mesh Tools > Edit Boundaries.....	586
	Tools > Mesh Tools > Sew Boundaries	594
	Tools > Mesh Tools > Retriangulate	596
	Tools > Mesh Tools > Global Remesh	597
	Tools > Mesh Tools > Optimize Mesh	600
	Tools > Mesh Tools > Edit Poly-Edge	606
	Tools > Mesh Tools > Virtual Rescan	608
	Tools > Mesh Tools > Rewrap	613
	Tools > Mesh Tools > Fill Holes.....	617
	Tools > Mesh Tools > Defeature.....	622
	Tools > Mesh Tools > Remove Marker	624
	Tools > Mesh Tools > Add Poly-Face	626
	Tools > Mesh Tools > Smooth	628
	Tools > Mesh Tools > Smart Brush.....	630
	Tools > Mesh Tools > Decimate	634
	Tools > Mesh Tools > Subdivide	637
	Tools > Mesh Tools > Enhance Shape.....	641
	Tools > Mesh Tools > Refit	643
	Tools > Mesh Tools > Insert.....	647
	Tools > Mesh Tools > Split	649
	Tools > Mesh Tools > Trim.....	652
	Tools > Mesh Tools > Mirror	655
	Tools > Mesh Tools > Offset	656
	Tools > Mesh Tools > Thicken	659
	Tools > Mesh Tools > Convert Body	662
7.5.3.	Tools > Region Tools	665
	Tools > Region Tools > Auto Segment	665
	Tools > Region Tools > Resegment	668
	Tools > Region Tools > Merge	670
	Tools > Region Tools > Split	671
	Tools > Region Tools > Insert.....	674
	Tools > Region Tools > Separate	675
	Tools > Region Tools > Enlarge	676
	Tools > Region Tools > Shrink	678
7.5.4.	Tools > Point Cloud Tools.....	679
	Tools > Point Cloud Tools > Filter Noise.....	680
	Tools > Point Cloud Tools > Sampling.....	682
	Tools > Point Cloud Tools > Smooth	685
	Tools > Point Cloud Tools > Offset.....	687
	Tools > Point Cloud Tools > Construct Mesh	689
	Tools > Point Cloud Tools > Convert Entities	693
	Tools > Point Cloud Tools > Separate Point Cloud.....	696
7.5.5.	Tools > Texture Tools.....	699
	Tools > Texture Tools > Add Texture	700
	Tools > Texture Tools > Edit Texture	705
	Tools > Texture Tools > Remove Texture	709
	Tools > Texture Tools > Export Texture.....	711
	Tools > Texture Tools > Convert Texture To Color.....	712
	Tools > Texture Tools > Convert Color To Texture.....	713
	Tools > Texture Tools > Copy Texture/Vertex Color	714

	Tools > Texture Tools > Match Color	715
7.5.6.	Tools > Sketch Entities	717
	Tools > Sketch Entities > Auto Sketch	719
	Tools > Sketch Entities > Line	722
	Tools > Sketch Entities > Rectangle	726
	Tools > Sketch Entities > Parallelogram.....	727
	Tools > Sketch Entities > Polygon.....	728
	Tools > Sketch Entities > Slot.....	730
	Tools > Sketch Entities > Circle	731
	Tools > Sketch Entities > Perimeter Circle.....	734
	Tools > Sketch Entities > 3 Tangent Circle	736
	Tools > Sketch Entities > Centerpoint Arc	737
	Tools > Sketch Entities > Tangent Arc	740
	Tools > Sketch Entities > 3 Points Arc.....	741
	Tools > Sketch Entities > 3 Tangent Arc.....	743
	Tools > Sketch Entities > Ellipse	745
	Tools > Sketch Entities > Partial Ellipse.....	746
	Tools > Sketch Entities > Parabola.....	748
	Tools > Sketch Entities > Spline.....	750
	Tools > Sketch Entities > Centerline.....	755
	Tools > Sketch Entities > Point.....	756
	Tools > Sketch Entities > Text	757
	Tools > Sketch Entities > By Fitting Polyline	759
	Tools > Sketch Entities > By Import	761
7.5.7.	Tools > Sketch Tools.....	764
	Tools > Sketch Tools > Fillet	765
	Tools > Sketch Tools > Chamfer.....	767
	Tools > Sketch Tools > Offset.....	769
	Tools > Sketch Tools > Convert Entities	770
	Tools > Sketch Tools > Silhouette	772
	Tools > Sketch Tools > Trim	774
	Tools > Sketch Tools > Extend.....	776
	Tools > Sketch Tools > Split.....	778
	Tools > Sketch Tools > Mirror.....	779
	Tools > Sketch Tools > Resize	780
	Tools > Sketch Tools > Merge	782
	Tools > Sketch Tools > Linear Pattern.....	783
	Tools > Sketch Tools > Circular Pattern	786
	Tools > Sketch Tools > Fit To Spline.....	789
	Tools > Sketch Tools > Text To Splines.....	791
	Tools > Sketch Tools > Insert Polyline.....	792
7.5.8.	Tools > Sketch Options	793
	Tools > Sketch Options > Snap On Entities	794
	Tools > Sketch Options > Snap On Polylines	796
	Tools > Sketch Options > Smartsnap	798
7.5.9.	Tools > Smart Dimension.....	799
7.5.10.	Tools > 3D Sketch Entities.....	802
	Tools > 3D Sketch Entities > Spline	803
	Tools > 3D Sketch Entities > Pencil.....	807
	Tools > 3D Sketch Entities > Section	809
	Tools > 3D Sketch Entities > Intersection.....	814
	Tools > 3D Sketch Entities > Boundary	815
	Tools > 3D Sketch Entities > Surface UV Curves.....	817
	Tools > 3D Sketch Entities > Fit From Points.....	819
	Tools > 3D Sketch Entities > Trace Feature Line	823

	Tools > 3D Sketch Entities > Extract Feature Curves.....	825
	Tools > 3D Sketch Entities > Create Curve Network	828
7.5.11.	Tools > 3D Sketch Tools	832
	Tools > 3D Sketch Tools > Offset	833
	Tools > 3D Sketch Tools > Project	835
	Tools > 3D Sketch Tools > Mirror.....	837
	Tools > 3D Sketch Tools > Convert Entities	838
	Tools > 3D Sketch Tools > Smooth	841
	Tools > 3D Sketch Tools > Trim	842
	Tools > 3D Sketch Tools > Extend	843
	Tools > 3D Sketch Tools > Split	845
	Tools > 3D Sketch Tools > Match.....	847
	Tools > 3D Sketch Tools > Merge	850
	Tools > 3D Sketch Tools > Insert Node.....	851
	Tools > 3D Sketch Tools > Remove Node.....	853
	Tools > 3D Sketch Tools > Remove Constraint.....	854
7.5.12.	Tools > Body Tools	855
	Tools > Body Tools > Find Defect	855
7.5.13.	Tools > Align	856
	Tools > Align > Wizard	857
	Tools > Align > Interactive Alignment	859
	Tools > Align > Quick Fit.....	864
	Tools > Align > Best Fit.....	865
	Tools > Align > Datum Match.....	868
7.6.	Measure.....	871
7.6.1.	Measure > Distance.....	871
7.6.2.	Measure > Angle.....	874
7.6.3.	Measure > Radius.....	875
7.6.4.	Measure > Section.....	876
7.6.5.	Measure > Mesh Deviations.....	880
7.7.	View	884
7.7.1.	View > Visibility.....	885
	View > Visibility > Show Selected Only.....	886
	View > Visibility > Show All	887
	View > Visibility > Hide All	888
	View > Visibility > Meshes	889
	View > Visibility > Regions.....	890
	View > Visibility > Point Clouds	891
	View > Visibility > Surface Bodies	892
	View > Visibility > Solid Bodies	893
	View > Visibility > Sketches.....	894
	View > Visibility > 3D Sketches	895
	View > Visibility > Ref. Points.....	896
	View > Visibility > Ref. Vector.....	897
	View > Visibility > Ref. Planes	898
	View > Visibility > Ref. Polylines.....	898
	View > Visibility > Ref. Coordinates.....	899
	View > Visibility > Measurements.....	900
	View > Visibility > Smart Visibility	901
7.7.2.	View > Zoom	904
	View > Zoom > Area	904
	View > Zoom > Previous Area	906
	View > Zoom > Fit	907

View > Zoom > PIP	907
7.7.3. View > Mesh Display Mode	909
7.7.4. View > Body Display Mode.....	911
7.7.5. View > Viewpoint.....	912
7.7.6. View > Viewport	915
7.7.7. View > Light Properties	918
7.7.8. View > View Clip	920
7.7.9. View > Walk Through.....	923
7.8. Add-Ins.....	924
7.8.1. Add-Ins > Import & Split.....	925
7.8.2. Add-Ins > IGES Point Importer	926
7.8.3. Add-Ins > Register Target.....	926
7.8.4. Add-Ins > Delete Poly-Faces	928
7.8.5. Add-Ins > Trim Mesh.....	930
7.8.6. Add-Ins > Flatten / Deform.....	933
7.8.7. Add-Ins > Target AutoMatch BETA.....	936
7.9. Help	940
8. GLOSSARY	943
9. INDEX	953

1. Introduction

Geomagic® Design™ X

The content of this online help is furnished for informational use only, is subject to change without notice, and should not be construed as a commitment by 3D Systems Corporation.

3D Systems, the 3D Systems logo, Geomagic, the Geomagic logo, Geomagic Design X and the Geomagic Design X logo referred to herein are either the trademarks or the registered trademarks of 3D Systems Corporation. All other trademarks within this online help are the property of their respective owners and are used for identification purposes only. Other than to identify this Software and publication, individuals or organizations purchasing the software are not entitled to use 3D Systems' trademarks without 3D Systems' prior written consent.

Correspondence regarding this publication should be directed to:

3D Systems Korea, Inc. www.rapidform.com
128 Hakdong-ro Gangnam-gu, Seoul 135-822, KOREA
info@rapidform.com Phone: +82.2.6262.9900 Fax: +82.2.6262.9999

Copyright © 3D Systems, Inc. All rights reserved.



To see the main contents of the help guide, press the F1 key while using a command or while in any mode. The **F1** key by default will open the contents help guide. This will display the help document for the particular command or mode that is being used.

Note:

You must be in an active command or in an activate mode to use the F1 function. Otherwise, either the intro document page will be displayed if the F1 function is used for the first time, or the last help page viewed.

1.1. Welcome to Geomagic Design X



Parametric CAD Models from 3D Scan Data

Thank you for choosing Geomagic Design X, the most comprehensive 3D Scan-To-CAD Software Solution. 3D Systems, Inc. is the leader in providing technologies that make 3D scanning a powerful tool for a variety of applications including manufacturing, R&D, quality inspection, medical research, civil engineering and more, and is now presenting the future of 3D scanning software technology with its next generation 3D scan data processing platform, Geomagic. Geomagic Design X makes the process of creating parametric CAD models from real world parts faster and easier by utilizing a design process and user interface that are instantly familiar to CAD users.

Parametric CAD Models from 3D Scan Data

Geomagic Design X allows engineers to capture the design intent and design parameters of real world parts that may have lost their defining features during the manufacturing process or may not have ever had a CAD model. 3D scanning technology and Geomagic Design X provide manufacturers with the freedom and flexibility to extract the design parameters from any real world part, including prismatic features and freeform surfaces. CAD models created in Geomagic Design X are fully parametric, so designers and engineers are able to modify the design parameters of the real world part to perfect the model for mass production within Geomagic Design X in a downstream CAD system.

- Intelligent tools for extracting design parameters from 3D scan data – *Redesign Assistant*
- Redesign within user-defined deviation tolerances – *Accuracy Analyzer™*
- Intelligently identify and align 3D scan data to an ideal design coordinate system – *Align Wizard™*
- Reduce design time by using 3D scan data as a design foundation
- Use outputted data, with modeling history intact, in CAD, CAM and CAE
- Modeling history and parameter management
- Mesh, freeform surface and parametric solid hybrid modeling functions
- Standard solid & surface modeling workflows such as extrude, round, revolve, sweep and loft
- Update existing CAD models to reflect changes in the as-build part - *CAD-to-Scan Refit*
- Seamless interoperability with full modeling history to other CAD applications

Immediately Realize the Benefits of 3D Scanning

The design tools used to create models in Geomagic Design X are immediately recognizable to those already familiar with CAD applications. Engineers capable of designing with SolidWorks, CATIA, Pro/ENGINEER or Siemens NX can immediately begin modeling in Geomagic Design X. The design process in Geomagic Design X utilizes common CAD modeling features and processes such as extrude, round, revolve, sweep and loft. These features serve to make both Geomagic Design X and 3D scanning technology readily accessible to engineers, while the ease of use of the product allows 3D scan-based to be institutionalized within the manufacturing process to increase the overall quality of products.

- Intelligent Design process which eliminates the need for complete scans of parts
- Create high quality CAD models from imperfect scan data
- Save processing time by eliminating the need for polygon mesh cleanup and surfacing
- Models are instantly editable in both Geomagic Design X and major CAD systems

A Comprehensive Scan-To-CAD Software Solution

Geomagic Design X is an entirely new software solution that provides a new approach, yet with a familiar process, to build a parametric CAD models from 3D scan data. Geomagic Design X creates highly usable models for a variety of applications that include rapid prototyping, CNC machining, CAE and export into CAD applications for further modifications. The values of these applications are now available in a simple software application that is readily accessible to a wider group of people by utilizing the common design tools of most CAD applications.

- End-to-end functional coverage – *from mesh operations to CAD modeling*
- Supports every possible reverse modeling workflow – *Reflect design intent (design parameter extraction) or Generate and exact duplicate (surface fitting)*
- Instant mesh optimization for direct use in RP, CAM, CAE and visualization
- Highly sophisticated yet familiar modeling capabilities utilizing universal solid and surface modeling features
- One-button quick B-rep surfacing or design analysis & verification applications
- Automatic re-meshing for generating CAE functional models – *Scan-to-CAE*

1.2. Key Themes of Geomagic Design X

DO MORE WITH 3D SCAN DATA



*Create parametric CAD models from 3D scan data
Send full history-based models to SolidWorks and other CAD System
Capture design intent and define design parameters
Design parts faster using familiar CAD modeling concepts*

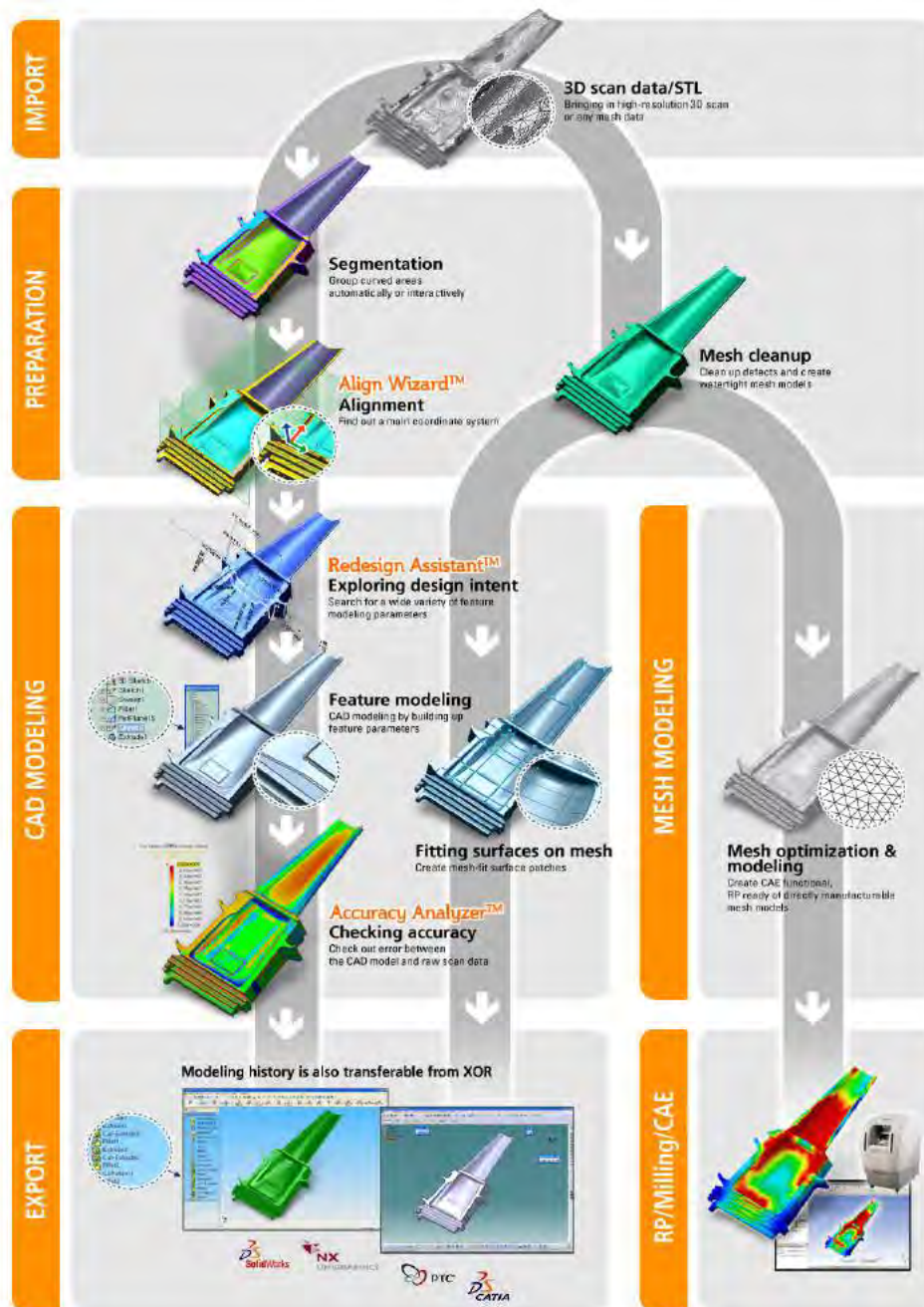
An Innovative Design Process

Geomagic Design X is absolutely unique in the market. Geomagic Design X is powerful software that provides an innovative and unique philosophy for building CAD models from 3D scan data. It supports every possible reverse engineering workflow and is suited to both polygon-based and CAD-based applications. Geomagic Design X provides the best way to utilize 3D scanning technology as a “mission-critical” solution for innovation in the product design and manufacturing process.

The design process and user interface in Geomagic Design X have been developed to be instantly familiar to users of popular CAD applications, leading to incredible time savings over traditional reverse engineering software while simultaneously creating output in the form of parametric CAD solid models. Time savings can be realized through:

- A reduced learning curve through the use of a familiar CAD interface and design process
- A design process that removes the time consuming requirement of cleaning the scan data of a part
- A single software solution that fulfills the scan data processing, mesh healing and CAD modeling needs
- A built in, real time deviation analysis tool

The interface and design process of Geomagic Design X were developed to be similar to popular CAD applications, so users can utilize their existing CAD modeling skills to immediately begin designing in Geomagic Design X. Internal benchmarking tests have shown a time savings of up to 80% when using Geomagic Design X over traditional surface fitting methods in great part because the design process removes the need to clean the scan data of a part. A complete scan of a part is unnecessary, provided there is enough data to recognize design parameters.



Massive Point Cloud Handling

Geomagic Design X's point cloud data processing has been innovated to handle massive point clouds. The data structures and some of the functions are optimized and enhanced for working with hundreds of millions of point clouds.

Unlimited Point Cloud Capacity

When large point clouds are imported that exceed the physical memory limitation of a computer, the application will automatically use a cache folder (hard disk drive) and load data onto it. While working with a point cloud, the application intelligently divides the point cloud and performs functions within the memory capacity. Therefore, almost unlimited point cloud

data sizes can be imported based on the storage size of the hard disk drive.

Real-Time Visualization of Massive Point Clouds

Geomagic Design X's auto frame rate function selectively displays necessary points with adaptable density to the 3D view of current your viewpoint. By utilizing a quick rendering algorithm of the point cloud, you can see and operate a complex and large point cloud in an optimized environment. Geomagic Design X automatically changes the display frame rate according to the data size of the point clouds to create the best working environment without cutting off display functionality.

To maximize the visualization of point clouds, Geomagic Design X supports three shading methods which help to analyze and visualize complex point clouds.

Depth – The closer points are displayed in black and the farther points are display in gray in relation from the view direction. If you click the Set button, you can define the depth range by using the slider bar. If points are out of the depth range, they will disappear from the Model View. This can be used to focus on specific areas by removing outer areas from the depth range.



None



Depth

X Ray – This mode shows specifically targeted areas by making areas closer and farther from the targeted area transparent. To define the transparent area, click the **Set** button. The **Near** and **Far** areas will be transparent and the **Middle** area will be clearly displayed.

For example, if a scanner scans the inside of a building, it can be difficult to see inside because of the outer walls. You can use the X-Ray function to make the outer walls transparent, thus letting you see inside clearly.



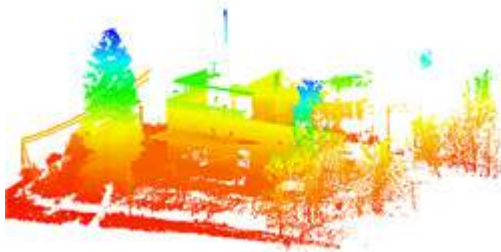
X-Ray Default



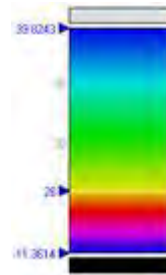
After Adjusting X-Ray Depth

This shading method makes points transparent in a certain range. When trying to select points, points nearer to the viewing direction will be selected first even though they are invisible. Therefore, this method should be used for data viewing purposes.

Height – The color of displayed points will change based on its height from the Z direction. Using this shading method, you can easily distinguish ground and other features. If you click the Set button, you can modify colors as well as height.



Height



Color Bar

For easier inspection of scan data from long range scans such as construction fields, buildings, factories, etc, Geomagic Design X includes a walk-through mode. In the previous version, the camera (viewpoint) was always fixed and model was rotated, zoomed and panned in the Model View. However this new walk-through mode enables the camera to travel through and position the viewpoint inside of the model.

Fully Editable Point Clouds

Geomagic Design X supports not only the visualization of point clouds like other point cloud software but also includes editing and generating CAD tools for massive point clouds.

Selecting points

Selecting only desired areas with as few steps as possible is important in massive point cloud handling. Geomagic Design X's selection tools, Extend To Similar and Extend To Sudden Change, which expands similar features from the seed point, enables the quick and easy selection of areas from complex point clouds.

Editing point cloud

The functions under Point Cloud Tools are fully optimized for handling large point clouds. When generating a mesh from a point cloud, the Mesh Construction method in Triangulation and the Mesh Buildup Wizard has no data size limitation and the 2D/3D Triangulation methods enable the reduction of the data input size with the Sampling option.

For the convenience of working with large point clouds in the application, a mesh can be generated with chosen selections from a point cloud. Copy and pasting the point cloud is possible for defining the working area and for dividing parts.

Modeling from point cloud

2D sketch entities can be generated from section polylines on point clouds or projected points within a defined range. Pipe modeling on the point cloud can also be done by directly picking seed points on the pipe feature. This Pipe Wizard is an easy and powerful function for fast modeling from complex point clouds with many pipe features.

New File Format for Massive Point Clouds

As well as new point cloud process methods, Geomagic Design X has added a new file format, XPC, and updated the existing format, XRL. These formats are not only optimized for minimizing file size, but also to include optimized data structures for massive point clouds.

For the publication of point clouds to a website, .icf and .rps files can be reduced in file size by sampling the point cloud.

1.3. Features of Geomagic Design X

New Key Features and Enhancements



Parametric CAD Models from 3D Scan Data

Geomagic Design X is an innovative new software solution that allows engineers to create fully parametric CAD solids and freeform surfaces from 3D scan data and polygon mesh data with a best-in-class user interface. In addition to its ability to create parametric CAD solids, Geomagic Design X differentiates itself from existing reverse engineering software by offering a design process that utilizes the skill sets that engineers already possess. The end result is a software solution that requires minimal training, reducing the time necessary to process 3D scan data. Users are provided with high quality results that are unavailable through other current reverse engineering options. Geomagic Design X enhances the functionality of both 3D scanning technology and existing CAD applications by sharing common technologies and processes. Geomagic Design X features powerful tools such as:

- Redesign Assistant
- Accuracy Analyzer
- True Hybrid Modeler - Solid, Surface & Mesh
- Align Wizard
- Best-in-class Mesh Operations
- Sophisticated Curve/Sketch Tools
- Scan-to-CAD Synchronization
- Quick Mesh-to-Surface
- Seamless Data Transfer
- Mesh Cleanup

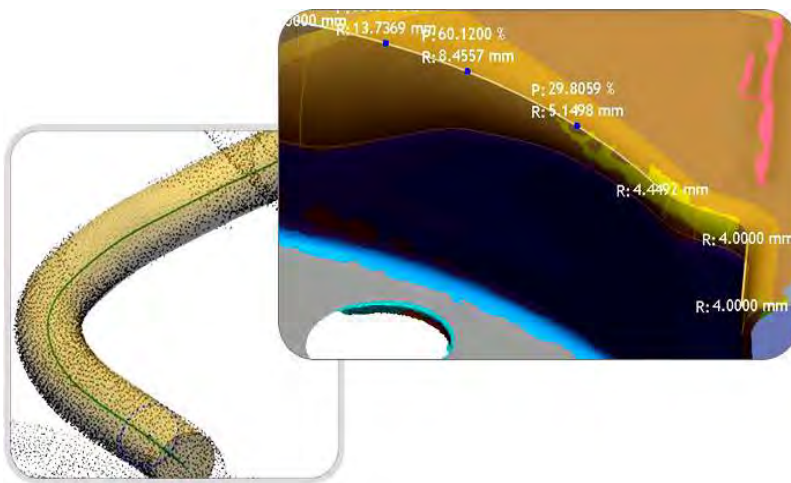
Redesign Assistant

The Redesign Assistant intelligently extracts the design intentions and design parameters throughout the process of generating a CAD model from 3D Scan Data through the following features.

- Unprecedented tools for extracting design parameters from 3D scan data
- Utilizing meshes as redesign parameters
- Automatic mesh region segmentation
- Automatic extraction of design feature parameters from meshes

Filet radius and center, sketch plane & profile (automatic sketching), sweep path curve, extrusion axis, mirroring plane, revolving center axis, pipe center axis, drafting angle, 3D section curve for lofting, feature curve, offset/thickening distance, cylinder/cone axis, slot axis, pattern axis and direction, silhouette-curve, parting line, bead line, etc

- Creating sketch profiles from mesh models
- Automatic extraction of 3D design features from mesh
- Intelligent real-time 2D/3D geometry recognition and snapping

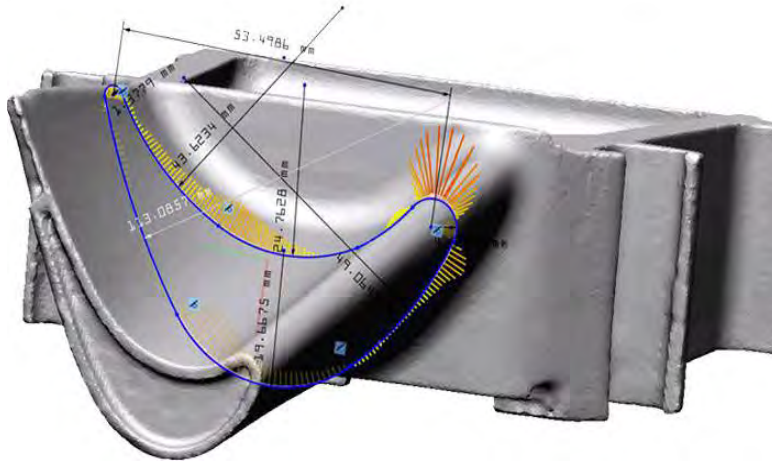


Redesign Assistant™

Accuracy Analyzer™

The Accuracy Analyzer™ tool provides users with real time deviation analysis based on user defined tolerances to ensure that a model is within specified allowable tolerances.

- Polygonal mesh deviation (comparison with original scan data and previous step model)
- Sketch/curve/surface/solid deviation (comparison with original scan data)
- Geometric property analysis (curvature, continuity, quality, normal vector, etc)
- Redesign within user-defined allowable tolerances
- Automatic and real-time error visualization
- Diverse object sensitive analysis tools



Accuracy Analyzer

True Hybrid Modeler

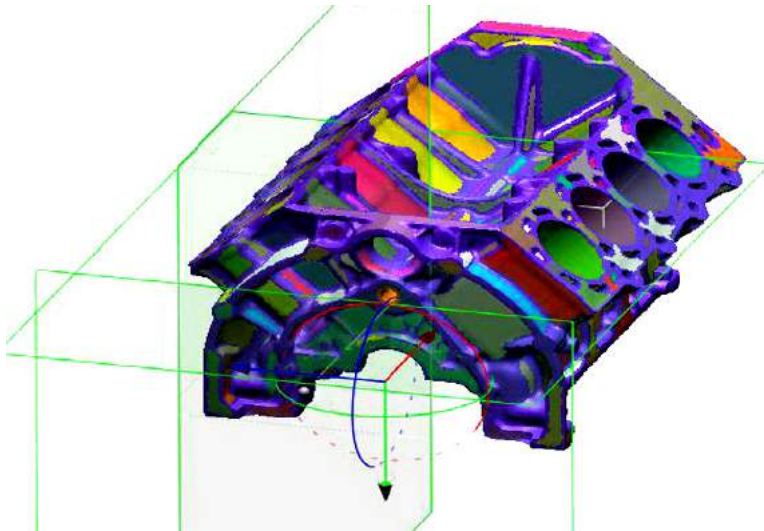
Geomagic Design X is a truly comprehensive 3D scan data processing application that offers parametric solid modeling capabilities, NURBS surfacing capabilities, and a hybrid modeling process that utilizes both capabilities for the creation of parametric CAD models that contain freeform features.

- Highly sophisticated yet familiar by utilizing widely accepted solid and surface modeling features
- Solid Features: extrude, sweep, revolve, pipe, thicken, draft, round with various radii, chamfer, hollow (shell), linear/circular/curve pattern, Boolean modeling
- Surface Features: drape, blend, extrude, revolve, sweep, loft, ruled, offset, mirror, fill face, extend, trim/untrim, match, heal
- Modeling history management (feature rebuild, reroute & reorder)
- Parametric feature management

Align Wizard™

Geomagic Design X includes the Align Wizard™, a proprietary tool that intelligently identifies the coordinate system in a file that the original designer had used. The user can either choose the coordinate system recommended by the Align Wizard™, or manually determine the coordinate system that is believed to be the most appropriate for the part by using an intuitive coordinate system setup tool.

- Wizard for aligning 3D scan data to an ideal design coordinate system
- Highly interactive toolset for coordinate alignment
- Quick fit, best fit, 3-2-1, Datum, By Ref. coordinates, etc



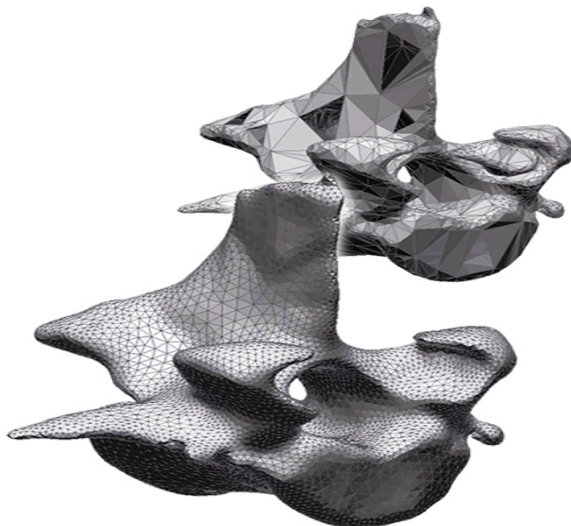
Align Wizard™

Best-in-class Mesh Operations

Geomagic Design X produces Class-A meshes in a single step. The mesh models generated by Geomagic Design X can be directly used as input data for target CAE and CAM software for reverse engineering or manufacturing. Geomagic Design X features a powerful mesh optimization tool that generates a mesh from 3D scan data that meets the specific requirements for CAE.

- Instant mesh optimization for direct use in RP, CAM & CAE
- Detail resolution controls (decimate & subdivide)
- Mesh smoothness controls (global & subdivide)
- Automatic re-meshing for CAE functional models
- Comprehensive interactive mesh region editing tools
- Advanced mesh modeling & optimization

Global remesh, remove, de-feature, hole filing, fix boundary, smooth boundary, fit boundary, fit region to analytic shape, split & trim, divide, thicken, offset are some of advanced mesh operations available.



Sophisticated Curve and Sketch Tools

Geomagic Design X provides users with highly sophisticated curve/sketch tools so that users can extract sketch profiles and feature curves from mesh automatically for comprehensive reverse modeling.

- Automatic extraction of sketch profiles and feature curves from mesh
- Automatic dimensioning & constraining
- Variety of 2D drafting tools
- Comprehensive 3D curve design tools
- Intelligent real-time geometry recognition
- Curvature-based curve network design

Scan-To-CAD Synchronization

Geomagic Design X offers the best solutions to synchronize with CAD applications for parametric CAD modeling from 3D scan data.

- Update of the original CAD model to represent the changes in the as-built part
- The import of CAD data in a variety of CAD file formats
- Fast and automatic CAD & mesh model coordinate alignment
- Refitting of CAD models to mesh models

Quick Mesh-To-Surface

- Automatic shrink wrapped surface model generation
- Optimized representations of original mesh data with negligible deviation errors
- Interface mesh-fit surfacing (region & boundary fit)

Seamless Data Transfer

- Transfer output model with full history to CAD system
- Export models in a variety of standard file formats

Mesh Cleanup

- Automatic mesh healing & cleaning
- Professional yet easy to use mesh cleaning & editing tools

2. Getting Started

This section lists the system requirements to run the application and the licensing information software.

2.1. System Requirement

	32-bit Edition		64-bit Edition	
OS	- Windows Vista (32-bit or 64-bit SP1 or above) - Windows 7 (32-bit or 64-bit) - Windows 8 (32-bit or 64-bit)		- Windows Vista (64-bit SP1 or above) - Windows 7 (64-bit) - Windows 8 (64-bit)	
CPU	- Intel® and AMD® processors. - Recommended: Multiple core processors. Hyper-threading and clock speeds above 3ghz can be beneficial but should be paired with a good balance of cores.			
RAM	- Minimum: 1 GB - Recommended: 2 GB - Virtual Memory: Vista, Win7: recommended to use the default option; "Automatically manage paging file size for all drives"		- Minimum: 2 GB - Recommended: 8-16 GB (Based on version of Windows used) - Virtual Memory: Vista, Win7: recommended to use the default option; "Automatically manage paging file size for all drives"	
Hard Disk	- 30 GB or more recommended - Temporary file cache requires about 3 GB free disk space for every 100 million points - Recommended: a SSD with more than 256GB of space, or a 10,000RPM HDD - Recommended: using multiple hard drives and assigning one as a "cache" drive for Geomagic temporary storage and paging files and assigning the other for the operating system			
Display	- OpenGL 1.2 or above, 32 bit true color required - Recommended 24-inch wide screen monitor, 1920X1280 resolution			
GPU	- Recommended: NVidia GTX series with 1GB of RAM or more			
Certified Graphics Card	Intel	Intel G45 Intel G645GM Intel G965 Intel HD 2000/3000	Intel	Intel G45 Intel G645GM Intel G965 Intel HD 2000/3000
	AMD	Rage XL Radeon 7500 Radeon 9600 Radeon 9600 XT Radeon X600 Radeon X600SE Radeon X700 Radeon X1650 Radeon X1600 Radeon X1800 GTO Radeon HD 2600 XT Radeon HD 3670M Radeon HD 3870	AMD	Rage XL Radeon X600SE Radeon 7500 Radeon 9600 Radeon X600 Radeon X1600 Radeon X1800 GTO Radeon HD 2600 XT Radeon HD 3870 Radeon HD 4770 Radeon HD 4800 1G Radeon HD 5700 Radeon HG 5770

	Radeon HD 4770 Radeon HD 4800 1G Radeon HD 5700 Radeon HG 5770 Radeon HD 5800 Radeon HD 6470M Radeon HD 6850 Radeon HD 6870 Radeon HD 6950 Radeon HD 6970 Radeon HD 7850 Radeon HD 7770		Radeon HD 5800 Radeon HD 6470M Radeon HD 6850 Radeon HD 6870 Radeon HD 6950 Radeon HD 6970 Radeon HD 7850 Radeon HD 7770
AMD	FireGL Z1 FireGL V3100 FireGL V3600 FireGL V5600 FireGL V7300 FireMV V2250 FirePro V2260 FirePro V2450 FirePro V3750 FirePro V3800 FirePro V4800	AMD	FireGL V3100 FireGL V3600 FireGL V5600 FireGL V7300 FireMV V2250 FirePro V2260 FirePro V2450 FirePro V3750 FirePro V3800 FirePro V4800
NVIDIA	GeForce2 MX 100/200 GeForce4 Ti 4400 GeForce FX 5200 GeForce FX 5500 GeForce FX 5600 GeForce FX 5600XT GeForce FX 5700 GeForce FX Go 5200 GeForce FX Go 5700 GeForce Go 5200 GeForce Go 7400 GeForce Go 7600 GeForce 6200 GeForce 6600 GeForce 6800 GeForce 7600GS GeForce 7600GT GeForce 7800GTX GeForce 7900GS GeForce 640M GeForce 8300GS GeForce 8400M GeForce 8400GS GeForce 8500 GeForce 8500GT GeForce 8500GTS GeForce 8600 GeForce 8600GT GeForce 8600GS GeForce 8600GTS GeForce 8600M GeForce 8800GT GeForce 8800GTS GeForce 9200M	NVIDIA	GeForce2 MX 100/200 GeForce FX Go 5200 GeForce Go 5200 GeForce Go 7400 GeForce Go 7600 GeForce 6200 GeForce 6600 GeForce 6800 GeForce 7600GS GeForce 7600GT GeForce 7900GS GeForce 8300GS GeForce 8400M GeForce 8400GS GeForce 8500 GeForce 8500GT GeForce 8600 GeForce 8600GS GeForce 8600GT GeForce 8600GTS GeForce 640M GeForce 8600M GeForce 8800GT GeForce 8800GTS GeForce 9200M GeForce 9300M GeForce 9300GE GeForce 9500GT GeForce 9600GT GeForce 9800GT GeForce 9800GTX GeForce G105M GeForce GT 220 GeForce GT 630

		<p>GeForce 9300M GeForce 9300GE GeForce 9500GT GeForce 9600GT GeForce 9800GT GeForce 9800GTX GeForce G105M GeForce GT 220 GeForce GT 630 GeForce GTS 250 GeForce GTX 260 GeForce GTX 285 GeForce GTS 335M GeForce GT 310M GeForce GT 360M GeForce GT 425M GeForce GTX 485M GeForce GT 520M GeForce GT 525M GeForce GT 540M GeForce GT 550M GeForce GT 555M GeForce GT 8600M GeForce GS 8600M GeForce GTX 550TI GeForce GTX 560TI GeForce GTX 465 GeForce GTX 560 GeForce GTX 570 GeForce GTX 580 GeForce GTX 670 GeForce GTX 680</p>		<p>GeForce GTS 250 GeForce GTX 260 GeForce GTX 285 GeForce GT 310M GeForce GTS 335M GeForce GT 360M GeForce GT 425M GeForce GTX 465 GeForce GTX 485M GeForce GT 520M GeForce GT 540M GeForce GT 550M GeForce GT 555M GeForce GT 8600M GeForce GS 8600M GeForce GTX 550TI GeForce GTX 560TI GeForce GTX 560 GeForce GTX 570 GeForce GTX 580 GeForce GTX 670 GeForce GTX 680</p>
	NVIDIA	<p>Quadro NVS 120M Quadro NVS 135M Quadro NVS 140M Quadro FX 350M Quadro FX 360M Quadro FX 500 Quadro FX 540 Quadro FX 570M Quadro FX 580 Quadro FX 880M Quadro FX Go 1400 Quadro FX 1400 Quadro FX 1500 Quadro FX 3500 Quadro FX 4500 Quadro FX 5200 Quadro FX 5700 Quadro 600 Quadro 1500 Quadro 1800 Quadro 2000 Quadro 2000D Quadro 3000 Quadro 3500 Quadro 3800 Quadro 4000 Quadro 4500</p>	NVIDIA	<p>Quadro FX 350M Quadro FX 360M Quadro FX 570M Quadro FX 580 Quadro FX 880M Quadro FX Go 1400 Quadro FX 1400 Quadro FX 1500 Quadro FX 3500 Quadro FX 4500 Quadro FX 5200 Quadro FX 5700 Quadro 600 Quadro 1500 Quadro 1800 Quadro 2000 Quadro 2000D Quadro 3500 Quadro 3800 Quadro 4000 Quadro 4500 Quadro 3000M</p>

	Quadro 3000M		
	- Not Recommended ATI Mobility Radeon 7000 IGP DDR ATI Radeon X800 GTO ATI Radeon X1600		- Not Recommended ATI Mobility Radeon 7000 IGP DDR
	<p>- If you use the AMD Radeon series, the panning or rotation in the application might not operate correctly. To solve this problem uncheck the Use Custom Settings option under 3D > Standard Settings in the Catalyst Control Center dialog box and move the slider to the left.</p> <p>It is recommended to set the anti-aliasing value lower than 8. Otherwise, detection of 3D edges may not work properly or there may be problems with panning or rotating within the application. If anti-aliasing is disabled, the edges and boundaries of shaded objects can be jagged if displayed at angles close to 0 or 90 degrees.</p>		
Other	<ul style="list-style-type: none"> - DVD drive - 3 button mouse with scroll wheel - Sound card - Keyboard - Network card - Internet Explorer version 7.x or higher - Adobe Acrobat version 5.x or higher - Third Party Application: Microsoft.NET 4.0 		

2.2. Installation

1. Insert the installation DVD into the DVD-ROM drive.
2. The installation program should appear automatically. If the autorun installation does not start automatically, browse the DVD-ROM directory and then double-click on the autorun.exe file to start the installation program.
3. Follow the instructions provided by the installation program.
4. After installation is complete, run the application.
5. After activating your license, you will see a dialog box which enables you to set the initial settings for Geomagic Design X.

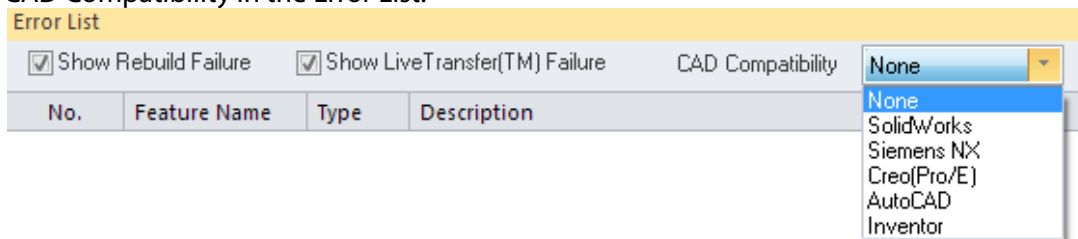


Enable Multi-thread – When set to **True**, multi-threading is utilized to speed up the calculation time of some functions. This is highly recommended. On a single processor, multithreading generally occurs by the processor switching between different process threads one at a time. On a multiprocessor or multi-core system, threads will process simultaneously, with each processor or core running a particular thread. Depending on how many cores the processor has, the same number of progress bars will be displayed when multithread is enabled and utilized.

User Name – The PC's name

Update Product Automatically – If it is set to **True**, whenever you launch the program, it will check if there are any updates.

Preferred CAD – Choose one of your CAD systems: None, SolidWorks, Siemens NX, PTC Creo(Pro/E), Autodesk AutoCAD or Autodesk Inventor. It is linked to View Manipulation Style and CAD Compatibility in the Error List.



View Manipulation Style – You can select one of the view manipulation style (zooming, panning, rotating); Rapidform, SolidWorks, Siemens NX, Creo(Pro/E), CATIA, PolyWorks, or Geomagic.

Preferred Scanner File Format –Select a scanner format that you frequently use, so that when you import a scanner file, you will see only the chosen format.

Preferred Scanner Direct Control –You can set your favorite scanners as **True** so that when you run **Insert > Scanner Direct Control > LiveScan(TM)**, you will see only your favorite scanners in the **Device** pull down menu. To apply this change, close and rerun the application.

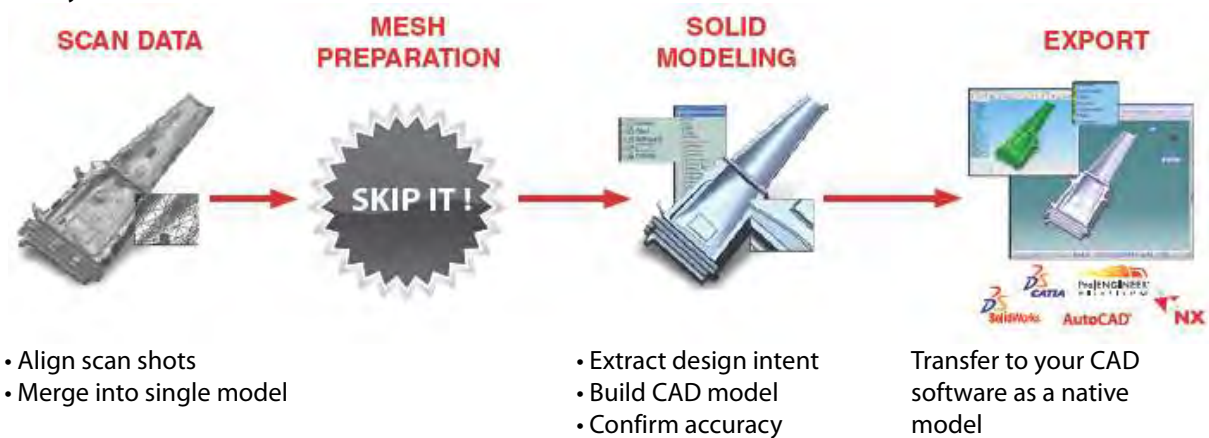
Unit – Select default units which will be used in the application.

Use Configurations Of Previous Version – When you install a new version of the application, you can preserve customized settings.

2.3. What Can Geomagic Design X Do?

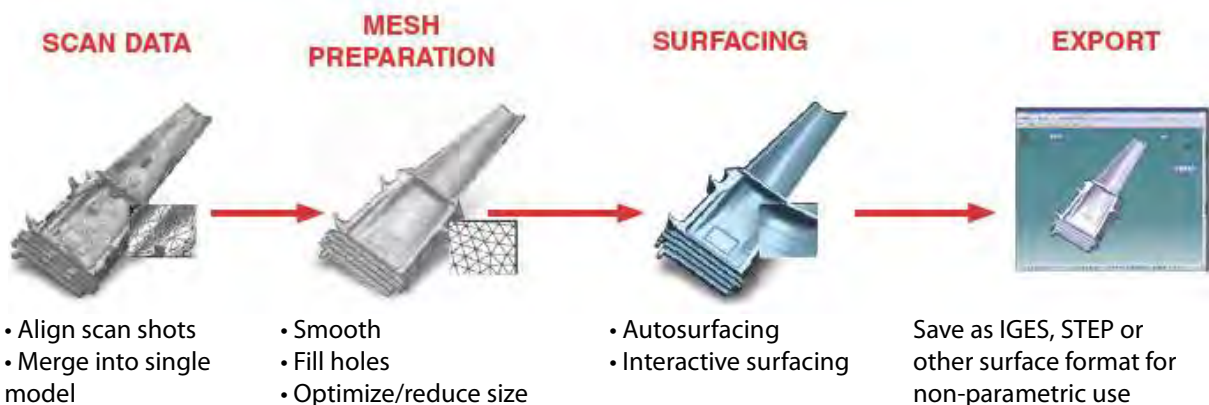
Scan to CAD

To make a CAD model suitable for manufacturing, Geomagic Design X employs a straightforward workflow that parallels today's well-known solid modeling process. Instead of modeling from scratch, Geomagic Design X's feature wizards and other automatic tools help you build the features directly from 3D scan data.



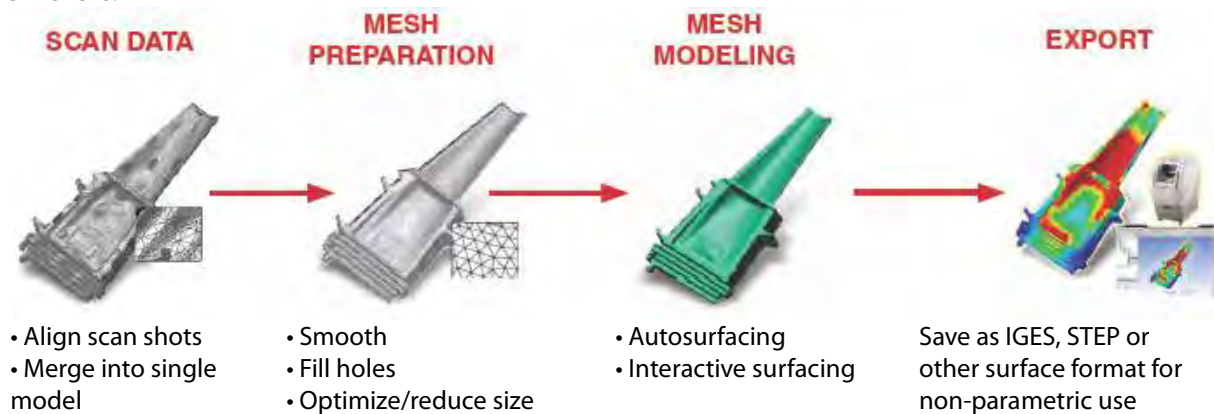
Scan to Surface

Geomagic Design X also supports NURBS surface fitting, for when you need a verbatim copy of what you've scanned, and don't need to edit it later on. Geomagic's powerful automatic surfacing technology can do all the work, or the surface network can be created manually.



Scan to Mesh

For archiving, animation, measurement, 3D printing and other polygon-based applications, Geomagic Design X includes a full suite of mesh processing tools. With the Mesh Buildup Wizard™, you can step through the entire scan alignment, merge and mesh optimization process with just a few clicks.



2.4. Basic Entity Terminology

The application handles many different types of entities ranging from scan data to CAD data. Entities can be an end result of modeling, or the by-products of a final model. This chapter explains what kinds of entities are used in the application, and its terminology definitions.

Point Cloud

A set of 3D points represents a real part or environment. Each point represents a digitized point on the object or in the environment, and collectively describes its shape and measurements. It usually comes from a 3D scanner or a coordinate measuring machine (CMM). A point cloud can be built up as a mesh in software by generating triangles between points, a process known as triangulation. Point clouds are used for the visualization of real parts or environments in reverse engineering to make a CAD model.

A point cloud may or may not have normal information, depending on the scanner and scanning type. Even if a point cloud does not have normal information, it can be generated within the application after importing the data.

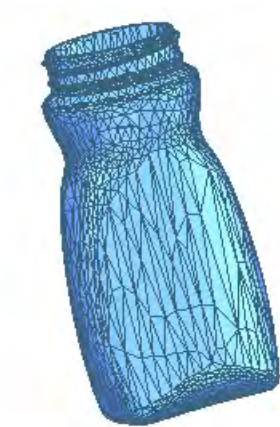


Point cloud data from a real bottle

Mesh

A mesh is a polygonal model that represents a physical object. A mesh consists of many triangles which connected and can be generated from a point cloud or CAD data.

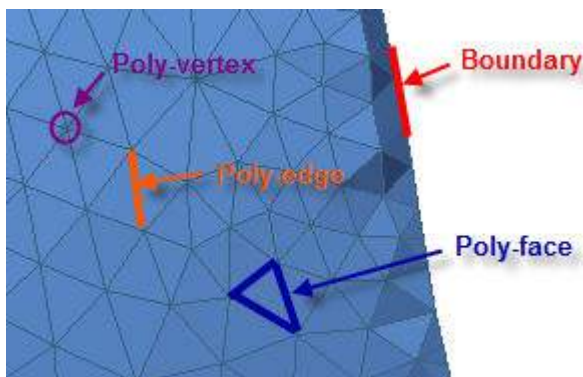
A scanned part usually consists of several scans from different directions, which necessitates alignment of the scans by overlapping and aligning common areas in software. Once aligned, several meshes can be converted into a single mesh by merging. When merging multiple scans, overlapping areas will be removed.



Mesh from a point cloud of a real bottle

In the application, the following terminologies are used to separate similar entities between a mesh and a body.

- Poly-Face: Triangles enveloped by three poly-edges.
- Poly-Edge: Connecting edge between two poly-vertices.
- Poly-Vertex: A point which represents a position in 3D space.
- Boundary: Any opening in the mesh, where there are no triangles.



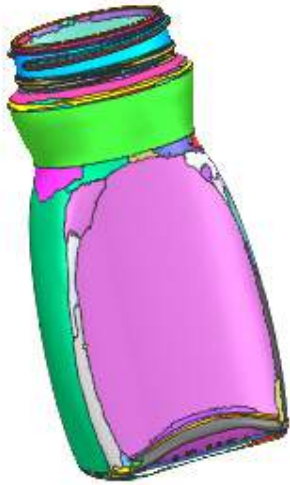
Terminology of mesh

Region Group

Region segmentation is a way to group sets of poly-faces that belong to one feature. Regions assist in the alignment of the mesh data to the global coordinate system and in selecting a region of interest for extracting the design intent, and converting it to a surface.

Regions can be automatically generated by analyzing curvature on the mesh. Afterwards regions

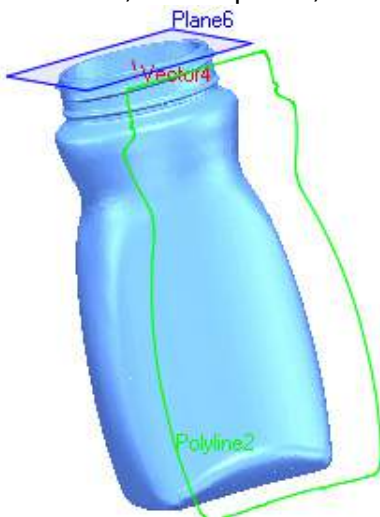
can then be modified manually if needed.



Classified regions from a mesh

Ref. Geometry

Reference geometries are used in the assistance of creating other features. These include basic geometric entities, such as planes, vectors, coordinates, points, and polylines.



A Plane, vector and polyline from a mesh

There are various types of ref. geometries that can be created. They include:

- Ref. Plane
- Ref. Vector
- Ref. Coordinate
- Ref. Polyline
- Ref. Point

Sketch and Mesh Sketch

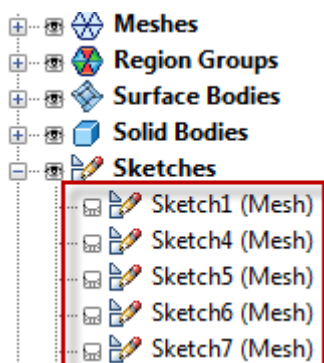
Most features start from generating a sketch. Sketch entities are drawn on a plane known as the base plane. The base plane can be a ref. plane, a planar body face or a planar region.

There are two kinds of sketches in the application. One is a Sketch and the other is a Mesh Sketch. A Sketch is used for freely drawing on a base plane in a similar fashion to other CAD software. A Mesh Sketch is used for drawing sketch entities onto section polylines by fitting lines, arcs, and splines onto selected polylines.



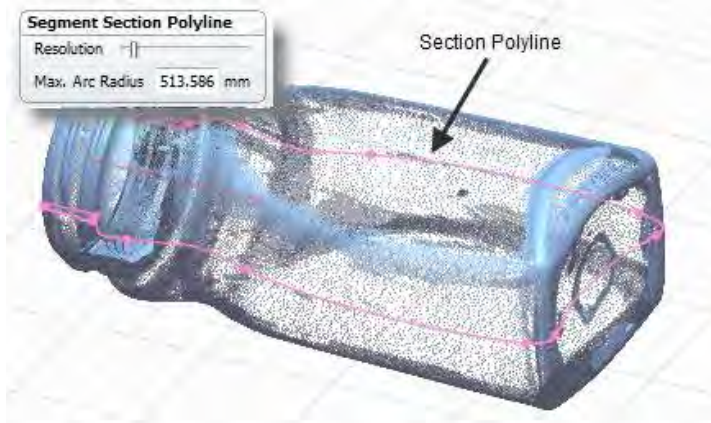
Sketch entities from a mesh

When a Mesh Sketch is generated, a suffix "(Mesh)" will be added to the end of the sketch entity's name in the Feature Tree and the Model Tree. For example see: Sketch1 (Mesh), Sketch2 (Mesh), Sketch3 (Mesh), etc.



Section Polyline

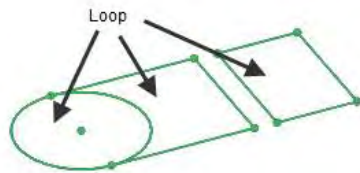
When a **Mesh Sketch** is used, the base plane is used to extract a sectional profile from the mesh or point cloud, the section polyline. Section Polylines are made from several small line segments, where the section would intersect a triangle; the resolution of the polyline is dependent on the quality of the mesh or point cloud. The application can automatically determine what areas of the polyline should be lines and arcs, and users can control the resolution by using the **Segment Section Polyline** option in the tool palette of **Mesh Sketch**.



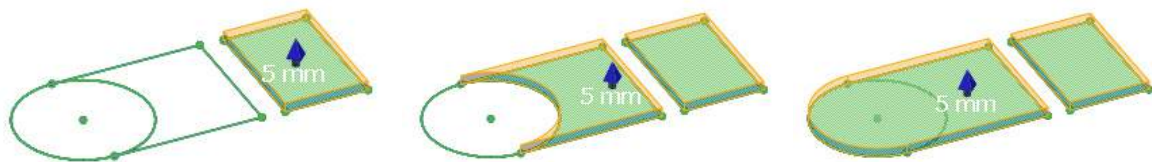
Section polyline from a mesh

Loop

A loop is an area closed by lines or arcs on the sketch. Loops can be used for generating a closed solid body. If several loops exist on a sketch, a single loop or multiple loops can be selected for solid body generation.

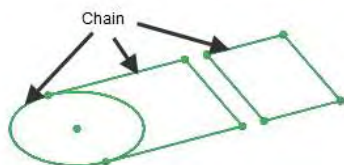


The following example shows various solid extrusion features generated from multiple loops on a sketch.

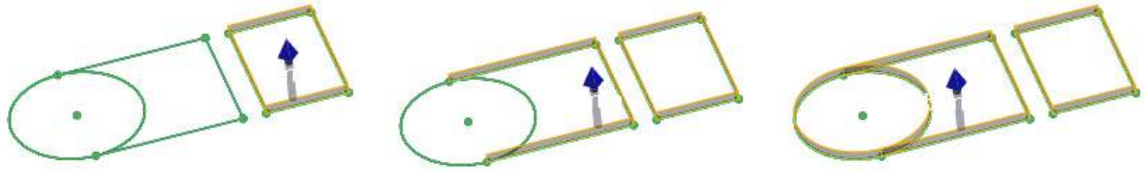


Chain

A chain is a group of sketch entities forming a boundary or shape. A chain can be either a closed or opened boundary. Chains can be used for generating surface bodies. If several chains exist on a sketch, a single chain or multiple chains can be used to generate a surface body.

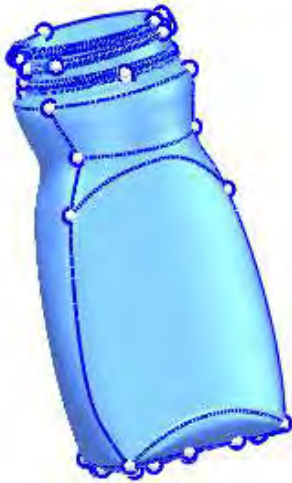


The following example shows various surface extrusion features generated from multiple chains on a sketch.



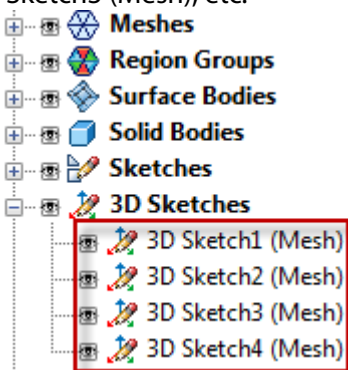
3D Sketch

3D Sketch uses spline curves which can be drawn anywhere (3D Sketch) in 3D space or drawn directly on a mesh (Mesh 3D Sketch). 3D Sketch is commonly used for the path of a loft or sweep body. 3D Mesh Sketch is mainly used for generating a curve network on a mesh. Then surfaces can be generated within the network boundaries.



Curve network by using 3D mesh sketch

If you generate a Mesh 3D Sketch, a suffix "(Mesh)" will be added to the end of the 3D sketch entity's name in the Feature Tree and the Model Tree. For example, 3D Sketch1 (Mesh), 3D Sketch2 (Mesh), 3D Sketch3 (Mesh), etc.



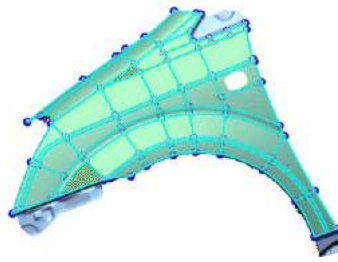
Surface Body

A body that represents a part with a set of opened surfaces, with no volume. A surface body is usually used to represent a complex shape. Surface bodies can be made with two kinds of surfaces. The first is a parametric surface which can be generated by using the extrude, loft, sweep, or revolve commands. The second is a fitting surface which fits directly to a mesh by using the mesh fit,

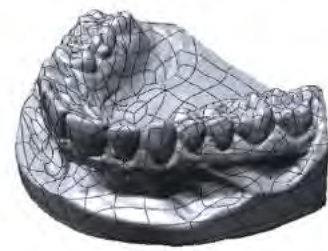
boundary fit, or auto surfacing commands. The fitting surface is used for making freeform shapes which are not parametric features, such as an organic natural object.



Parametric Surface Bodies



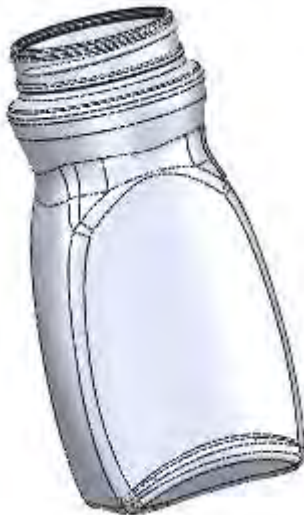
Surface Bodies Generated Using Boundary Fitting



Surface Bodies Generated Using Auto Surfacing

Solid Body

A body that represents a part with a set of closed surfaces, enclosing a volume. A solid body is usually used to represent a combination of simple base features. In the application, bodies made by using commands in the [Insert > Solid](#) menu are registered as solid bodies.



Solid body

The following terminologies are used for a solid body and a surface body.

- Face: A single, contiguous shape.
- Edge: Defines the boundary of a face.
- Vertex: The Points at a corner of a face.



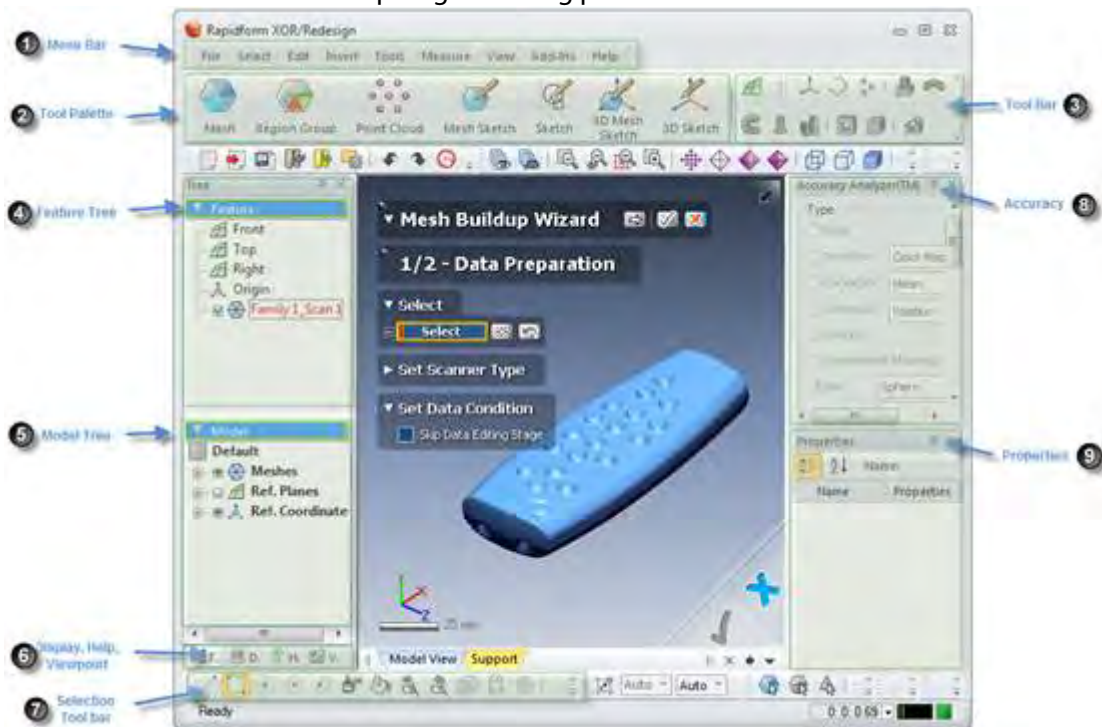
Terminology of a solid body

2.5. Introduction to the Interface

This guide is designed to help beginners understand the basic and essential usage of the user interface of Geomagic Design X.

2.5.1. User Interface Overview

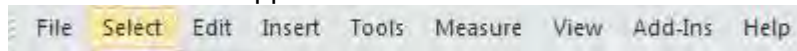
The user interface of Geomagic is designed to be easy to learn and use because it functions similarly to other CAD software. It uses familiar tools for modeling with the addition of other unique tools for an efficient and confidence-inspiring modeling process.



The following images shows elements which are frequently used in the application.

Menu Bar

All functions in the application are contained in the menu bar.



Tool Palette

The application features different modes that allows for the generation and editing of entities with only the appropriate functions displayed. Multiple modes are featured because the application can handle and manipulate many different types of entities that normal CAD programs cannot. To simplify and streamline the process there are different modes that can be used to handle the types of entities that are being worked with.



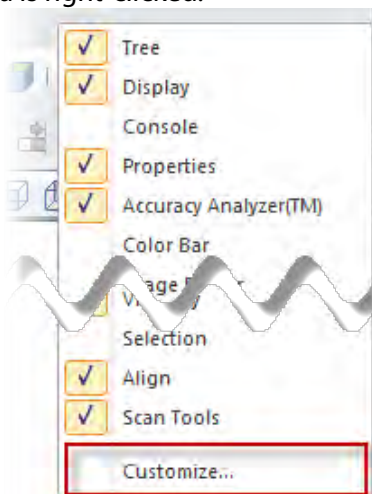
For more detail, see the chapter [User Interface > Mode](#).

Toolbar

A group of icons is called a toolbar. When icons are dimmed, they will not be available.



Toolbars can be customized by selecting **Customize...** in the menu that appears when the toolbar area is right-clicked.



Opening Customize

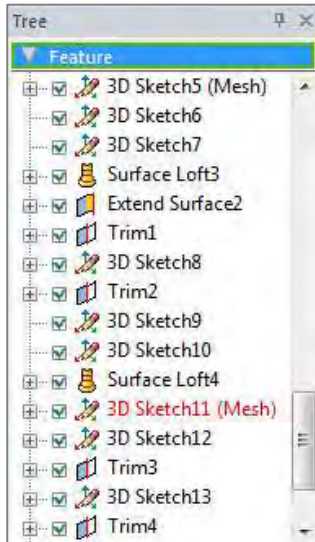


Customized toolbar

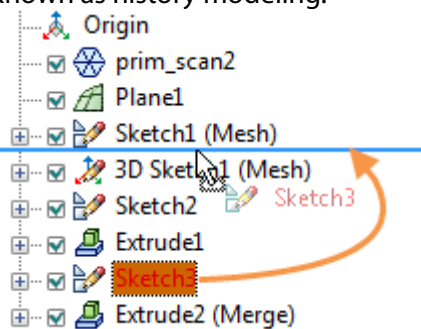
Feature Tree

Geomagic uses a method known as parametric history modeling. Parametric history modeling

allows the creation of objects by storing the operations used to construct the geometry, as well as the order of the operations and their relationships to each other.



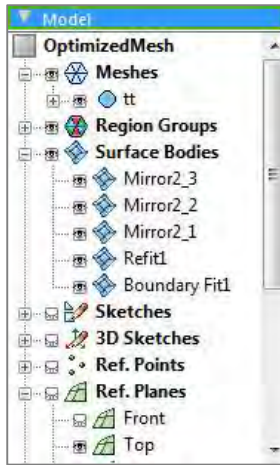
Whenever a feature such as a sketch, solid body, surface body, etc., is generated, it will be stacked in the Feature Tree in chronological order. The Feature Tree shows all changes and created entities in chronological order as well as the history for each change or creation. The order of features can be altered by dragging and dropping features. It is also possible to go back to a certain point in the modeling history by using the [Rollback](#), [Roll Forward](#), [Roll To End](#) commands. This type of modeling is known as history modeling.




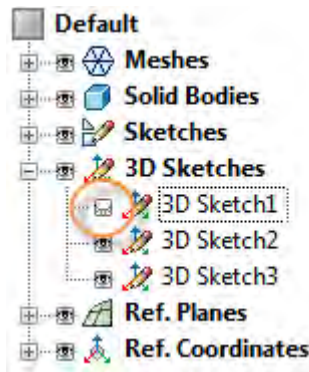
Changing the feature history by dragging and dropping

Model Tree

The Model Tree shows all created entities by categories. This pane can be used for selecting and controlling the visibility of an entity.



Clicking the  show / hide icon next to an entity will toggle between showing and hiding the entity. If a parent entity is hidden, its child entities will also not be displayed even if they are toggled to be shown.



Hiding the 3D Sketch1 entity

Display, Help, Viewpoint

Display, Help, and Viewpoint are located in the same window as Feature and Model Tree. Navigate between using the tabs at the bottom.



Display – The Display pane is used to manage how entities are displayed. For example, a model can be set to be transparent or brightly lit. Textures, normals, faces, boundaries, etc., can be shown or hidden as well.

Help – The Help pane shows the table of contents of the user guide. It can be navigated through by selecting a title on the table of contents in the Help pane or by pressing the F1 key when a command dialog tree is floating on the Model View.

Viewpoint – The Viewpoint pane can save the view states that are currently being used. It works much like taking a picture. If a viewpoint is taken, the visible state entities and viewing direction are saved.

Selection Toolbar

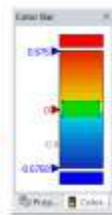
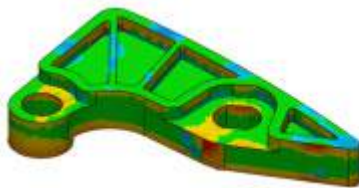
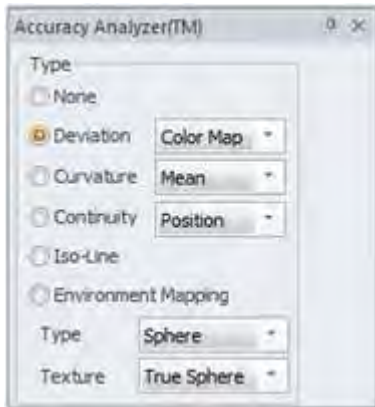
When poly-faces and poly-vertices are to be selected, the application offers various selection

methods to use. For example, poly-faces can be chosen by dragging a line, circle, or rectangle. Painting the desired area on the mesh is also possible, and can be expanded to connect all poly-faces. The selection method can be changed in this toolbar.



Accuracy Analyzer(TM)

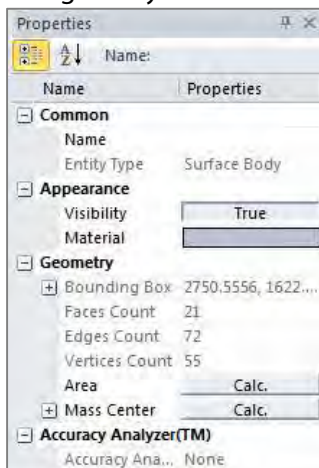
The Accuracy Analyzer™ pane is very useful for checking the quality of bodies, meshes and sketches. After a surface body is created, deviation between the scan data and the generated body can be immediately checked. The Accuracy Analyzer™ is available in default mode, Mesh mode and all 2D / 3D Sketch modes.



In this image the deviation of created bodies to the original mesh is shown. The green color is the desired user-defined ideal deviation and the red and blue colors are deviations outside of the desired range.

Properties

After an entity is selected, its properties can be viewed and changed. For example, after a mesh is selected, its bounding box size can be checked in the Properties pane. The mesh color can be changed as you desired.



2.5.2. Understanding User Interface Philosophy

The following chapters introduce general rules when you use the application.

Dialog Tree Convention

How to use a Dialog Tree

Selecting Entities to Actively Control

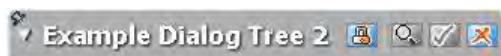
If a command is executed, a dialog tree will appear in the Model view. Some dialog trees require the user to select entities for various fields in order to proceed. The active control is a box highlighted in a yellow color by default. It indicates that the currently selected control is ready for the selection of entities. When an entity is selected the name will be registered under the active control. To select entities for another control, just click the desired control box to make it active.



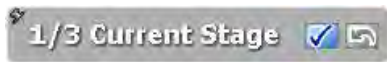
Below are examples of what a user might typically see in a dialog tree. Each command or tool has its own set of options in its dialog tree so the images are used as a general guideline on the layout and different functions a user might see.



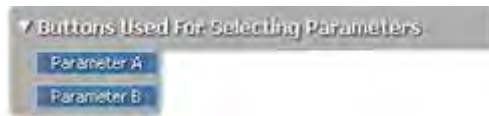
The name of the command will be at the top of the dialog tree. In this example there are multiple stages and the first two buttons are the **Previous Stage** and **Next Stage** buttons that will go forward and backward through stages. The next two buttons are the **OK** and **Cancel** buttons to apply the command or exit out of it.



This second example shows a command that has two more useful buttons. The first is the **Lock button**. It is used when there are multiple changes that need to be made using the same tool. With this button pressed, the **OK** button will not quit the command, but instead restart the command again. The button with the magnifying glass is the **Preview** button which allows the user to preview changes made to a model in real-time without actually applying the changes.



This shows what stage a command or tool is in. A command must have more than one stage for this type of dialogue box to appear. The first button is the **OK** button which applies changes made in the current stage without exiting the entire command, and the second button is the **Undo** button which will deselect the last selection made.



Some options will have selection buttons that allow selection for different parameters. This example has two selectable parameters, but commands will vary in how many they might have.



Option check boxes toggle different parameters on or off.



Radio buttons are used in a dialog tree to select an option. Only one radio button can be chosen at a time.



This is a selection area. The input box is used to make a selection in the viewing area. In this example an entity labeled "teeth" has been selected. There are two buttons next to the input box. The first is the **Select None** button will delete all selected entities and the second button is the **Undo** button which will deselect the last chosen entity.

Dialog Tree Controls

The dialog tree is designed intuitively allow you complete a task; through the use of common tools. All tools have icons; the icons are only shown if needed in a specific dialog. The following icons can be found in the dialog tree.



Lock(Don't Quit Command With OK) – If this icon is toggled the dialog tree will remain open even if the OK icon for the command is selected. It is designed for commands that are consecutively applied several times in a row.



Break – If an operation is executing, it can be stopped by using this button. The keyboard shortcut is the Esc key.



Delete Selected Entities – This button will remove chosen entities.




Preview – If this button is clicked, the result of the command will be displayed without executing it.





Enable Realtime Preview – If this button is toggled, the results of the command will be displayed in real-time, whenever an input condition is changed.






Measure Distance, Radius, and Angle – After clicking these icons, points on the model can be picked to extract distance, radius, and angle, respectively. The measured value is directly inputted into the text or spin box.

 **Estimate** – If selected, the application will automatically recommend an appropriate value.

 **Flip Direction** – This button reverses the direction of a control.

 **Add** – This button adds an additional entity set.



 **All** – This button selects all entities which are available.

 **Auto** – Some text boxes have the value of Auto. If the value is set as Auto, the application will estimate and use an appropriate value by analyzing the size of meshes and poly-faces. If the  estimate button is selected, the estimated value can be seen. A numerical value can be used instead of Auto. To reset to the Auto value, enter “0” as a numerical value.

Selection Convention

How to Select Entities

Look at the Cursor Icon

In the Model View, the mouse cursor has two modes. One is for the selection mode , and the other is for the viewing mode . These can be switched by clicking the middle mouse button. Entities can only be selected when the mouse cursor is in the selection mode.

How to select entities

Dragging Selection – Clicking then dragging can be used to select a single entity or multiple entities.

Picking Selection – A single entity can be chosen by single clicking on it.


From Feature, Model Tree – An entity or entities can be selected directly from the Feature Tree or Model Tree.

Selecting Multiple Entities – Multiple entities can be chosen by using the Shift key during selection. To de-select entities, use the Ctrl key.

Using Select Modes

To select a specific area to edit (or remove) poly-faces or poly-vertices can be difficult because the mesh can be complicated and can consist of thousands or millions of vertices and faces. The Select Mode tool bar located at the bottom left of the application enables the selection of poly-faces and poly-vertices in a variety of ways. This tool bar can accelerate working speed while managing mesh or point cloud data.



The last icon on the toolbar is the  Visible Only selection mode that allows for only the selection

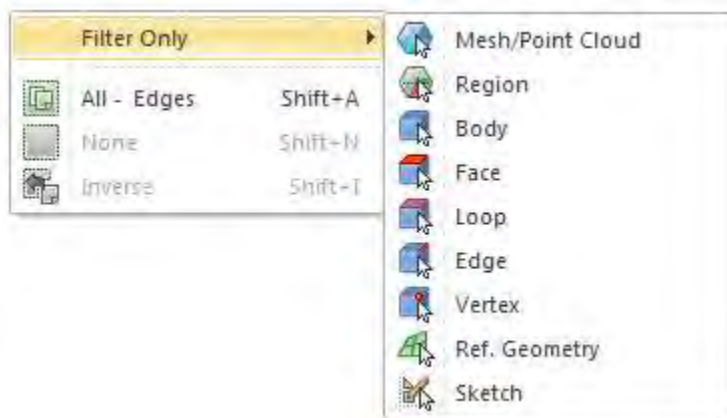
of visible poly-faces and poly-vertices in the model view. It is useful for avoiding the selection of back-side poly-faces and poly-vertices.

Using the Select Filter

The application can create and manipulate various types of entities. For example, the application can utilize and create meshes, regions, bodies, faces, edges, sketches, dimensions, etc. The Select Filter command enables the selection of only desired entities and intelligently works within a command or a chosen mode. For example, if the Smooth command is executed, only the Regions, Poly-Faces and Poly-Vertices filters will be offered in the Select Filter toolbar based on the selectable entities for the current input condition.



Filters can also be directly accessed by clicking the right mouse button in the Model View.




Mode Convention

Understanding Mode Convention

Accept and Cancel a Mode

When a mode is entered, two buttons will be shown in the right bottom corner of the Model view.

 **Accept** – This button applies all changes made in the current mode, and then exits the mode.

 **Cancel** – This button cancels all changes made in the current mode, and then exits the mode.


Undo and Redo

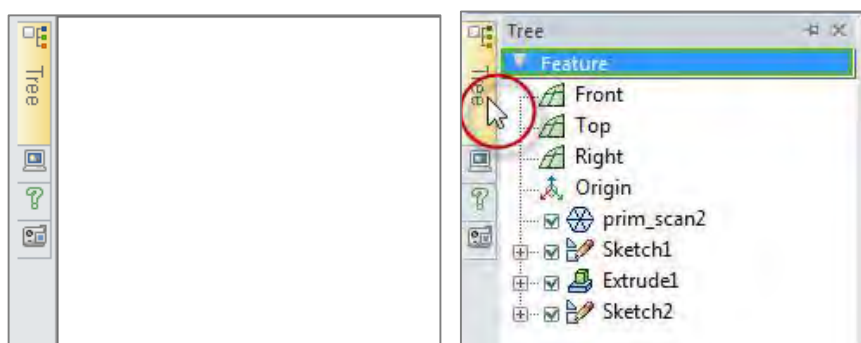
Undo can be applied step-by-step in a mode but if the mode is exited, the undo history will be reset.


Docking Bar Convention

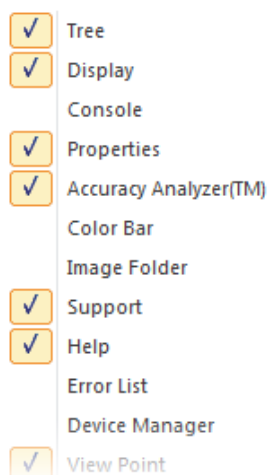
Docking Bars

Docking bars are located at the left, right, and bottom sides of the application. Docking bars are used to hide frequently used windows to make more room for the model view or if a user desires to simply make the look of the program cleaner and more organized. These docking bars can be repositioned by dragging and dropping a docking bar into that particular space or into another docking area. There are icons that can be located on the right top side of a docking bar.

-  **Auto Hide** – This button hides the docking bar to the side frame. If the cursor is hovered over the name of a docking bar, it will automatically re-appear.



-  **Close** – This button closes the docking bar. To show the docking bar again, click the right mouse button on any tool bar area, and then check the desired docking bar.



To hide all current docking bars, press the **F12** key, which will maximize the Model View. It is a toggle function, so if the **F12** key is pressed again, all hidden docking bars will appear.

2.5.3. Navigating Entities

Rotate, Zoom, and Pan

In the Model View, the mouse cursor has two modes. One is selection mode and the other is viewing

mode. These modes can be switched by clicking the middle mouse button. Entities can only be selected when the mouse cursor is in selection mode. The Rotate, Zoom, and Pan functions will be explained below for each mode.

In Selection Mode

Rotate – Right mouse button

Zoom – Shift + Right mouse button (or scroll wheel)

Pan – Ctrl + Right mouse button (or Left and Right mouse buttons together)

In Viewing Mode

Rotate – Left mouse button (or Right mouse button)

Zoom – Shift + Left or Right mouse button (or scroll wheel)

Pan – Ctrl + Left or Right mouse button (or Left and Right mouse buttons together)


Note:

The cursor operates similarly in viewing mode as it does in selection mode, except it does not have the ability to select or make changes to any entities. The purpose of viewing mode is to simply view the model; and avoid making any selections, or unwanted alterations.

In the images below a model is shown being rotated with the cursor in selection mode.


1



In this image, the mouse is in selection mode. Entities can be selected and modified. 


2



In this image, the right mouse button is pressed and held down, while the mouse is moved; transforming the cursor into the rotate mode cursor. 

3



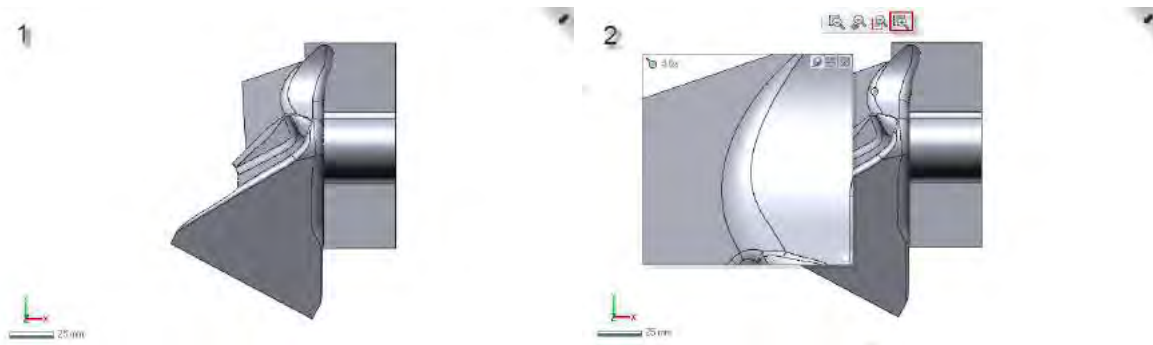
In this image, the right mouse button has been released, stopping the rotation of the model, and returning the mouse cursor back to its default selection state. 

Using Zoom Functions

The Zoom functions are used to enlarge the display of an area, in order to examine it more closely or to fit a model to the screen. There are four types of zoom functions which can be accessed from the tool bar, or by using the [View > Zoom](#) menu.



In the first image below, the model is open in the viewing area. In the second image the PIP zoom function has been selected, as highlighted in red, allowing a portion of the model to be enlarged for examination.



Below are the different Zoom commands and their functions:

Zoom Area – Zooms to a selected area. (Shift + Z)

Zoom Previous – Returns to previous zoom states. (Shift + X)

Zoom Fit – Zooms to fit all shown entities. (Ctrl + F)

Zoom PIP – Shows zoom window, picture-in-picture. (Ctrl + W)

Changing Viewpoints

Viewpoint functions are used to view the model from defined directions, based on the global coordinate system. Different viewpoints can be selected from the tool bar, or by using the View > Viewpoint menu.



In the first image below, a model is shown in a random view orientation. Using the selected viewpoint, highlighted in red in the second image, the model view is rotated and perfectly aligned to the Front view, based on the global coordinate system.



Below are the different viewpoints and their functions:

Front Viewpoint – Changes viewpoint to the front view. (Alt + 1)

Back Viewpoint – Changes viewpoint to the back view. (Alt + 2)

Left Viewpoint – Changes viewpoint to the left view. (Alt + 3)

Right Viewpoint – Changes viewpoint to the right view. (Alt + 4)

Top Viewpoint – Changes viewpoint to the top view. (Alt + 5)

Bottom Viewpoint – Changes viewpoint to the back view. (Alt + 6)

Isometric Viewpoint – Changes viewpoint to a standard isometric view. (Alt + 7)

Rotate Viewpoint CCW – Rotates the viewpoint counterclockwise by a quarter-turn (90 degrees).

Rotate Viewpoint CW – Rotates the viewpoint clockwise by a quarter-turn (90 degrees).

Flip Viewpoint – Flips the viewpoint horizontally along the vertical axis.

Showing and Hiding Entities

The application can handle many types of entities; but sometimes the need to filter out certain entities is required. The Visibility toolbar can show and hide the most frequently used entities, to reduce the number of on-screen elements. Entities can quickly be shown or hidden by clicking icons in the tool bar or by using the [View > Visibility](#) menu.



In the first image below, only surface bodies are shown by the Visibility toolbar. In the second picture, meshes have also been shown.



Below are the different visibility options and their functions:

Meshes – Show or hide meshes. (Ctrl + 1)

Point Clouds – Show or hide point clouds. (Ctrl + 3)

Surface Bodies – Show or hide surface bodies. (Ctrl + 4)

Solid Bodies – Show or hide solid bodies. (Ctrl + 5)

Sketches – Show or hide sketches. (Ctrl + 6)

3D Sketches – Show or hide 3D sketches. (Ctrl + 7)

Ref. Points – Show or hide Ref. points. (Ctrl + 8)

Ref. Vectors – Show or hide Ref. vectors. (Ctrl + 9)

Ref. Planes – Show or hide Ref. planes. (Ctrl + 0)

Ref. Polylines – Show or hide Ref. polylines.

Ref. Coordinates – Show or hide Ref. coordinates.

Measurements – Show or hide Measurements.

2.6. Data Processing

This guide is designed to help beginners understand how to import and export data; and basic workflow from scan data to CAD data, which will be your final destination.

2.6.1. Changing Units

When installing the software, the default measurement unit will be set to "mm" when measuring in the software and importing scan data. If the working environment uses a different measurement unit than "mm", the unit type can be changed in the application.

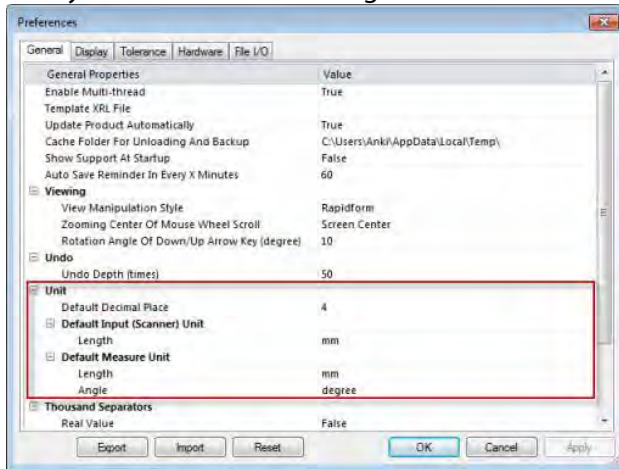
Setting Unit Type After Software Installation

When running the application for the first time after installation, the **Initial Setup Dialog** box will be displayed which will enable the setup of general configurations. Unit system measurements can be changed under the **Unit** category in the dialog box. The **Default Input (Scanner) Unit** option is applied only when importing scan data but **Default Measure Unit** governs all measurement and display units shown while using the application.



Setting Unit Type While Using the Application

This **Initial Setup Dialog** box will only be shown once during the initial start up, but you can change settings at anytime while the application is running. Go to **File > Preferences** and click the **General** tab. System units can be changed under the **Unit** category.

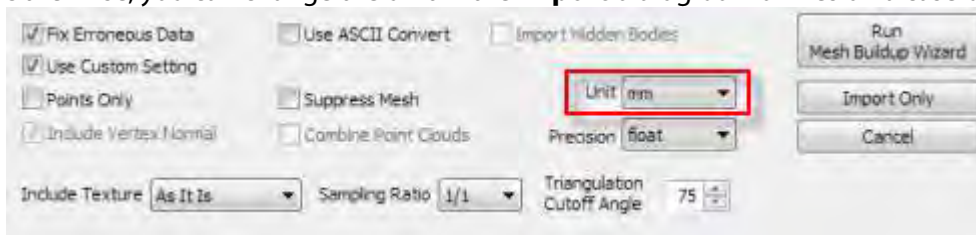


Note:

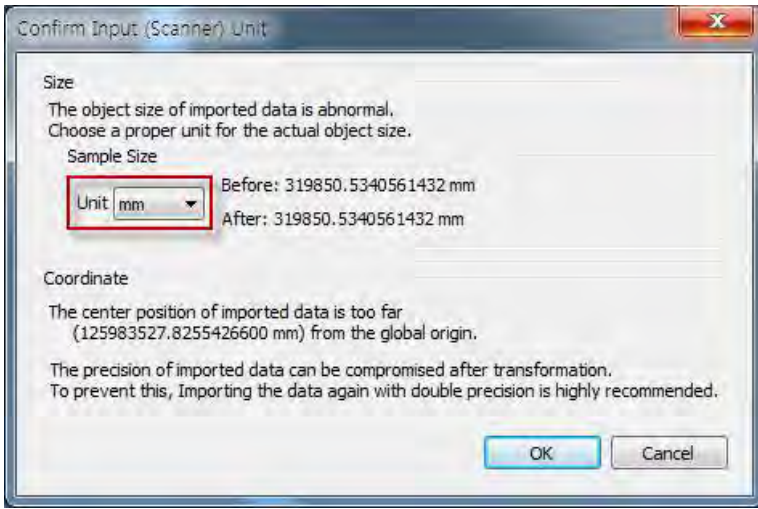
The software environment can be setup in the **Preferences** box for better performance and ease of use. For more detail see [File > Preferences](#).

Setting Unit Type While Importing Scan Data

Scan data is usually stored in files as unit-less data, meaning no unit is specified, unlike CAD files. Most scanners can only capture data in either millimeters (mm) or meters (m). If your scanner supports measurement in another unit, you can set that unit globally from the **Preferences**. Otherwise, you can change the unit in the **Import** dialog box for files on a case by case basis.



If an imported scan data part is very small or large, it could cause problems while generating features based on the mesh. If this occurs, the application will show the 'Confirm Input (Scanner) Unit' dialog box to verify the input measurement unit of the scan data. Normally long range scan data is very large in size and located very far from the global origin. If this type of file is to be imported, it is recommended to import the scan data with the Double Precision option to preserve the accuracy of the scan data.



Note:

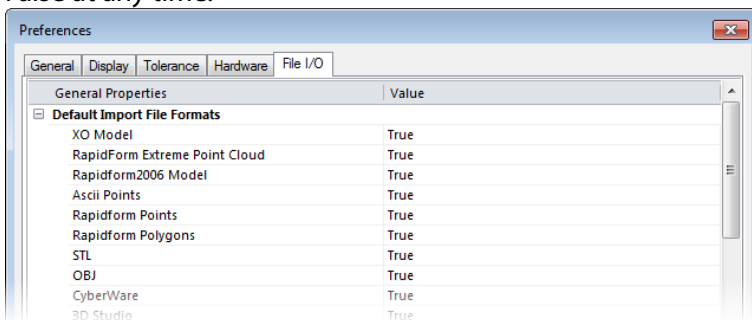
After scan data is imported you can also change the default measurement unit type. Go to **Tools > Scan Tools > Transform Scan Data** and check the Scale option to change the scan data default measurement unit.

2.6.2. Importing Scan Data

Settings for Favorite File Format

The first step of mesh modeling and reverse engineering is to import scan data. The application supports not only standard mesh formats, but also various scan data formats from multiple scanner types and manufacturers.

When scan data is imported, many kinds of file formats are available in the drop-down list box. If a scanner format is not frequently used it can be removed from the drop-down list box by setting the value to **False** in the **File I/O** tab in **File > Preferences**. Values can be changed between True and False at any time.



Note:

Some OEM products have limitations on importing other scan data formats.

Importing Scan Data

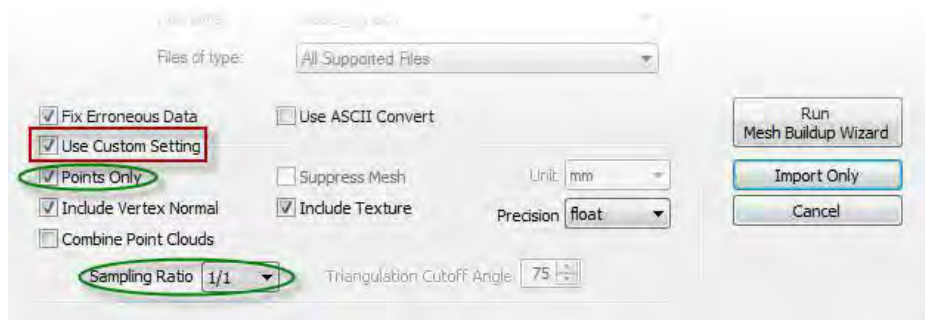
To import scan data, use the **Insert > Import** command. The Insert menu adds features in the Feature Tree, and constructs a history. Importing scan data is also treated as a feature in the history tree.

Single or multiple scan data files can be imported through the Import dialog box.

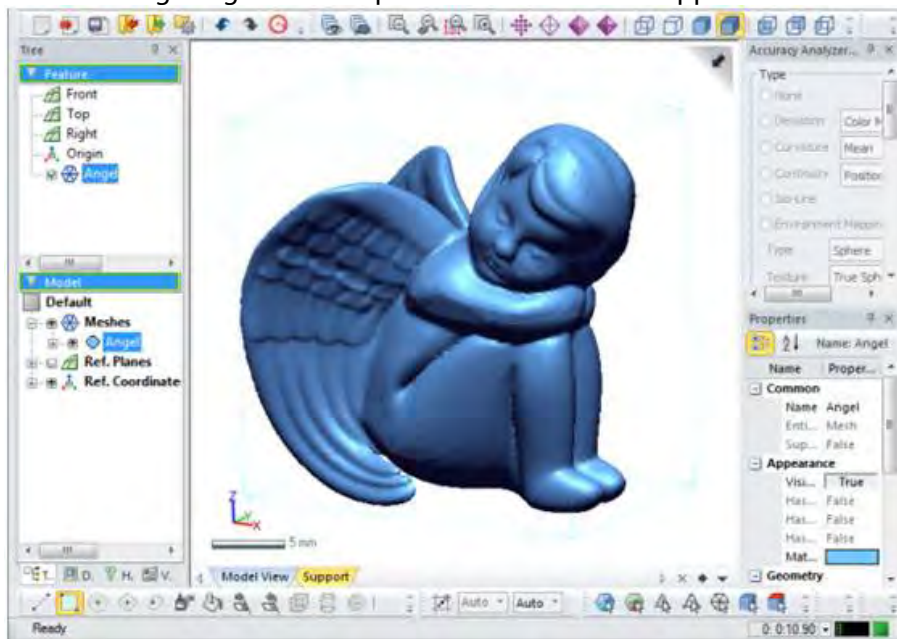
Name	Date modif...	Type
Output	2010-02-0...	File folder
1_tube.fcs	2010-02-0...	FCS File
2_tube.fcs	2010-02-0...	FCS File
3_tube.fcs	2010-02-0...	FCS File
4_tube.fcs	2010-02-0...	FCS File
5_tube.fcs	2010-02-0...	FCS File

Using Importing Options

Certain parameters can be set during the import process. If importing mesh data as point data is desired, check the **Use Custom Setting** and the **Point Only** options in the Import dialog box. If a scan data file is very large in point density and importing it as a lower is desired, the **Sampling Ratio** option can be used.



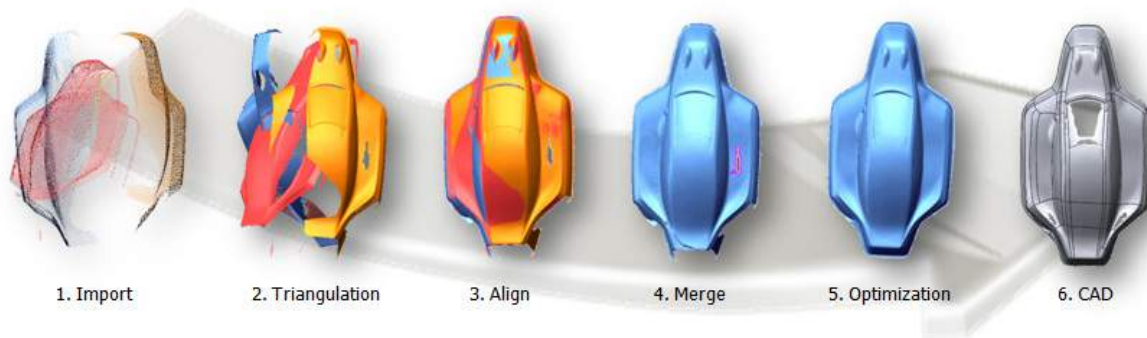
The following image shows imported scan data in the application.



After importing scan data, the following workflows can be used:

- **Basic Scan Data Processing Workflow:** Generate, edit, and enhance a mesh for FEM, RP, and visualization.
- **Basic Reverse Modeling Workflow:** Generate surface or solid bodies for reverse engineering.

2.6.3. Basic Scan Data Processing Workflow



Note:

1. If a mesh is imported, the "Triangulation" stage can be skipped.
 2. The "Align" stage can use not only meshes, but also point clouds (for best results, mesh data is recommended).
 3. The "Merge" stage can only be used with point clouds, however a mesh will be generated.
- The "Triangulation" to "Optimization" commands can be completed in a single command by using the Mesh Buildup Wizard. (See. [Tools > Scan Tools > Mesh Buildup Wizard](#))

Import

Imports scan data. The scan data can be point clouds or meshes.

Related command: [Insert > Import](#)

Triangulation

Generate a polygon model by connecting neighboring points in a point cloud. The result is called a mesh. If scan data is already in mesh form, this stage not necessary.

Related command: [Tools > Scan Tools > Triangulate](#)

Align

Align scan shots to each other, by overlapping the common.

Related command: [Tools > Scan Tools > Align Between Scan Data](#)

Merge

Uses the overlapped scan shots from the align stage and removes overlapped areas while sewing the remaining areas to create a single mesh.

Related command: [Tools > Scan Tools > Merge](#)

Optimization

A mesh can be modified to a desired level of optimization. Erroneous triangles and holes on a mesh can be removed to make a water-tight mesh and the shape of the triangles can be optimized by unifying them. As well, the number of triangles can be reduced or increased. A mesh also can be reconstructed by cutting, offsetting, re-scanning, etc.

Related command: [Tools > Mesh Tools](#)

CAD (Reverse Design)

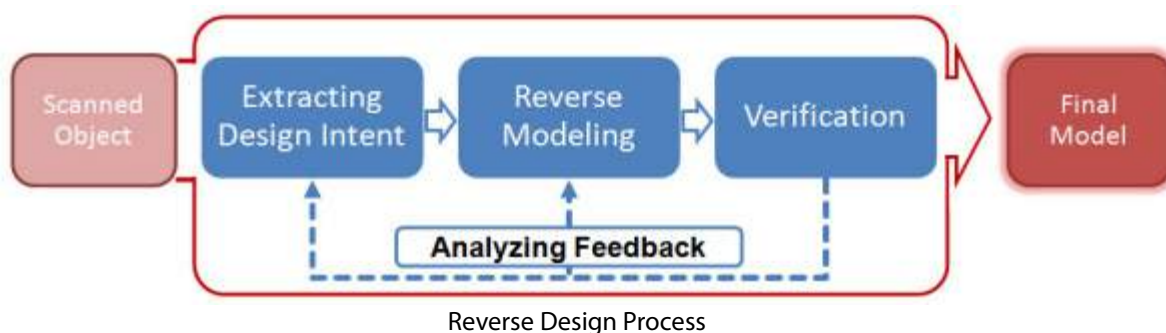
When a user desires to make a complete mesh, this stage is not necessary. However if CAD data is the goal, there are two methods. The first is to make a fitting surface by wrapping surfaces on a mesh. With this method the quality of a surface is directly related to the quality of the mesh. The second method is parametric CAD modeling. By extracting design intent from the features on a mesh, ideal solid or surface bodies can be created. With this method, the quality of the mesh is not critical but it needs enough details to analyze a feature.

Related command:

- Fitting Surface Generation: [Insert > Surface > Mesh Fit](#), [Boundary Fit](#), [Auto Surfacing](#)
- Parametric CAD Generation: [Insert > Solid](#), [Surface](#), [Modeling Wizard](#), [Modeling Feature](#)

2.6.4. Basic Reverse Design Workflow

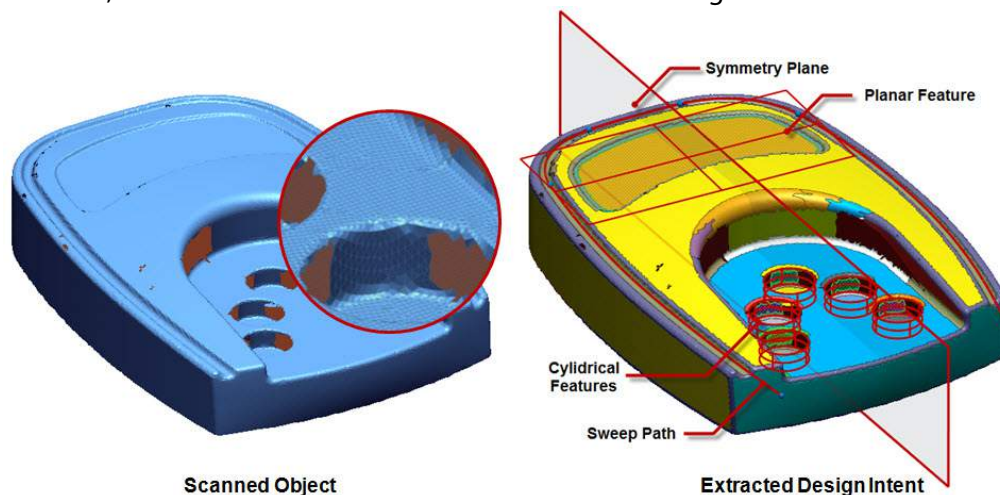
General product design follows several processes, usually separated by job discipline. Reverse Design, however, is much more linear. After the scanning process, to capture the shape of a part, the modeling process can begin. The modeling process can be divided into three processes, Extracting Design Intent, Reverse Modeling, and Model Verification. Model Verification is used to check in-progress work, and the cycle repeats often, creating a closed loop system.



Extracting Design Intent Process

Design Intent is the original idea of the designer, how he intended something to work. The best results are achieved by uncovering and using that Design Intent in the mode. See the examples in the image below.

Any design elements extracted from the scan data will be helpful in completing the modeling process. For example, if a scanned 3D object has cylindrical or conical features, a rotational axis can be automatically or semi-automatically be extracted. A central path from a scanned pipe shape can also be extracted, as well as an accurate radius value from a fillet region in a scanned 3D object.



Extracting design elements from 3D Scan Data

Reverse Modeling Process

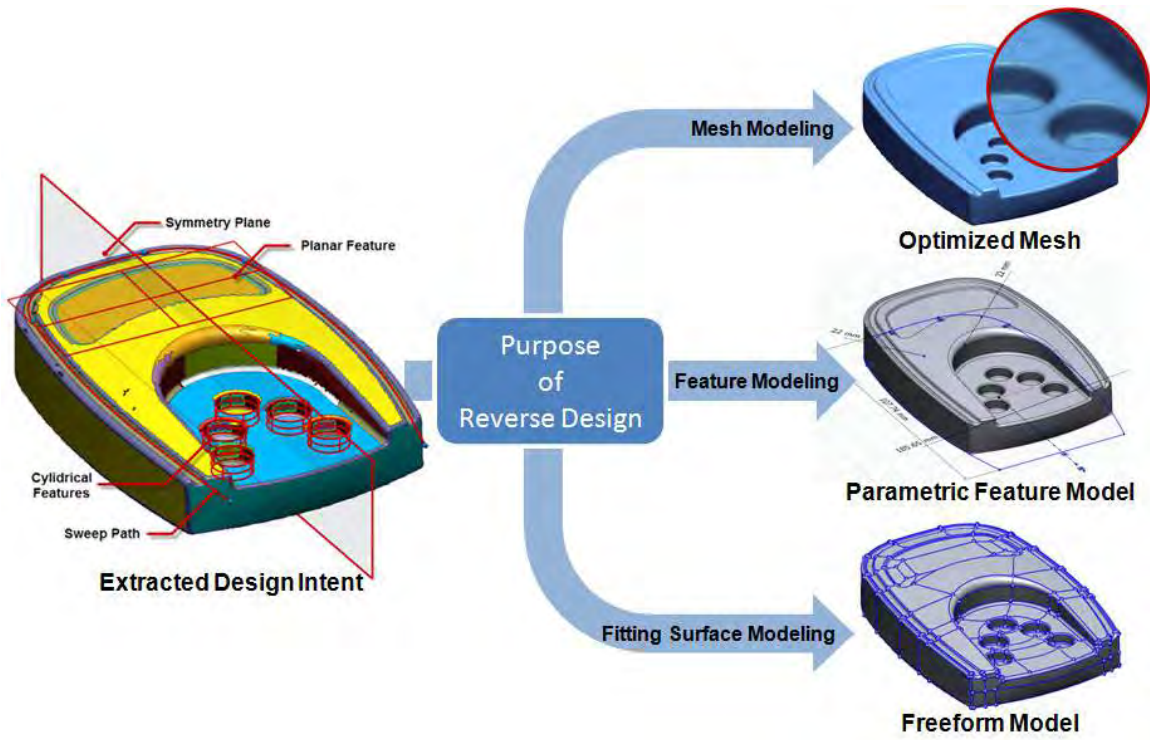
The Reverse Modeling process is the process of creating an optimal 3D model from 3D scan data (Point Cloud or Mesh), that is generated during the Scan Data Processing phase. This process is the core of Reverse Design, where optimized mesh and 3D features, such as 3D curves, 3D surfaces, and 3D solid bodies, are created by using various modeling methods.

The Mesh Modeling method creates an optimized mesh that contains important information by applying various geometric and mathematical operations.

The Feature Modeling method creates 3D geometric feature shapes based on extracted design intent and elements from 3D scan data

The Fitting Surface Modeling method creates fitted freeform surfaces on complex freeform feature shapes.

The Hybrid Modeling method creates a complex feature model from 3D scan data by using the Feature Modeling method in conjunction with the Fitting Surface Modeling method.

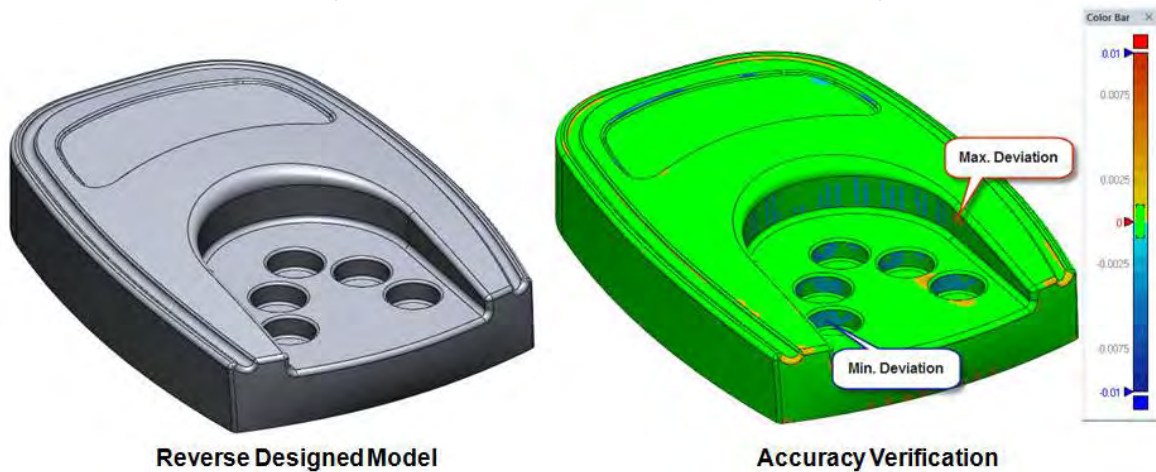


Various modeling methods based on Reverse Design intents

Model Verification Process

The Model Verification process is the process of analyzing intermediate and final models with raw 3D scan data and verifying the results, as shown in the image below.

In this process, the analysis of deviation between the intermediate and final results created in the Reverse Modeling process and raw 3D scan data can be performed. Additionally, the model can be checked for problems or defects that would make it undesirable for manufacturing. The analyzed results can be used for creating an optimal 3D model within allowable design tolerances.

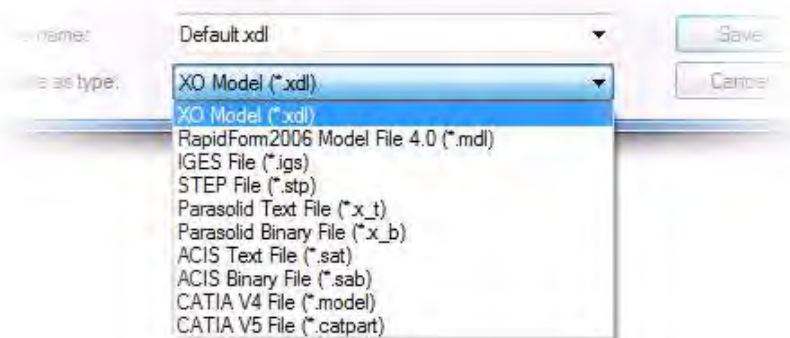


Model Verification

2.6.5. Exporting a Model

Export

Data can be exported out of the application in a variety of formats. Many multi-platform formats are offered, as well as some program-specific formats. Exporting can be done through the **File > Export** command or pop-up menu on the Model Tree to save the entity as a specific file format, ready for manipulation in another program.



LiveTransfer

The **File > LiveTransfer** command which is a unique Geomagic function that allows the real-time transfer of data into other CAD applications. You can export not only CAD data but also its modeling

history which includes parametric information.

The following CAD application is supported under the LiveTransfer command.

SolidWorks
Siemens NX
Creo (Pro/E)
AutoCAD
Inventor

3. Understanding Scan Data

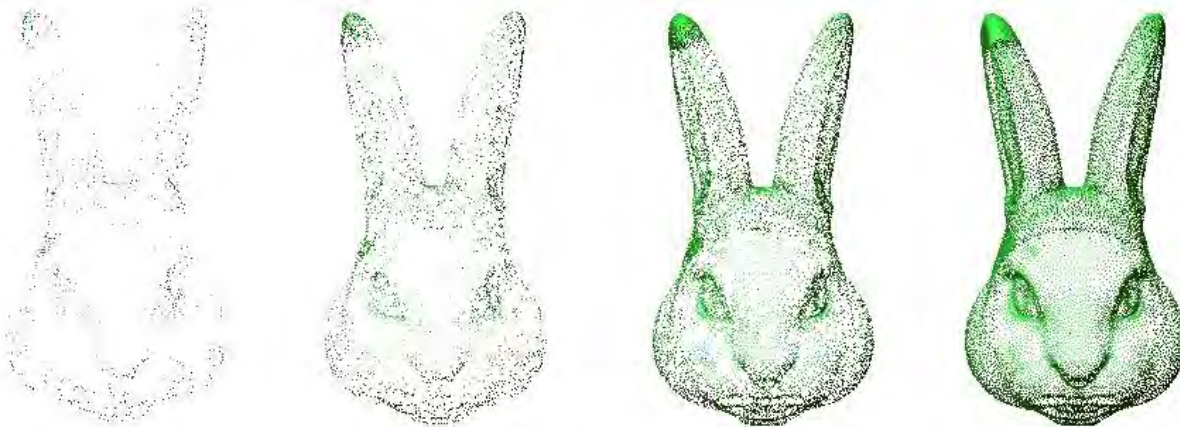
This guide is designed to help beginners understand the basic information about the characteristics of scan data when deal with in the application.

3.1. Why 3D Scanning?

3D scanning is getting popular in various fields, and its usage is different for each purpose. This means that there are different kinds of scan data, and many details to understand. This guide will give basic information of 3D scan data for you to understand.

3.1.1. Power of 3D Scan Data

A 3D scanner is a device that captures a real object or environment as 3D shape data. The collected 3D information is converted into digital data commonly called 3D scan data or scan data. 3D scan data is a set of points. A point represents a location on a real object and contains the X, Y, and Z coordinates. Numerous points can be used to describe a real object. For example, a digital photo with a high resolution pixel count can represent the detailed shape of a real object.



3D scan data by resolution

A point set, also known as a point cloud, can be converted into an informative digital model with software operations and used in various industrial fields. 3D scan data has the following strengths:



- Quickly create digital versions of real objects
- Accuracy
- Capture complex freeform surfaces
- Capture small to large scale objects (depending on 3D scanner)
- Obtain color information (depending on 3D scanner)
- Simulate environments and situations
- Change to different scales and measurements easily
- Easily extract length, height, width, volume and position data
- Extract sectional information
- Compatibility in a general PC environment

3.1.2. Applications of 3D Scan Data

Since 3D scan data can represent a real object with high accuracy, it is used for various purposes. The use of scan data is also increasingly expanding to custom markets as scan technology becomes easier to use and more intuitive. Currently, 3D scan data is used primarily for the following purposes:

3D Printing, Rapid Prototyping (RP)

The technology of 3D Printing or Rapid Prototyping (RP) can create a quick physical mockup of digital 3D data (generally .stl data is used) by laying down successive layers of material using sectional information of a model. Using RP, product designers can make a mockup faster than with other conventional manufacturing technologies, such as NC machining. A mockup can be used to analyze a part, check collisions with other parts, decide ergonomic designs, and reduce design iterations; amongst other uses.

Reverse Design

The process of making CAD data from a real object is called reverse designing. A real-world model is not ideal to use in designing CAD environment, because it can be deformed by heat, stress and abrasion, causing the 3D scan data of that model to also include those deformities. Depending on the purpose, the decision to make a CAD file with an objects deformed shape, or redesigning it by extracting design intent is important to consider.

Measurement and Inspection

3D scan data is increasingly used for metrology or inspection. The more accurate points that are used to measure a feature will increase the credibility of an inspection. The scan data of a manufactured part can be aligned with its CAD model (nominal data), and subsequently compared to check for differences and whether or not they pass/fail within set tolerances.

Medical

3D scan data is used in the medical field to make customized teeth, organs, and other vital parts for a patient. For example, a 3D scanner can scan a patient's dental features and generate a digital model, in order to create a treatment and to simulate tooth movement. 3D scanners are changing the traditional dental treatment methods used to generate dental impressions and handmade artificial teeth.

Cultural Heritage

The cultural heritage domain also uses cutting-edge 3D scanning technology. 3D scanners can generate accurate 3D files which can be used to restore historic buildings and objects. 3D scan data is also used to simulate or visualize historic places.

Entertainment

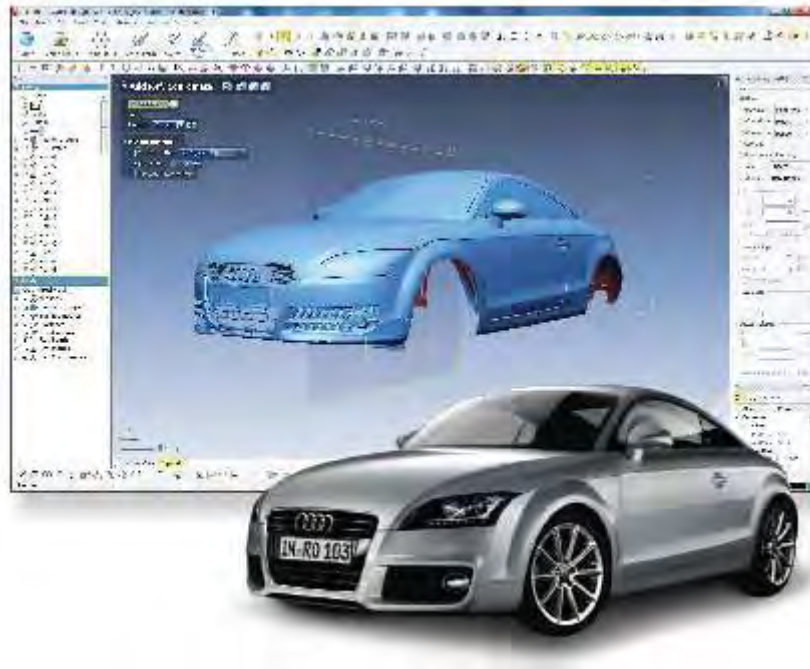
The entertainment and video game industry also uses 3D scanners to reduce modeling time and make more realistic 3D graphics. Scanning a real object or a human is much faster than modeling them in 3D graphics software by hand.

Survey

Traditionally, the construction and civil engineering fields relied heavily on surveying, but it took too much time and the workers would be exposed to the environment being surveyed. Even using these methods there was often a lack of information, which caused delays in schedules and generated additional costs. 3D scan technology is now used in these types of projects by removing uncertainty, finishing the project quickly, and reducing costs.

3.1.3. Is 3D Scanning the Only Step in Creating a CAD-Ready File?

When a user has never experienced 3D scanning before they might think that after scanning an object, a CAD-ready data file will be generated instantly. The truth is that CAD systems cannot handle millions or even thousands of data points. This might confuse a first-time user because 3D scan data is displayed in 3D on a PC; however, 3D scanning is not the end of the design process to create a CAD-ready file, but just the first step. 3D scan data is merely raw information with a large number of points pertaining to the surface of an object, and usually not directly usable without further intervention. 3D scan data usually needs to be processed to be useful and can be done in scan data processing software, such as Geomagic Design X. This type of software is necessary to quickly and accurately get a high quality mesh or CAD model. Geomagic Design X is the best tool available to transform raw scan data into valuable CAD-ready data.



Geomagic Design X has the following features that are useful in processing raw scan data:

Point Cloud & Mesh Cleanup – Remove unnecessary scanned areas, outliers, and heal point clouds and meshes.

- Automatic point cloud/mesh healing & cleaning
- Automatic scan data processing from multiple point clouds into qualified mesh – Mesh Buildup Wizard™
- Creation of watertight and optimized mesh from raw scan data just with one click – Rewrap™
- Automatic hole filling with curvature continuity

Best-in-class Point Cloud & Mesh Operations – Change resolution, optimize triangle size, edit shape, and make a water-tight mesh.

- Cross sectioning of freeform curve designs with point clouds and meshes
- Instant mesh optimization for direct use in RP, CAM & CAE
- Detailed resolution controls (decimate & subdivide)
- Smoothness controls (global & local smoothing)
- Professional and highly interactive point cloud/mesh editing tools
- Powerful operations such as remove marker, de-feature, hole filling, fix boundary, smooth boundary, fit boundary, fit region to analytic shape, split & trim, divide, thicken, offset, etc.

Direct Color Texture Editing – Add texture or modify texture for visualization.

- Color-texture aware mesh operation and texture preservation
- Color parameter adjustments and editing
- Automatic color balancing between multiple scans
- Single texture creation from multiple textures while minimizing mosaic textures

- 3D data compression and streaming for web publishing

Align Wizard™ – Align scan data to the global coordinate system.

- Wizard for aligning 3D scan data to an candidate design coordinate system automatically
- Highly interactive toolset for coordinate alignment
- Multiple alignment methods, such as Quick Fit, Best Fit, 3-2-1, Datum, By Ref. Coordinates, etc.

Accuracy Analyzer™ – Analyze deviation between original and modified model at any time.

- Redesign within user-defined allowable tolerances
- Automatic and real-time error visualization
- Diverse object sensitive analysis tools (mesh to mesh, mesh to CAD, etc.)

Quick Mesh-to-Surface – Generate a CAD by fitting surfaces to a mesh.

- Intelligent primitive (reference geometry) extraction from point clouds and meshes
- Automatic shrink wrapped surface model generation
- Optimized representations of original mesh data with negligible deviation error
- Interactive mesh-fit surfacing (region fit & boundary fit)

Sophisticated Curve Tools – Generate 3D curves on a mesh or 2D section curves.

- Automatic extraction of design curve and feature curve from mesh
- Intelligent dimensioning & constraining
- Comprehensive 3D curve design tools e.g. section curves
- Intelligent real-time geometry recognition
- Curvature-based curve network design

LiveScan™ & Direct Interface with 3D Scanning Devices – Direct interface with 3D scanners for real-time scanning.

- Real-time scanning
- Scan and generate design features on-the-fly
- Allows digitizing with a hard probe

3.2. Types of 3D Scan Data

3D scan data can be a point cloud or a mesh. The point cloud is a raw file type from a scanner but many scanning software can convert it to a mesh data. Both data types can be used to reach a final goal. This guide will give you detail information regarding a point cloud and mesh.

3.2.1. Understanding Point Clouds

A point cloud is a representative data type which is composed of a group of vertices in a three-dimensional coordinate system. Each vertex is defined by the X, Y, and Z coordinates; and corresponds to one position on the surface of an object. While point clouds can be directly visualized on a PC, they are not generally directly usable in most 3D applications. Point clouds usually need to be converted into mesh models or CAD models through processes referred to as mesh modeling, or reverse designing.



Point Cloud by Data Type

The file type that a scanner creates will differ based on the type of scanner used. The following are the different classifications for internal structures of point clouds.

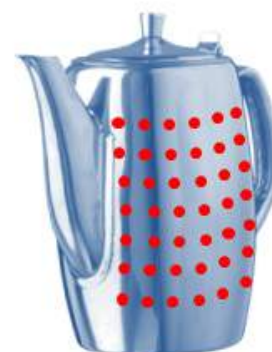
Random-type point cloud

This type contains only random position information (no relationship between points), and basic information such as color and normal can be added. Generally, neutral data (ascii format) or CMM data belong to this group type.



Grid-type point cloud

As measured points are extracted from a scanner's grid pattern, points will also be extracted from the corners of the grid. Points are marked with an X-Y index and each index has a depth to the scanning direction. This type of scan data can be a 2.5D image (image + distance on each pixel) using the projection direction and the scanning position. Generally, structured light and long range scanner produce this type of point cloud. A grid type point cloud can easily be converted into a mesh by projecting onto a planar, cylindrical, or spherical coordinate used with



the scanner.

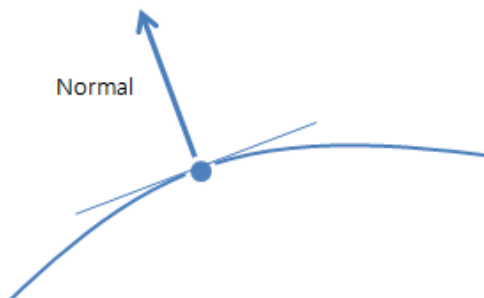
Line-type point cloud

Generally, an arm-type device with a laser scanner or a hand held scanner produces a line type point cloud. The device emits dozens of lines of lasers every second and point clouds are gathered from the scanning trace on the surface of an object. The point density will be different depending on the line direction and the scanning speed, therefore the post processing to generate a mesh from the point cloud can be more difficult.



Point Cloud with Normal

The normal is a perpendicular vector to a plane. In the case of a rounded surface, the normal is a perpendicular vector to a tangential plane on a position of the rounded surface.



If a point cloud has normal information, it can be very helpful for visualizing a model. Normal information is used for shading a model by analyzing the incidence angles and reflection angles of light on a point. Normal information is also used to determine the front or back of a surface. This is very important to accurately find and calculate face pairs between scan data and CAD data, especially in the inspection software. The following image shows a shaded point cloud in the software.



Display of Point Cloud with Normal Information

If a point cloud does not include normal information, the scan data will only be a group of points displayed in a single color. Recognizing features and working with such scan data can be difficult and tedious. The following image shows a point cloud without normal information.



Display of Point Cloud without Normal Information

A point cloud without normal information also can be visualized by the “depth shading” method, but the effect is not as effective as a point cloud with normal information. The closer points are displayed in black and the points farther away are display in gray in relation from the viewing direction. This method cannot determine the front or back of a surface but is still more helpful for feature recognition than displaying with no shading at all. The following image shows the depth shading method on a point cloud which does not have normal information.



Display of Point Cloud without Normal Information - Depth Shading

Point Cloud with Textures

A point cloud has not only three-dimensional coordinates, but also R, G, and B color information for each point. Geomagic uses this color information as a texture. A point cloud with color information can be used to generate realistic visuals and help users to recognize complex features. A textured point cloud is very efficient and easy to use for a continuous work flow.



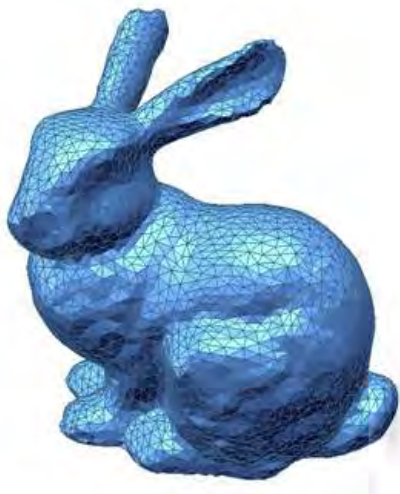
Point Cloud with Textures



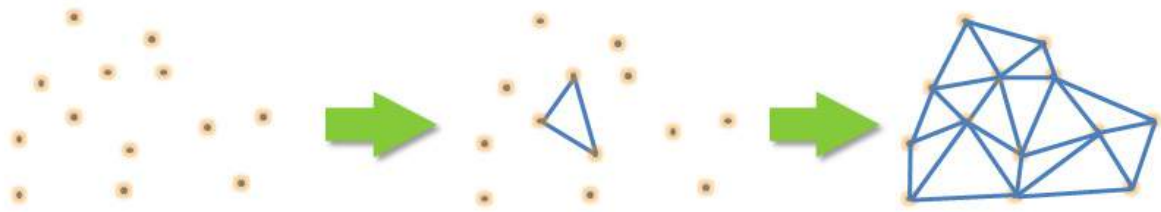
Point Cloud without Textures

3.2.2. Understanding Mesh

A mesh is polyhedron-based 3D digital data which consists of points, edges, and faces (usually triangles). The mesh is frequently used in CAD/CAM/CAE and 3D computer graphics programs. A 3D scanning application generally uses a mesh which is created from a scanned point cloud.



Modern graphic cards are not optimized for rendering point clouds but have advanced technology to display meshes. Therefore meshes often more advantages in regards to a smooth visualization of complex surfaces and structures of an object. The triangulation, or meshing, is a process to connect 3 points and construct surface. When connecting points, the edge length can be an important parameter to make a correct object shape.

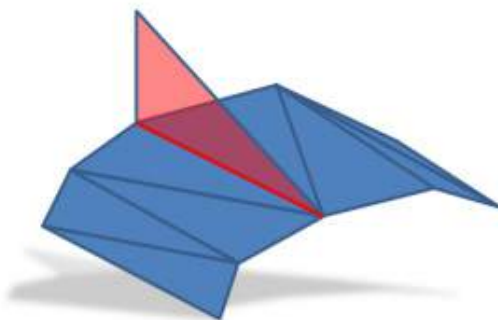


What is a Perfect Mesh?

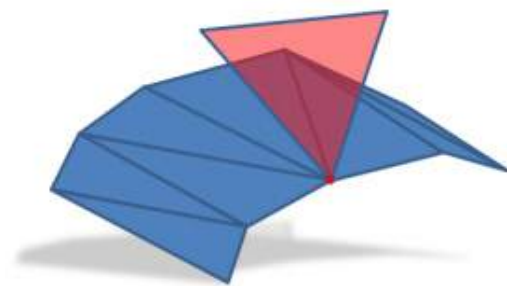
Erroneous Data

A mesh represents a surface by successively connected triangles. However, erroneous triangles such as non-manifold, redundant, crossing, and reversed triangles cannot make a complete mesh. These errors should be removed because they will affect the downstream workflow, i.e. RP, NC machining, FEM (finite element method), and fitting surfacing for reverse design. The following examples show errors on a mesh.

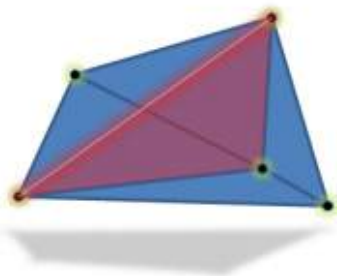
- Non-manifold Triangle – a single edge shared by three or more triangles
- Redundant Triangle – ignoring boundaries, a point must have an identical number of triangles and connecting edges
- Crossing Triangle – edges sharing a poly-vertex intersection with one another
- Reversed Triangle – opposite normal direction to neighboring triangles



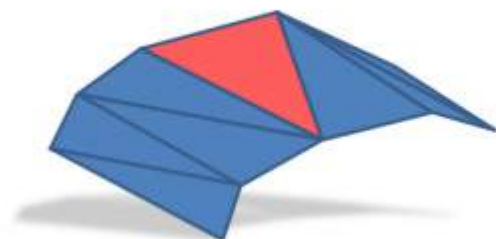
Non-manifold Triangle



Redundant Triangle



Crossing Triangle

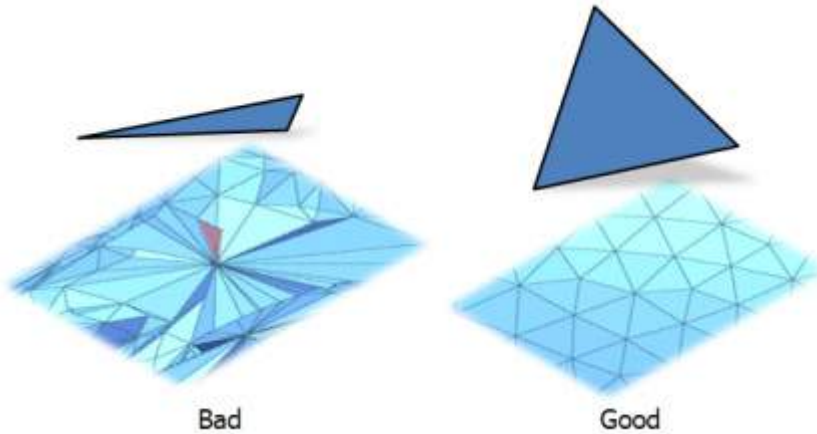


Reversed Triangle

Proper Shape of Triangle

The shape of a triangle is important for quality results. The aspect ratio, or the width to height on

a single triangle, should not be large. A triangle with a high aspect ratio will appear as a very thin and sharp triangle, and can produce errors when using a model for RP, NC machining, FEM and fitting surfacing. The ideal shape for a triangle is an equilateral triangle.



Noise on Mesh

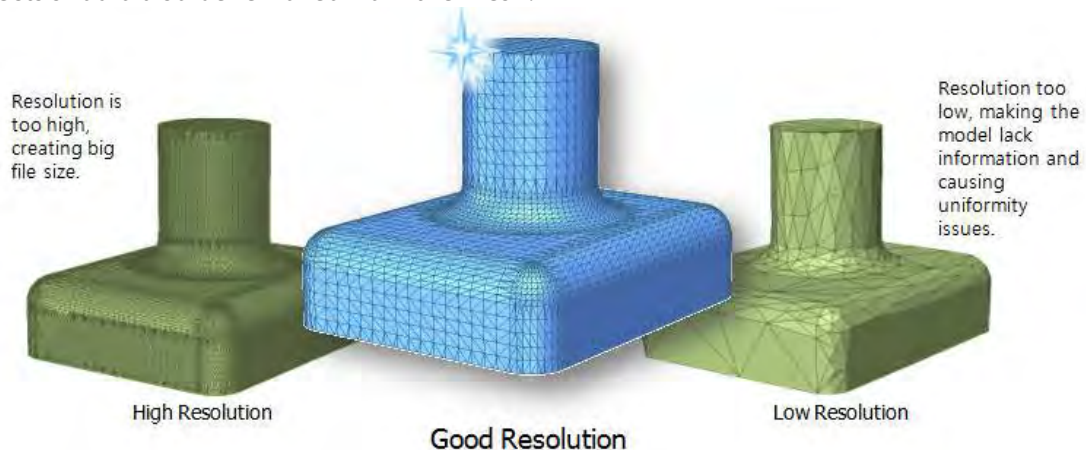
The scan data may have noise produced by a scanner. For example when you scan a very shiny object, the reflection generates noise. The noisy data generate a rough surface on the mesh. These noises can be reduced by the smoothing algorithm in the software, that removes spiky points by averaging with surrounding points. However, strong smoothness may remove sharp edges and change feature accuracy.



Removing noise by smoothing

Mesh and Resolution

A high resolution mesh generates a large file size and using it in software is not easy. Conversely, a low resolution mesh is easy to use in software but cannot adequately represent the feature information of an object. An ideal mesh should sufficiently express features with the minimum amount of triangles within a desired tolerance. A high curvature area will usually have a dense triangles structure and a low curvature area will usually have a sparse triangle structure. Noise and defects should also be removed from the mesh.



Resolution is too low, making the model lack information and causing uniformity issues.

Is an ideal mesh always necessary?

Is an ideal mesh always necessary? Not always. Depending on what the end goal is for mesh, the modeling result can be different. For example, if the purpose of modeling is for RP, NC machining and fitting surfaces; holes and twisted, erroneous triangles should be removed. If the purpose of modeling is for FEM, uniform triangles are ideal. If parametric modeling or inspection is the goal, obtaining a mesh from raw data with as few deformities is ideal, and significant features should be present on the mesh.

Mesh and Texture

As a point cloud can include texture information, a mesh can also include texture information. In the following image, the left example shows a mesh without texture and the right example shows a mesh with texture. The image illustrates how a mesh with color information can generate realistic visuals.



Mesh only and colored mesh

There are two methods to express colors on a mesh. The first one is to use an image file (jpg, bmp, etc) since each point has mapping information on a bit map, allowing color to be smoothly filled between points. The second method is using the R, G and B color values of points or triangles. In the case of a dense mesh, it can be sufficient for visualization, but for sparse areas color information will be missing and the displayed output will not look natural. If a mesh has texture information, it can be converted from an image to a point color, and vice versa.

What Can Generate a Mesh?

A mesh can be converted from scan data by the triangulation process and is commonly used in CAD, CAM, CAE and computer graphics. The following software can generate a mesh.

- Scan bundle software can often generate a mesh from a point cloud.
- Special scan data software, such as Geomagic Design X, can generate and modify a mesh.
- CAD, CAM, CAE software can generate mesh such as .stl format file.
- 3D computer graphics software can generate and do mesh modeling.

3.2.3. Mesh vs. Point Cloud

Mesh

Scan data is basically a collection of 3D points (point cloud). It can be converted into a mesh by connecting adjacent points with triangles, which are the simplest unit of a face. Generally mesh has the following advantages over using a point cloud:

- Can recognize features intuitively
- Can distinguish front and back faces
- Can measure the curvature of a surface
- Can apply environment mapping
- Can measure volume
- Can easily extract neighboring point and triangle information
- Can accurately calculate alignment between scan data
- Can be directly used in RP, FEM, and NC machining
- Can be used for reverse design
- Can be used for metrology or inspection
- Can be used for visualization and documentation
- Can be used for animation



Mesh for a complex model

Point Clouds

A point cloud can contain additional normal or color information for each point or the information can be omitted. Constructing a shape from a lower resolution point cloud may be difficult due to the lack of data, however using a high resolution point cloud can actually aide in the visualization of an object in comparison to using mesh. If a point cloud is converted to mesh, the advantages of

a mesh can be used. However long range scan data is usually a very large file size and triangulation can be very difficult. The processing time may be lengthy, and the results can be difficult to predict. If the purpose of using long range scan data is for CAD modeling, it is more efficient to do parametric CAD modeling with a point cloud instead of mesh or partially convert a point cloud into a mesh. The benefits of using point clouds are as follows:

- Can handle larger data files
- Can do simple measuring
- Can be used for reverse design
- Can be used for surveying
- Can be used for metrology or inspection
- Can be used for visualization and documentation



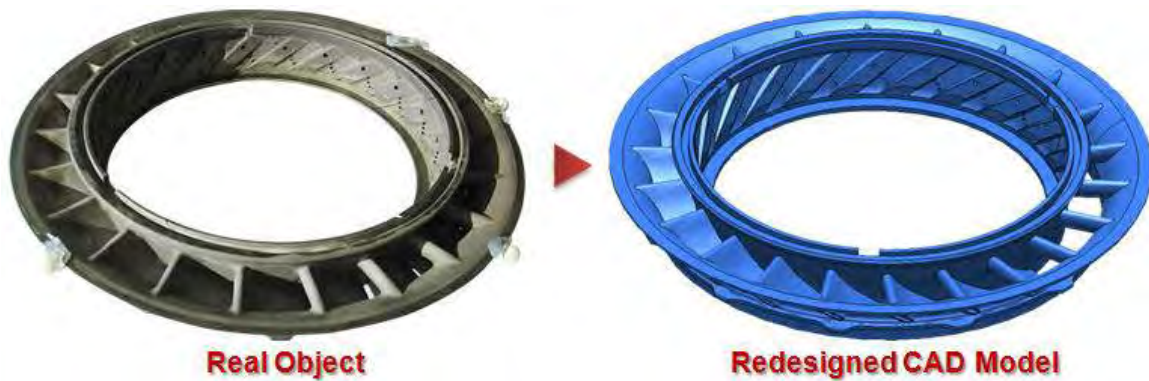
Visualization of Point Cloud

4. Understanding Reverse Design

4.1. What is Reverse Design?

Reverse Design is the technology of creating digitized design data to be used for the creation of products from a real object as shown in the image below.

The Reverse Design shown below was internally done by leading technology manufacturers as a unique design technique. The traditional design process is the creation of a final product through a chosen design process, such as concept designing or detailed designing used to create "something" from "nothing". The Reverse Design process can create valuable design data by capturing digitized shape information from a real object by using 3D scanning technologies, such as CMMs (Coordinate-Measuring Machine), laser scanners, structured light digitizers, or Industrial CT Scanning (computed tomography). Design intent can be extracted from shape information.



Reverse Design from a Real Object

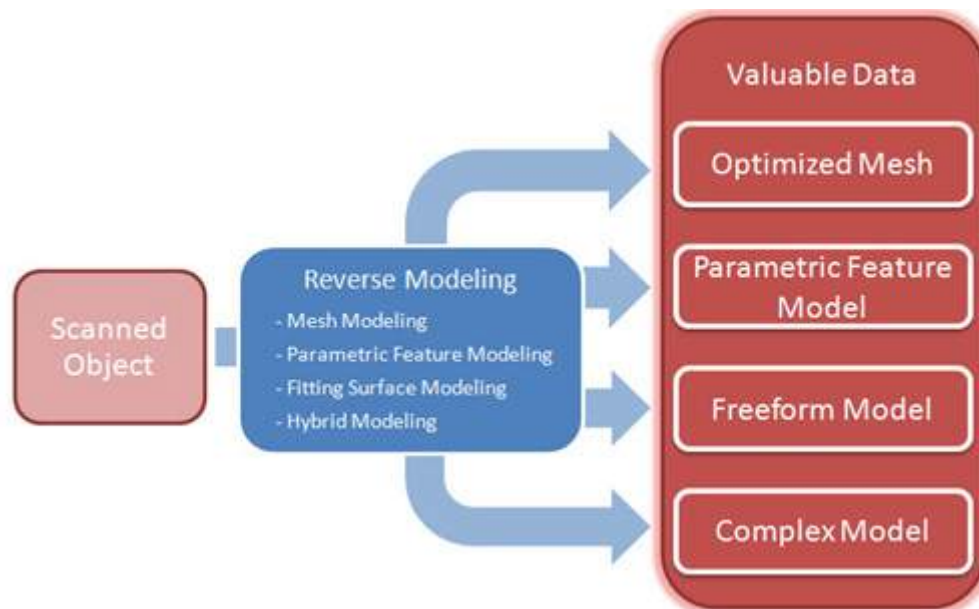
Reverse Designing is widely used in manufacturing fields such as automotive, aerospace, medical, etc., as well as graphics fields such as game design, animation, computer graphics, etc. In the field of manufacturing, reverse design is being effectively applied to product design, development, production, and quality management, and has seen a key role in reducing overall product development cycle times.

Reverse Design is used in:

- Duplicating a target product for which no drawings or CAD data exists
- Manufacturing of a new product with enhanced functions based on an existing product
- Designing a new product from a mock-up model or clay model in the automotive industry, where aesthetic design factors are critical
- Customizing products fitted for the human body (Mass Customization)
- Reconstructing damaged cultural assets
- Using digitized shape information for animation
- Manufacturing a products in the medical or dental fields
- Analyzing design elements from competitors' products

4.2. Reverse Modeling Strategy

The Reverse Modeling process is the process of creating the best 3D model from the 3D scan data (Point Cloud or Mesh) generated in the Scan Data Processing phase.



Various modeling methods in Reverse Modeling process

These methods can be used solely or mixed with the other methods depending on the desired output.

Mesh Modeling

Purpose

Creating an optimized mesh from 3D scan data

Benefit

You can create the best mesh from 3D scan data by cleaning defects, filling missing holes, modifying its shape, and optimizing mesh structure, according to the purpose of utilizing.

What is it useful for

- Analyzing the model shape
- Creating a reference prototype model by using Rapid Prototyping technologies

Parametric Feature Modeling

Purpose

Creating a 3D feature model based on the analyzed design intent and elements from 3D scan data

Benefit

You can define features with parameters based on the analyzed design intent and elements from 3D scanned object. The created features can be reused, redefined, modified, and translated by controlling parameters.

What is it useful for

- Manufacturing a new product with enhanced functions based on existing product
- Duplicating the target product which does not have a CAD model
- Analyzing design elements of another part or product

Fitting Surface Modeling

Purpose

Creating a 3D freeform model based on the optimized mesh from 3D scan data

Benefit

You can effectively and rapidly extract an accurate freeform surface from the shape of a 3D scanned object, by using various tools; such as Auto Surfacing, Mesh Fitting, and Boundary Fitting.

What is it useful for

- Creating outer forms of aesthetic model, such as car bodies and outer cover of consumer products
- Customizing a new product to be fitted for the human body (Mass Customization)
- Manufacturing a new product based on the human body shape in the medical or dental fields
- Reconstructing the damaged cultural assets
- Creating animations from digitized shapes

Hybrid Modeling

Purpose

Creating a complex 3D model from 3D scan data

Benefit

You can rapidly create a complex 3D feature model by mixing parametric features created by using Parametric Feature Modeling method and freeform surfaces created by using Fitting Surface Modeling method. The resulting 3D feature model can be reused, redefined, modified, and translated by controlling parameters maintaining its outer freeform shape.

What is it useful for

- Designing a new product from a mock-up or clay model
- Designing a new product for assembly

4.2.1. Mesh Modeling

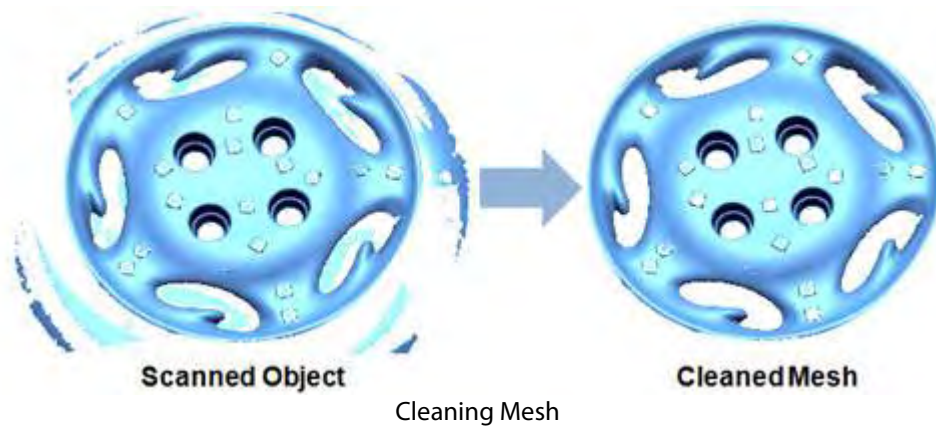
The Mesh Modeling method can be used to create an optimized mesh from a 3D scan of an object.

When can I use this modeling method?

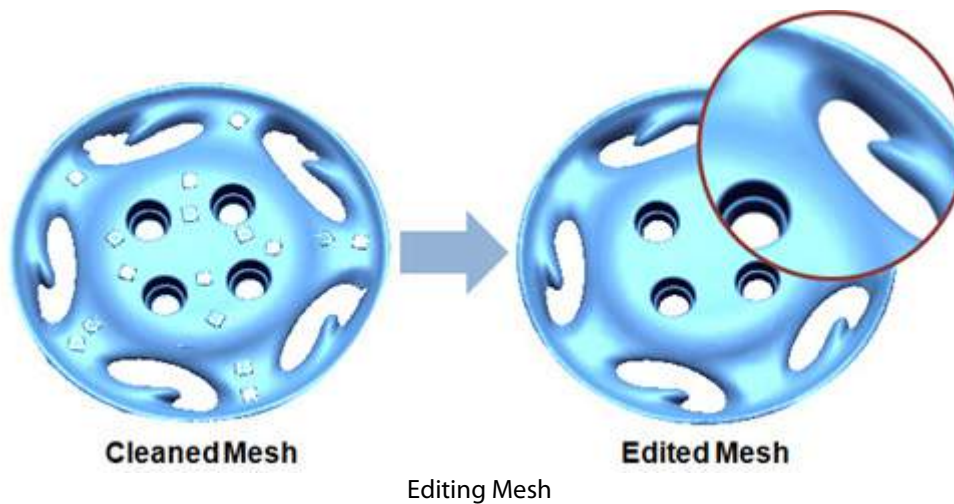
The Mesh Modeling method can be used for analyzing the shape of a model and creating a reference prototype model with Rapid Prototyping technologies. It can also be used to optimize mesh for use as a graphic model in animations, or for use as raw data in creating 3D feature models.

The tools that are used for creating optimized meshes from 3D scan data can be divided into 4 different types, as listed below:

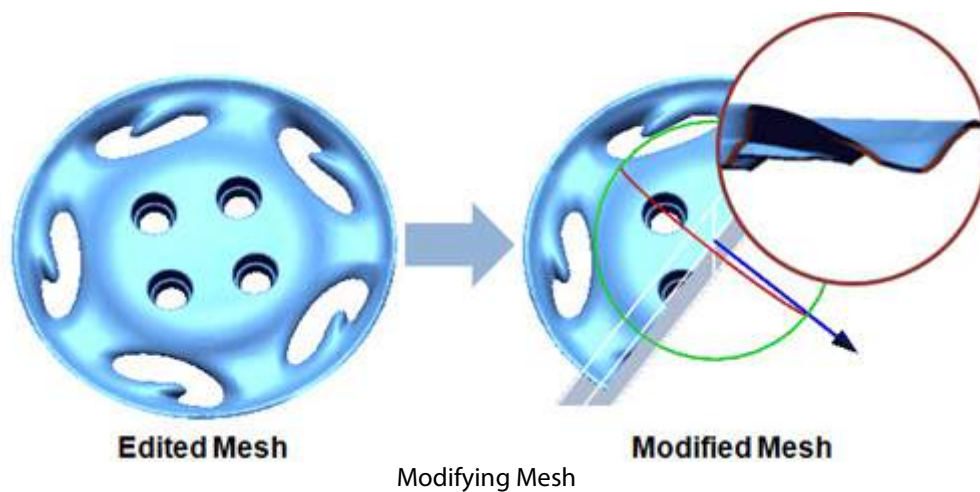
Cleaning Tool – Used for deleting data and healing defects (abnormal poly-faces) from 3D scan data (Point Cloud or Mesh)



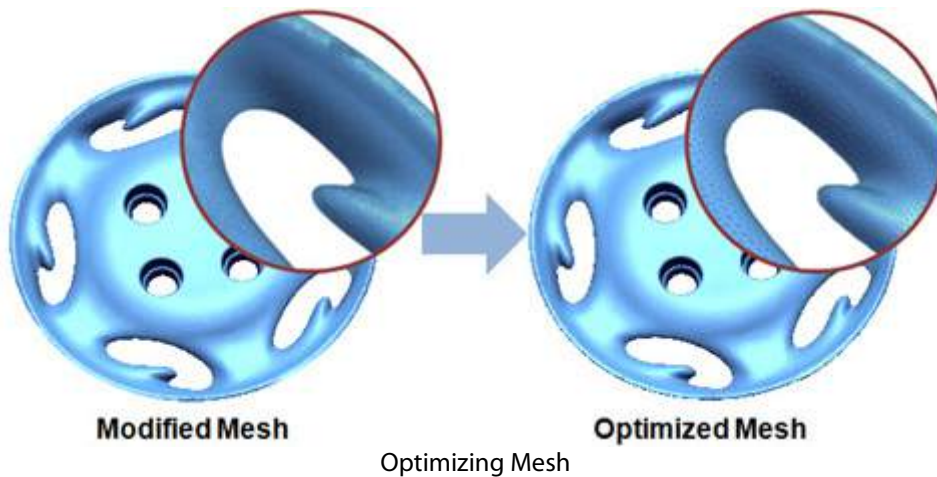
Editing Tool – Used for filling missing holes that could be generated by shadows from other features, to smooth the roughness of a mesh, or to re-arrange its boundaries.



Modifying Tool – Can be used to add thickness to a model or to modify its shape (like split, mirror, merge, subtract, intersect, etc.)



Optimizing Tool – Used to optimize the topology of faces on a mesh or to create a watertight mesh.



These tools can be used by themselves or in conjunction with other tools to optimize mesh based on the desired outcome during the Mesh Modeling process.

4.2.2. Parametric Feature Modeling

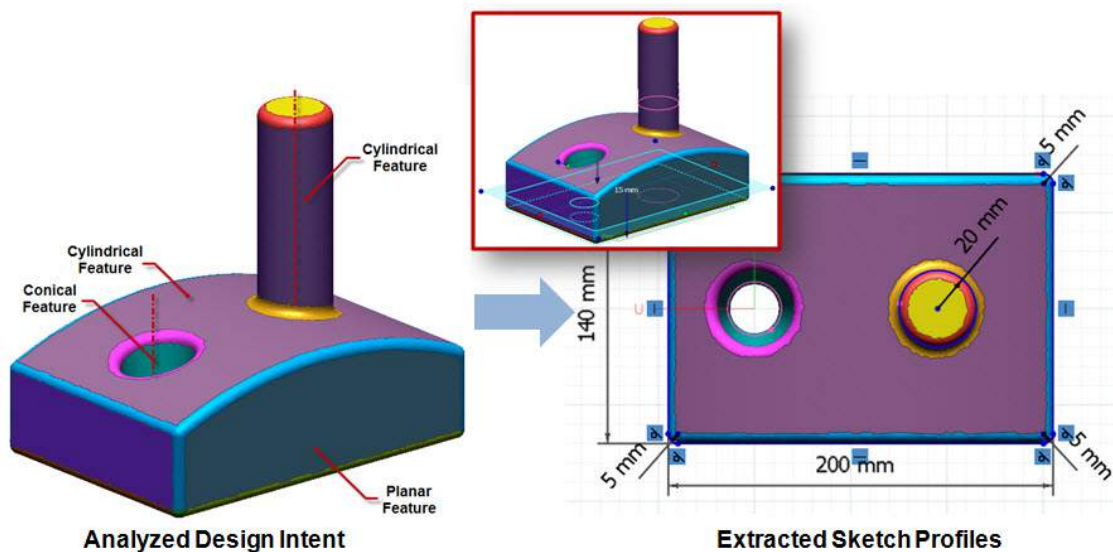
The Parametric Feature Modeling method can be used to create 3D feature models based on design intent and elements from 3D scanned objects.

When is this modeling method useful?

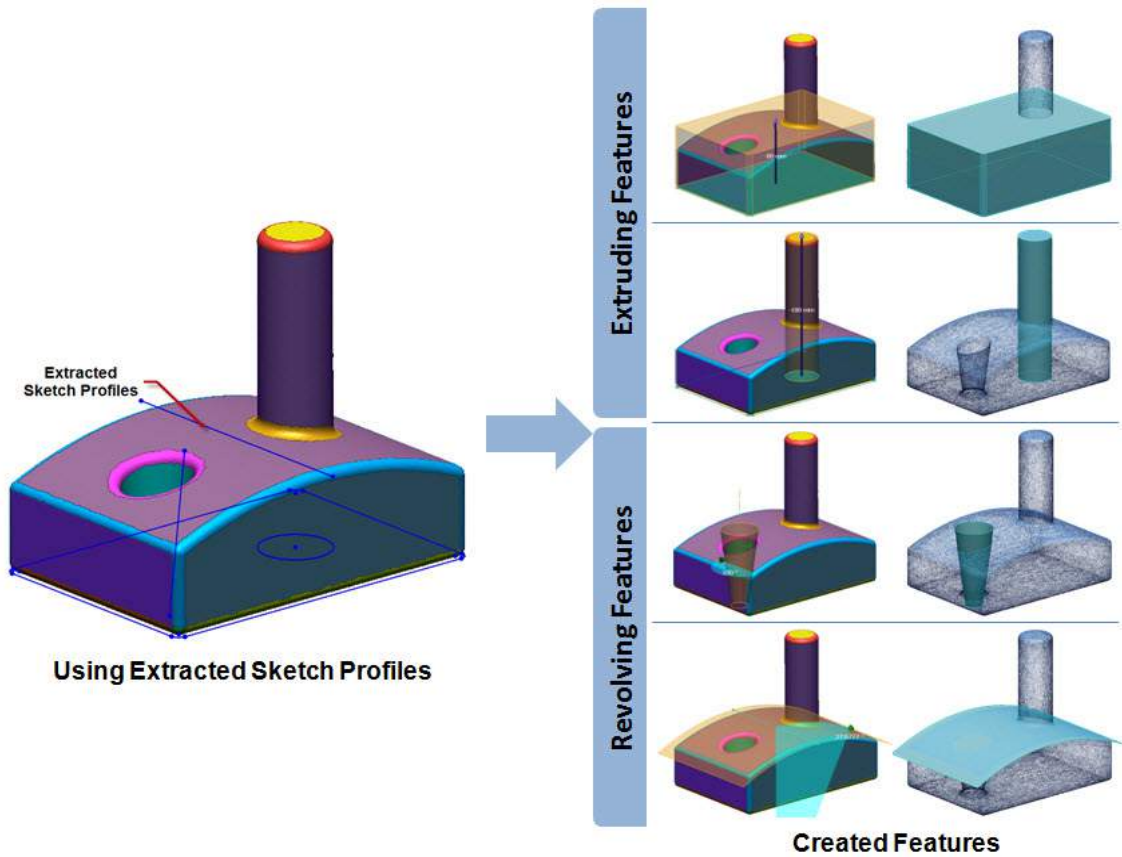
The Parametric Feature Modeling method based on Parametric Modeling Technology (see: Note) is used for manufacturing a new product with enhanced functions based on an existing product. It can also be used to duplicate a target product which does not have a drawing.

This method consists of 3 different sub-methods for creating a 3D feature model. The methods are:

2D/3D Sketching method – Extracts 2D section profiles or 3D curves from 3D scan data.

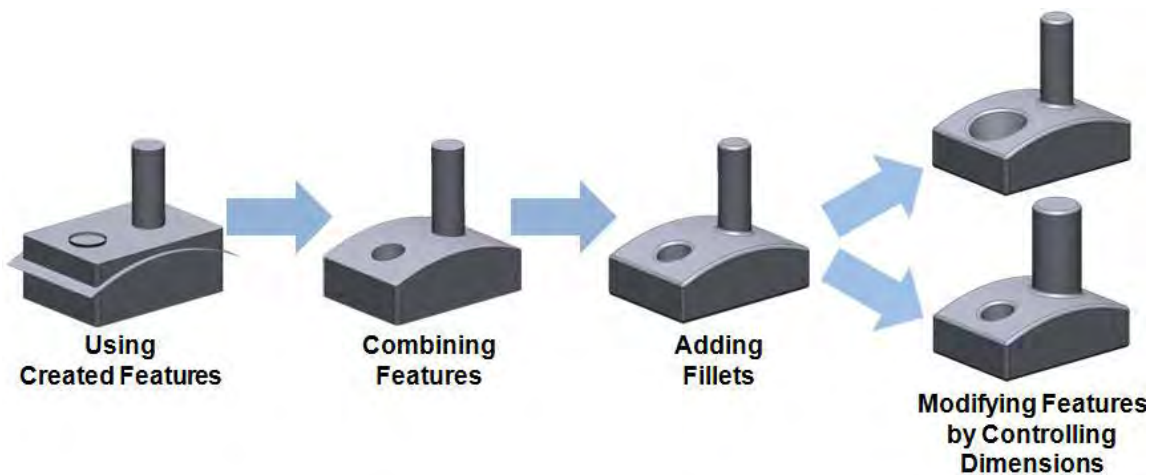


Feature Modeling method – Creates a 3D CAD-ready feature model by using sketch profiles.



Creating Features by using the Extracted Sketch Profiles

Editing method – Combines created features or subtracts features by using geometric and mathematic operations, and modifies the created features by controlling the defined dimensions.



Completing a 3D CAD feature model by combining the created features

Note:

Parametric Modeling Technology

Parametric Modeling Technology is based on design intent, and is a unique technique in the Reverse Design process. It sets the parameters required for defining features based on the design intent and creates 3D features using those parameters.

Design data can be prepared for a final product by modifying design iterations, trying out, and reviewing the results throughout the design process. The design data is managed in the same database throughout the product development process from the manufacturing stage through the analyzing stage; and finally ending with the evaluation stage. Design data modeled with Parametric Modeling Technology can be easily modified and reviewed according to the results of the product development process. The features managed by the parameters in the design data can be reused, redefined, modified, translated, and updated by controlling the parameters for each one. With parametric Modeling Technology, development time and cost can be effectively reduced in the creation of a product.

4.2.3. Fitting Surface Modeling

The Fitting Surface Modeling method can be used for creating a 3D freeform model based on an optimized mesh from a 3D scanned object.

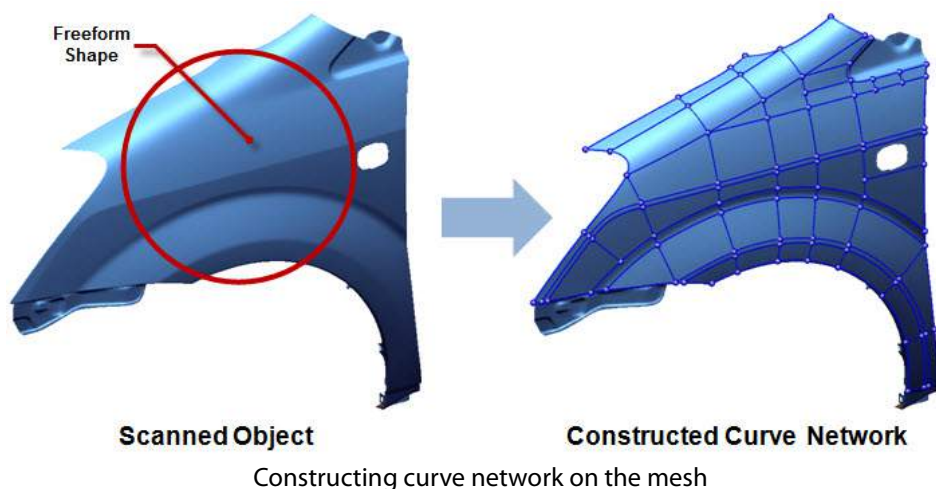
When can I use this modeling method?

This Fitting Surface Modeling method with Surface Fitting Technology (See Note 1.) is mainly used for quickly and effectively creating outer forms of aesthetic models, such as car bodies and outer covers of consumer products. It is also used for customizing new products to be fitted for the human body (Mass Customization), manufacturing new products based on the human body shape in the medical or dental fields, reconstructing damaged cultural assets, using digitized shape information for animations, etc.

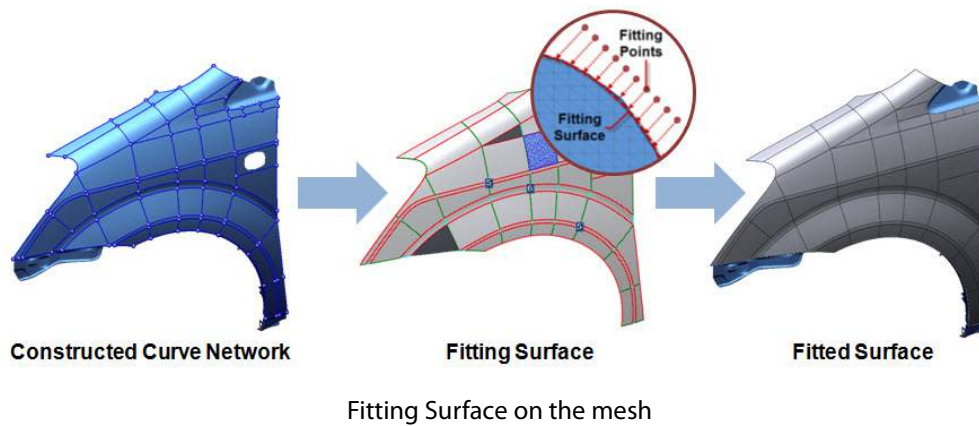
The Fitting Surface Method consists of 3 different sub-methods.

The 3 sub-methods are:

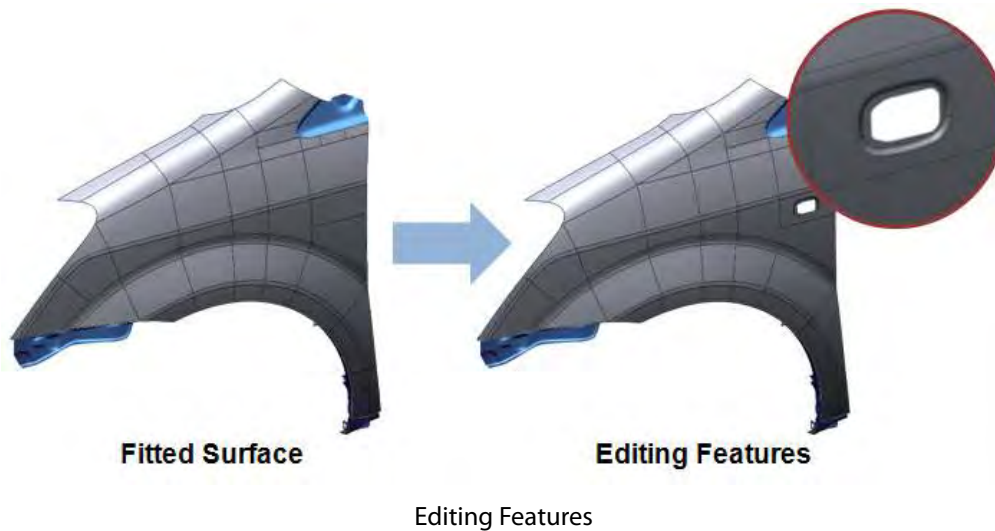
Curve Networking method – Constructs a curve network by connecting 3D curve loops projected onto freeform regions on a mesh.



Surface Fitting method – Fits surface patches within a constructed curve network and creates a freeform surface body by connecting the surface patches.



Editing method – combines created surface bodies or subtracts surface bodies by using geometric and mathematic operations.



Note:

Surface Fitting Technology

Surface Fitting Technology is a unique technique in the Reverse Design process that provides an effective way to easily and quickly create 3D freeform surface bodies from a freeform mesh shape. It creates surface patches by projecting uniform points within curve loops constructed on a freeform mesh shape, and fitting to the projected points. A 3D freeform surface body is created by connecting fitted surface patches. This technique is usually used for creating a highly accurate 3D freeform surface body from 3D scan data.

4.2.4. Hybrid Modeling

The Hybrid Modeling method is used to create complex 3D models based on the design intent and

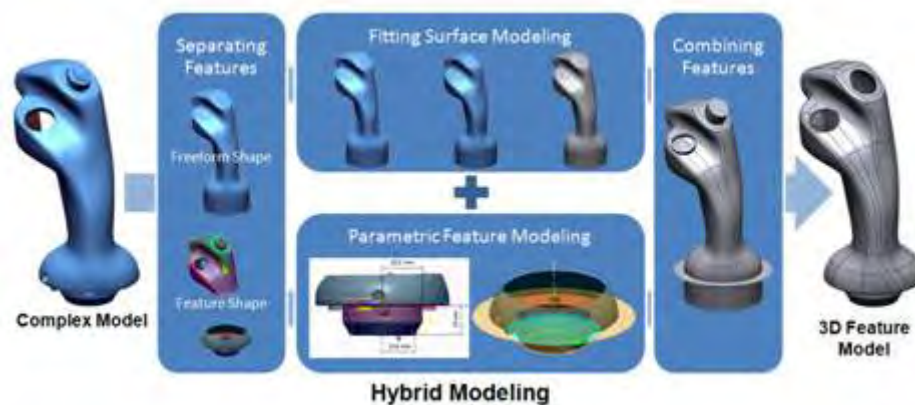
elements of a 3D scanned object or 3D mesh shape.

When can I use this modeling method?

The Hybrid Modeling method mixes with both Parametric Modeling Technology and Surface Fitting Technology; and can be used for managing overall feature shape information as well as design parameters from complex models. It can also be used when designing new products for assembly or designing new products from mock-up models or clay models, especially in the automotive industry or other industries where aesthetic design factors are critical.

This method consists of 5 different sub-methods. The 5 sub-methods are:

- 2D/3D Sketching method: Extracts 2D section profiles or 3D curves from the 3D scan data.
- Feature Modeling method: Creates a 3D CAD feature model by using the sketch profiles.
- Curve Networking method: Constructs a curve network by connecting 3D curve loops projected on the freeform region in the mesh.
- Surface Fitting method: Fits surface patches within a constructed curve network and creates a freeform surface body by connecting the surface patches.
- Editing method: Combines the created features or subtracts features, using geometric and mathematic operations.



Creating a complex 3D Model by using the Hybrid Modeling method

4.3. Powerful Tools for Reverse Modeling

Geomagic Design X provides various tools to quickly and effectively create the best 3D parametric CAD model from the 3D scan data.

These tools are unique, innovative, and intelligent; and provide the best reverse design process. You can automatically extract design intent; process parameters, and geometric information from the 3D scan data by using these tools, and quickly create a 3D parametric feature model. Furthermore, the created 3D parametric model with history and design parameters can be transferred to the other CAD systems, and it can be managed in the CAD system.

The following tools are innovative and intelligent tools provided in Geomagic Design X:

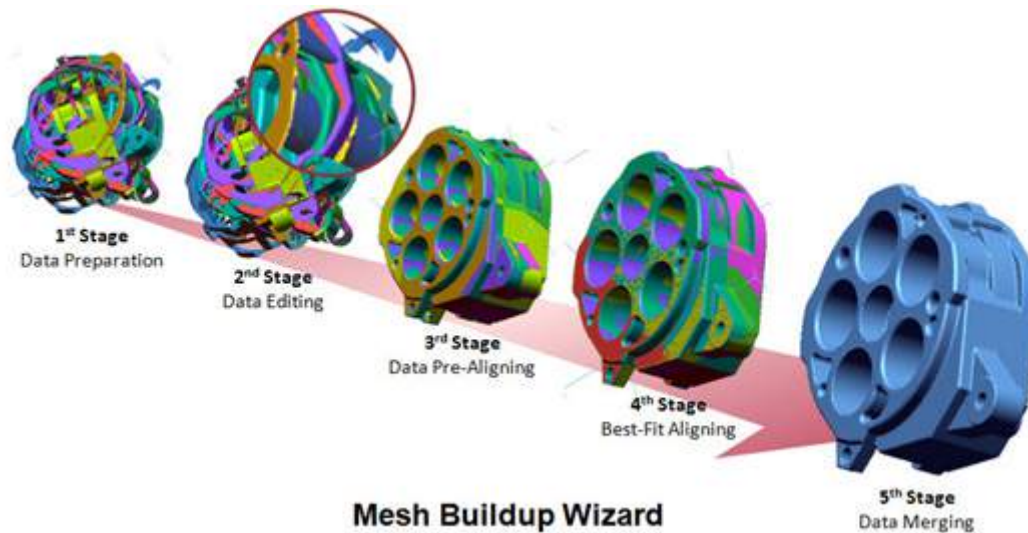
- Automatic scan data processing tool for creating a watertight mesh from raw 3D scan data - *Mesh Buildup Wizard™*
- Automatic coordinate alignment tool for CAD modeling from 3D scan data - *Align Wizard™*
- Innovative and intelligent tool for extracting design parameters and elements from 3D scan data - *Redesign Assistants™*
- Innovative and intelligent tool for extracting 2D/3D sketch profiles and feature information from 3D scan data - *Modeling Wizard™*
- Real-time analysis tool for creating the best 3D feature model from 3D scan data within the design tolerance - *Accuracy Analyzer™*
- Automatic process for transferring 3D parametric models with the modeling history and design parameters to other CAD systems - *LiveTransfer™*

4.3.1. Mesh Buildup Wizard(TM)

The Mesh Buildup Wizard is used in the Scan Data Processing phase, and is a wizard style interface for creating defect-free and watertight mesh models from raw 3D scan data.

The Mesh Buildup Wizard tool employs the most advanced mesh technology; but is user-friendly, so even a novice user can easily create complete mesh from 3D scan data. This tool consists of 5 stages. Each stage can be executed in just few clicks, enabling the speedy creation of optimal mesh for use in the final reverse design stages. The 5 stages are:

- **Data Preparation stage:** Analyzes the state of scan data and defining the next process according to the type of target scan data
- **Data Editing stage:** Remove noisy clusters and unnecessary point clouds or meshes
- **Data Pre-Aligning stage:** Aligns multiple scan data files quickly.
- **Best-Fit Aligning stage:** Aligns multiple scan data files by using geometry shape information.
- **Data Merging stage:** Merges scan data and creates an optimized mesh



Creating optimal mesh through each stage in the Mesh Buildup Wizard

When can I use this tool?

When creating an optimized mesh from multiple 3D scan data files that have not been aligned, optimal mesh for the final reverse design process can be made by using this tool. An optimal mesh can be created by applying this tool after importing multiple 3D scan data files and clicking the "Run Mesh Buildup Wizard" button.

Where is this tool?

- [Insert > Import](#) (By clicking "Run Mesh Buildup Wizard" button)
- [Tools > Scan Tools > Mesh Buildup Wizard](#)

How do I use this tool?

Refer to the following topic below:

[Tools > Scan Tools > Mesh Buildup Wizard](#)

4.3.2. Auto Surfacing

Auto Surfacing is used in the Modeling phase and is an innovative and automatic tool for fitting surface patches onto a mesh and creating a surface body. This is important when designing a complex freeform part, creating model data for manufacturing a freeform shape, or for analyzing a geometric shape. This tool creates a surface body that can envelope an entire geometric shape of a target mesh.

The Auto Surfacing tool provides 2 different methods:

- **Evenly Distributed Network** - Generates a surface body from a target mesh by projecting an evenly distributed curve network
- **Feature Following Network** - Generates a surface body from a target mesh by constructing a feature following curve network

These two methods are used differently based on desired results and how the final model will be utilized.



Creating Surface Body by using Auto Surfacing Tool

Where is this tool?

[Insert > Surface > Auto Surfacing](#)

How do I use this tool?

Refer to the following topic below:

[Insert > Surface > Auto Surfacing](#)

4.3.3. Align Wizard(TM)

The Align Wizard is used in the Modeling phase and is a wizard style interface used to quickly and accurately align 3D meshes to a Global Coordinate system, based on design intent.

The Align Wizard tool employs the most advanced alignment technology; but is user-friendly, so even a novice user can easily align 3D mesh to a Global Coordinate system. The tool consists of 2

stages. Aligning target 3D mesh to the Global Coordinate system without manually selecting and setting each element can be done with the following 2 stages:

- **Defining Target Mesh stage:** Defines the target mesh which is not aligned to the global coordinate
- **Choosing Best Coordinate stage:** Chooses the best coordinate from the coordinate candidates sorted by analyzing geometric feature information



Aligning by using the Align Wizard tool

When can I use this tool?

If understanding a complex feature shape and manually defining a coordinate for it is difficult, easily align the 3D mesh of the target to the Global Coordinate system by using the Align Wizard tool.

Where is this tool?

[Tools > Align > Wizard](#)

How do I use this tool?

Refer to the following topic below:

[Tools > Align > Wizard](#)

4.3.4. Redesign Assistant(TM)

The Redesign Assistant is used in the Modeling and Extracting Design Intent phases, and is an innovative and intelligent tool for extracting design parameters and elements from 3D scan data. This tool helps users to easily define feature shapes with accurate parameters and design elements.

When can I use this tool?

When extracting design intent and parameters from 3D scan data, this tool is widely used within the other tools provided by Geomagic Design X in the following cases:

- Extracting Distance from 3D scan data

When extruding a defined sketch profile to create an extruded shape or subtracting a feature from other existing features, this tool makes it possible to extract distance information from the 3D scan data without the need for manual measurement.

- Extracting Radius from 3D scan data

When creating a fillet shape in a fillet region that has a constant radius or variable radius, this tool makes it possible to extract radius information from 3D scan data without needing to measure on the section.

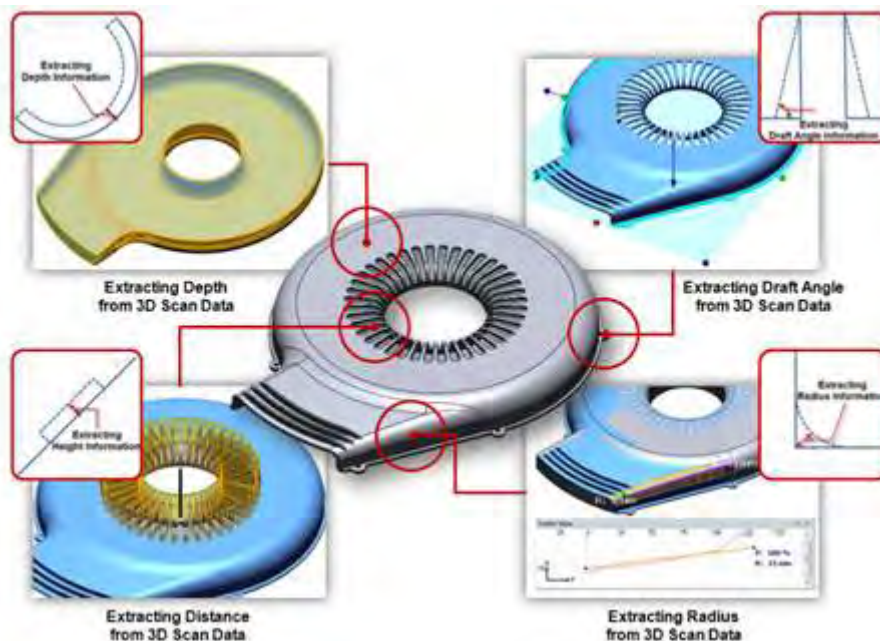
- Extracting Thickness from 3D scan data

If a model is being thickened, this tool makes it possible to extract thickness information from 3D scan data.

- Extracting Draft Angle from 3D scan data

When extracting an accurate sketch profile from the scanned object which has an amount of taper, this tool makes it possible to extract draft angle information from 3D scan data.

Furthermore, when creating a revolved shape or a spiral shape like a drill, it is possible to extract important design parameters such as height, radius, pitch, and revolving ratio as well as design elements such as section profile, 3D sweeping patch, rotational axis, etc. The extracted design parameters and elements will be used for defining an accurate 3D feature shape as reference information.



Parametric Feature Modeling by using the Redesign Assistant

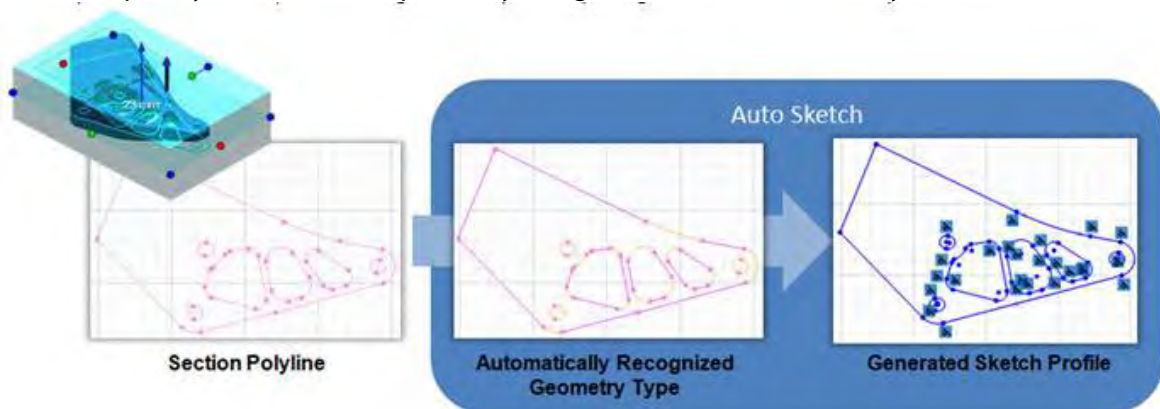
4.3.5. Auto Sketch

Auto Sketch is used in the Modeling phase and is an innovative and automatic tool for creating a sketch profile based on section polylines.

The sketching process extracts lines and arcs from complex section polylines and connects them to

create a sketch profile that may otherwise be difficult and time consuming. While creating a sketch profile, Auto Sketch intelligently recognizes a relationship between lines and arcs and automatically sets constraints to the sketch profile.

You can completely create a sketch profile by using the Auto Sketch tool only.



Creating a Sketch Profile from Section Polyline by using Auto Sketch Tool

Where is this tool?

Go to: [Tools > Sketch Entities > Auto Sketch](#)

How do I use this tool?

Refer to the following topic below:

[Tools > Sketch Entities > Auto Sketch](#)

4.3.6. Various Modeling Wizards(TM)

The Modeling Wizard is used in the Modeling phase and is an innovative and intelligent tool for extracting 2D and 3D sketch profiles and feature information from 3D scan data.

The Modeling Wizard provides 5 different wizard tools:

- **Extrusion Wizard:** Creates an extruded feature.
- **Revolution Wizard:** Creates a revolved feature.
- **Sweep Wizard:** Creates a swept feature along a 3D path curve.
- **Loft Wizard:** Creates a lofted surface by connecting 3D section curves.
- **Pipe Wizard:** Creates a pipe feature by using a section profile and 3D path curve.

Extrusion Wizard / Revolution Wizard / Sweep Wizard

Extrusion Wizard

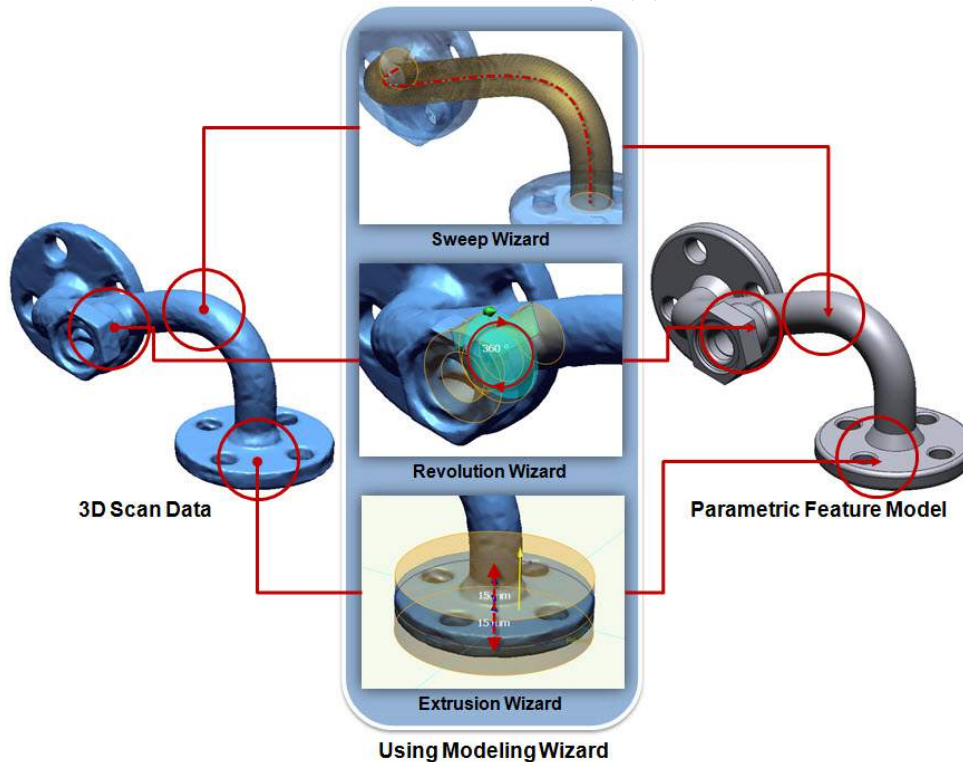
From a mesh region, the Extrusion Wizard intelligently generates an extruded solid/surface and its 2D/3D section sketch profile, while satisfying given deviation error automatically.

Revolution Wizard

From a mesh region, the Revolution Wizard intelligently generates a revolved solid/surface and its 2D/3D sketch profile and revolving axis, while satisfying given deviation error automatically.

Sweep Wizard

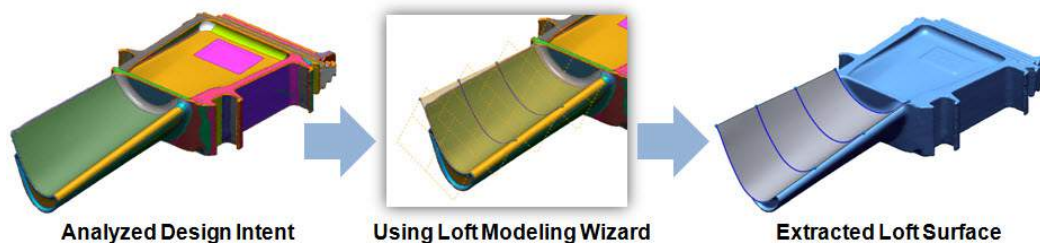
From a mesh region, the Sweep Wizard intelligently generates a swept solid/surface and its 2D/3D sketch profile and 3D path curve, while satisfying given deviation error automatically.



Feature Modeling by using the Extrusion Wizard / Revolution Wizard / Sweep Wizard

Loft Wizard

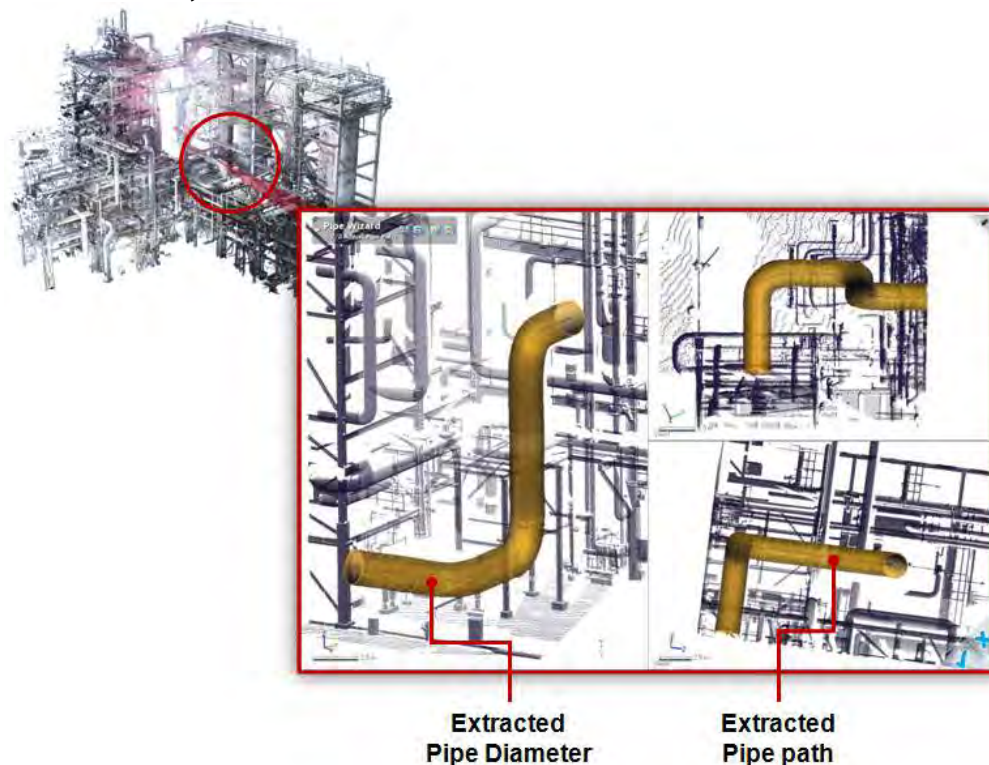
From a mesh region, the Loft Wizard intelligently generates a smooth lofted surface and its profile curves, while satisfying given deviation error automatically, as shown in the image below.



Creating a smooth lofted surface by using the Loft Wizard

Pipe Wizard

Using 3D scan data (Point Cloud or Mesh Region), the Pipe Wizard intelligently generates a swept solid/surface and its 2D/3D sketch profile and 3D path curve, while satisfying given deviation error automatically.



Creating a Pipe Feature by using the Pipe Wizard

Where is this tool?

- [Insert > Modeling Wizard > Extrusion Wizard](#)
- [Insert > Modeling Wizard > Revolution Wizard](#)
- [Insert > Modeling Wizard > Sweep Wizard](#)
- [Insert > Modeling Wizard > Loft Wizard](#)
- [Insert > Modeling Wizard > Pipe Wizard](#)

4.3.7. Accuracy Analyzer(TM)

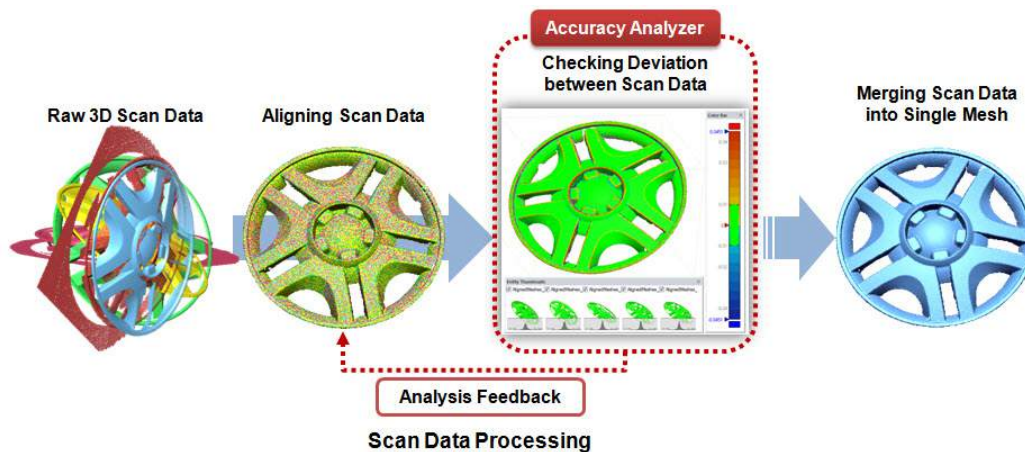
The Accuracy Analyzer is used in the Model Verification phase, and is a real-time analysis tool for creating the best 3D feature model from 3D scan data within design tolerances. This tool analyzes deviations between the intermediate or final model with the raw 3D scan data and verifies the results. And also, the quality and continuity of the faces of a model can be checked in order to ascertain whether the final model can be manufactured without any problems. The analyzed results can be used for creating an optimal 3D model within allowable design tolerances.

When can I use this?

Whenever the analysis of intermediate/final result is required, this tool can be used during the modeling process in the following cases:

Aligning between scan data

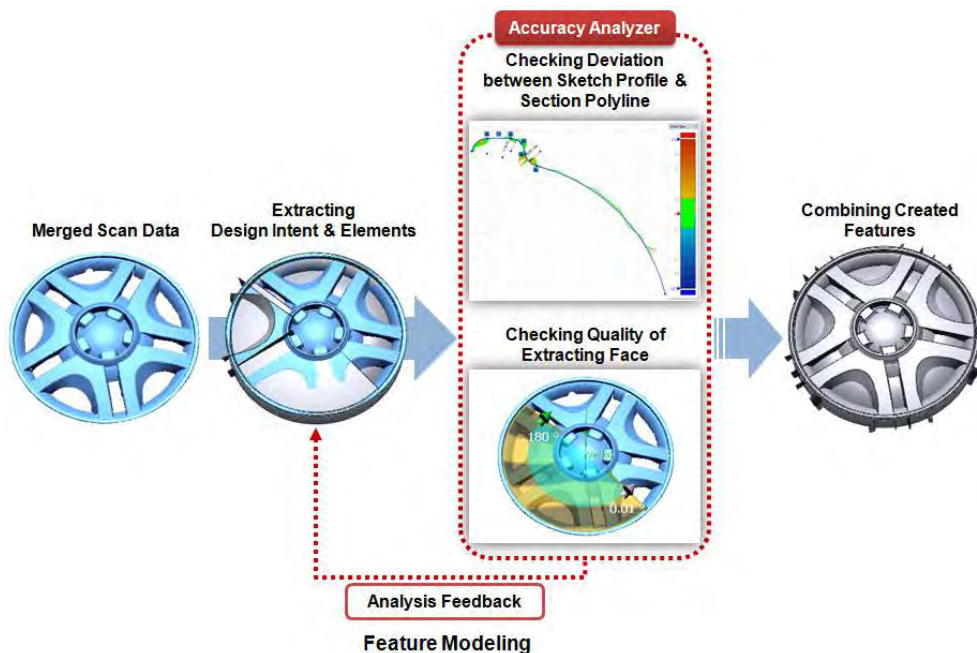
You can check the deviation between each scan after alignment is performed in the scan data processing phase, and get the best result by using the analysis feedback



Accuracy Analyzer during the scan data processing phase

Extracting design intent or elements

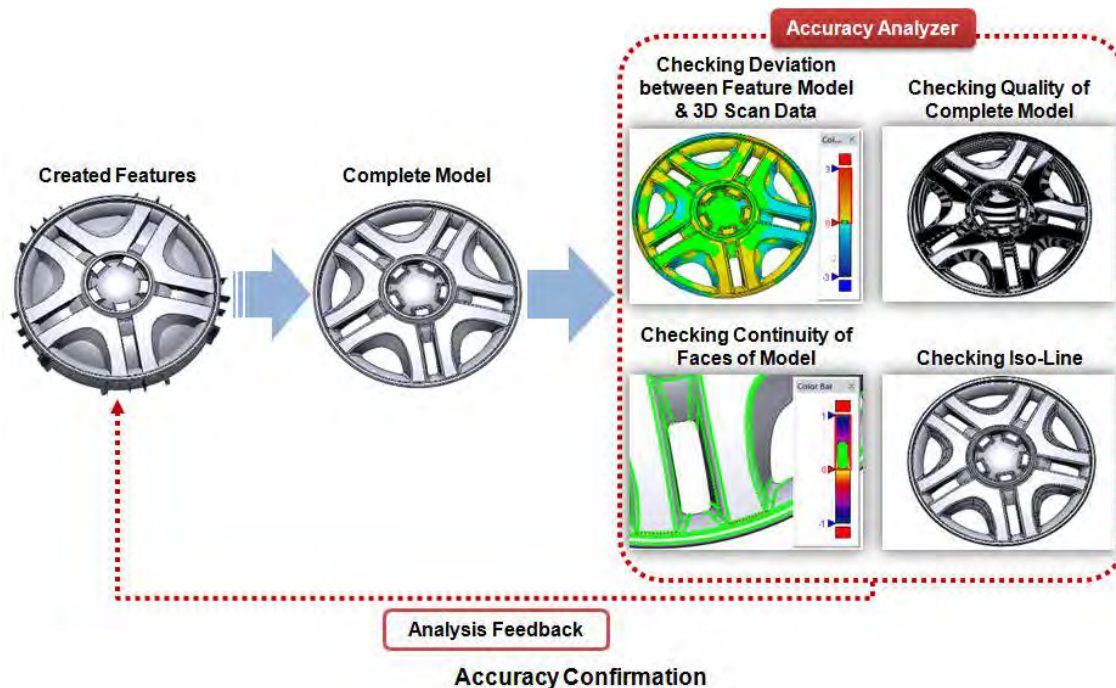
You can check how the extracted elements and feature information compare to the original scan data.



Accuracy Analyzer during the modeling phase

Confirming final modeling result

You can finally check the accuracy and quality of the model.



Accuracy Analyzer of the final result

4.3.8. LiveTransfer(TM)

LiveTransfer is an automatic tool in Geomagic for transferring 3D parametric models and their modeling history and design parameters to another CAD system. It is a unique and powerful tool and is used after completing the modeling process.

After completing data model in Geomagic Design X, the 3D parametric model, with its modeling history and design parameters can be transferred to other CAD systems, such as SolidWorks, PTC Creo (Pro/Engineer), Siemens NX, Autodesk Inventor, CATIA V4/V5, and AutoCAD. This process creates a native and parametric model in the target CAD system. Then the file can be used as normal for all downstream applications; such as manufacturing, drafting, tooling, assembly, etc.



Where is this tool?

- [File > LiveTransfer\(TM\) > To SolidWorks](#)
- [File > LiveTransfer\(TM\) > To Siemens NX](#)
- [File > LiveTransfer\(TM\) > To Creo\(Pro/E\)](#)
- [File > LiveTransfer\(TM\) > To AutoCAD](#)
- [File > LiveTransfer\(TM\) > To Inventor](#)
- [File > LiveTransfer\(TM\) > Save As CATIA V4/V5](#)

5. Getting Help

To get technical support:

If you have any problems or suggestions, you can contact support@rapidform.com.

To access FAQ, Technical Tips, and other resources:

Go to the **Support** tab.

To access the online help documentation:

- While in that operation, press **F1**. The documentation will be displayed in the **Help** tab of the viewing window.
- Look at the .chm file that contains a search function. It is available in the Help subdirectory of a root directory.
- Also a PDF version of the documentation is available in the User Guide subdirectory of the root directory.

To check the version of the application and the Product ID:

- Click Help > About
- When you send your request to technical support, include this information.

To view the Quick Start Guide

Click **Help** > **Quick Start Guide**.

To activate a Product license:

Click **Help** > **Activate License**.

To Deactivate a Product license:

Click **Help** > **Deactivate License**.

To view the performance of a system

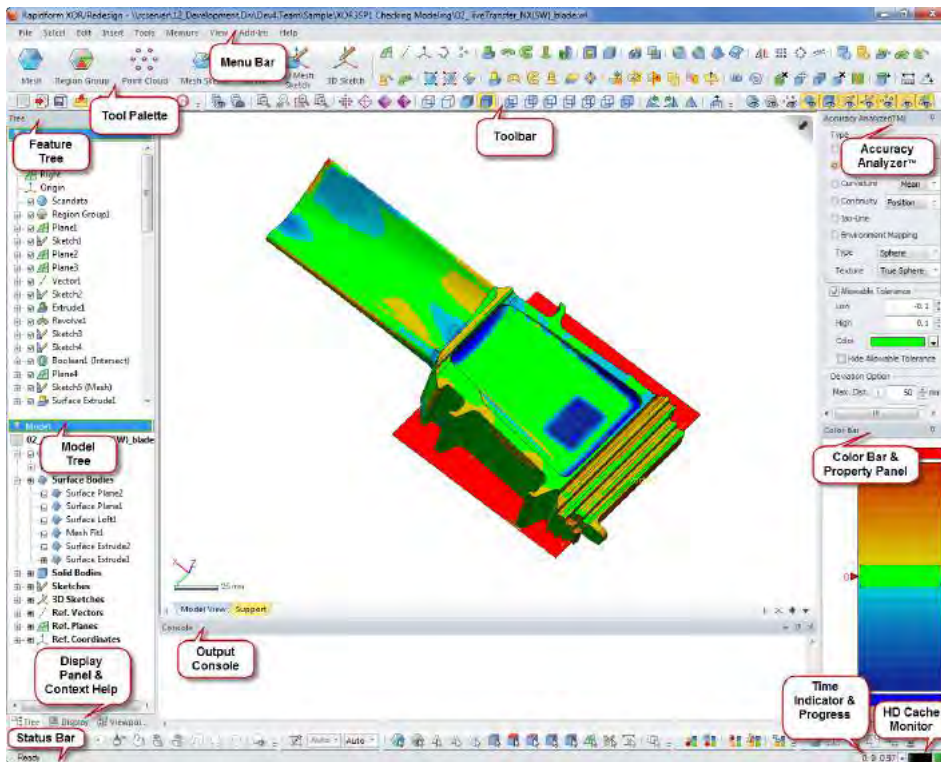
Click **Help** > **Launch Performance Profiler**.

To obtain the latest information:

- Click www.rapidform.com
- Online resources includes up-to-date product information, frequently asked questions (FAQ's), software downloads, and technical support contact information.

6. User Interface

The user interface of Geomagic Design X is intuitive and easy to use. It is a flexible and dynamic interface that is simple to learn. The user interface windows and toolbars can be modified so that they are always shown, dynamically hidden, or never shown, via the right mouse button menu in the toolbar area.

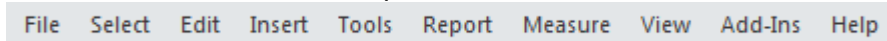


6.1. Title Bar

The title bar shows the full path of the model currently being worked on at the top of the application. For example, if the file “~<Product Installation Path>\Sample\xxx.xrl” is imported, you can see its name in the title bar, but only after it is saved.

6.2. Menu Bar

The menu bar shows the top level of all of the menus. These include File, Select, Edit, Insert, Tools, Measure, View, Add-Ins and Help menus.



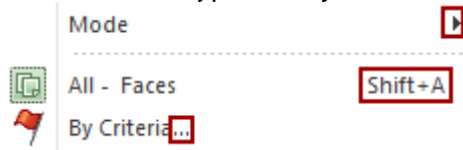
Moving menu bar

The pull-down menu bar can be freely docked anywhere and some commands have icons for quicker referencing.



Menu types

There are three types of objects in a menu:







- Arrow (▾)
If there is an arrow beside a menu option, there is a sub menu.
- Shortcut key (**Shift+A**)
Often used commands also have shortcut keys to the right of the command name.
- Ellipsis (...)
If there is an ellipsis "..." beside a dialog box or dialog tree, the command name has been shortened and will be shown after it is clicked.

6.3. Mode

The application features different modes that allows for the generation and editing of entities with only the appropriate functions displayed. Multiple modes are featured because the application can handle and manipulate many different types of entities that normal CAD programs cannot. To simplify and streamline the process there are different modes that can be used to handle the types of entities that are being worked with.



When you enter a mode the working environment (tool palette, toolbars, selection toolbar, and options under the Accuracy Analyzer pane) is automatically set to the current mode state. The application has the following modes.

-  **Mesh** – The Mesh mode contains functions to edit, heal, enhance and optimize meshes. To enter the Mesh mode, select a mesh and then click the Mesh icon or double-click the mesh in the Feature / Model tree or in the Model View.
-  **Region Group** – The Region Group mode contains functions to classify features by colors and groups. After region segmentation is performed a feature on a mesh can easily be selected instead of selecting a feature by dragging or painting it. To enter the Region Group mode, first select a mesh and then click the Region Group icon.
-  **Point Cloud** – The Point Cloud mode contains functions to clean and edit point clouds; and to generate meshes. To enter the Point Cloud mode click the Point Cloud icon. Functions in the Point Cloud mode apply to all point clouds in the model view.
-  **Mesh Sketch** – The Mesh Sketch mode contains functions to draw and edit sketch entities such as lines, arcs, and splines; by fitting a section polyline onto the model that can be

extracted from point clouds or meshes. To enter the Mesh Sketch mode, a base plane which can be a ref. plane, planar face, or planar region needs to be defined. These sketches are used to generate a surface / solid body.



Sketch – The Sketch mode contains functions to draw and edit sketch entities such as lines, arcs and splines; with or without a mesh or point cloud. Tools in the Sketch mode are the same as those found in Mesh Sketch mode, however fitting is not available. To enter the Sketch mode, a base plane must be defined. It can be a planar entity. These sketches are used to generate additional surfaces / solid bodies without using the section information from point clouds or meshes.



3D Mesh Sketch – The 3D Mesh Sketch mode contains functions to draw and edit 3D curves based on point clouds or meshes. Whenever a curve is drawn or edited, it will be projected onto the point cloud or mesh. To enter the 3D Mesh Sketch mode, select a mesh and then click the 3D Mesh Sketch icon. Curves created in this mode can be used for generating fitting surfaces.



3D Sketch – The 3D Sketch mode contains functions to draw and edit 3D curves as featured in the 3D Mesh Sketch mode, but differs in that any curves created in 3D Sketch mode are not projected onto a mesh. 3D curves can be freely drawn in space or on any entities. To enter the 3D Sketch mode click the icon. Curves created in this mode can be used to get a center line of a pipe or to create a path for a 3D loft or sweep body.

Activating and exiting a mode

How to activate and escape a mode is below.

- Activating mode

Click a mode. When a mode is activated, the corresponding toolbars are also activated.

- Exiting a mode

To accept all changes in the mode click the  icon or the mode button (e.g. ) again. If the  icon is clicked, all changes in the mode will be canceled.

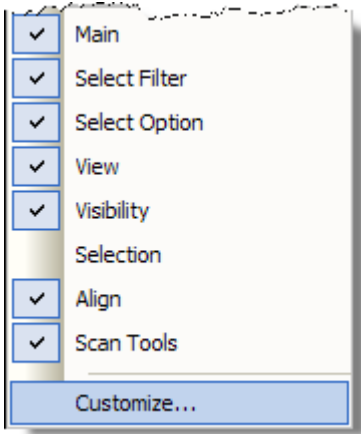
6.4. Toolbar

You have menus or commands that are used frequently. Toolbar is a collection of the menus and commands.



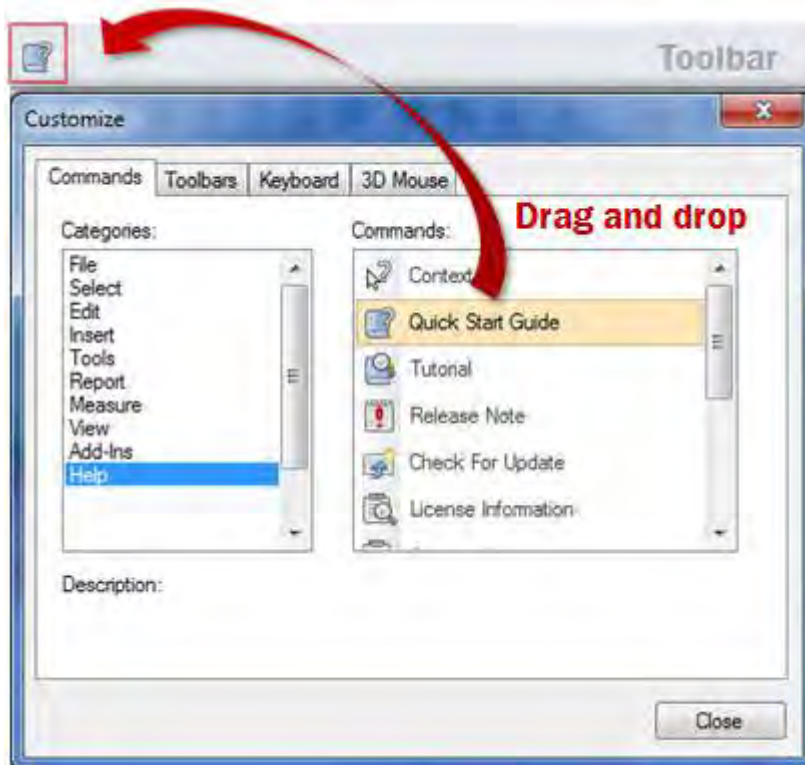
Customizing the toolbar

To customize the toolbar, right-click on it and choose **Customize**.



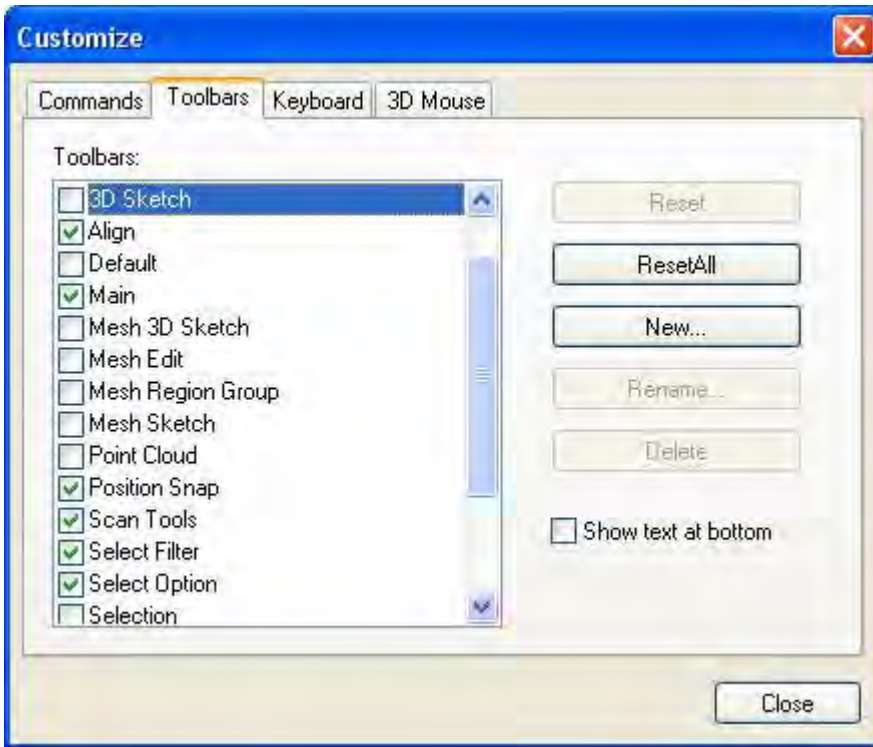
The Commands tab

All commands are listed. When a command is selected, there will be a short description about it. You can drag-and-drop a command into a toolbar.



The Toolbars tab

If you check the toolbar in the toolbars list, the toolbar will be shown immediately.



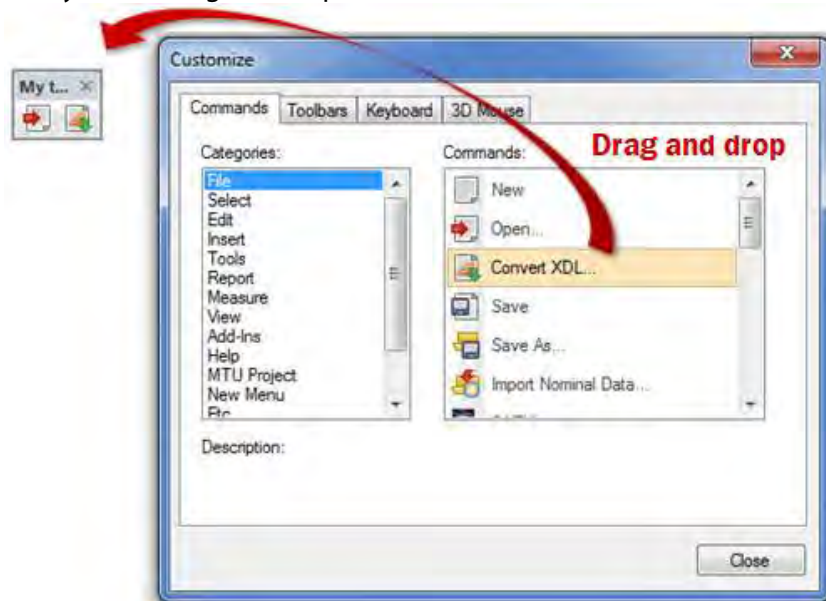
To add a new toolbar, click the **New** button and enter a name for the new toolbar.



When you click **OK**, you will notice the newly created toolbar which is empty.

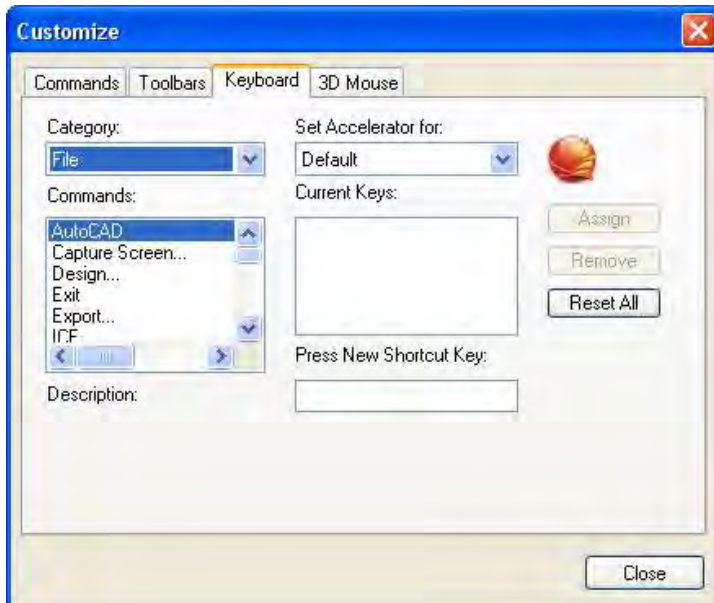


Now you can drag and drop icons from the Command tab to customize the toolbar.



If you click **Reset All**, the icons on the toolbar will reset to default. **Reset All** does not move the position of the toolbar immediately. To finish resetting, close and open the program. To change the name of a toolbar, click the Rename button. To delete a toolbar, click the Delete button. When the Show text option at the bottom is checked, the names of icons will be displayed at the bottom.

The Keyboard tab

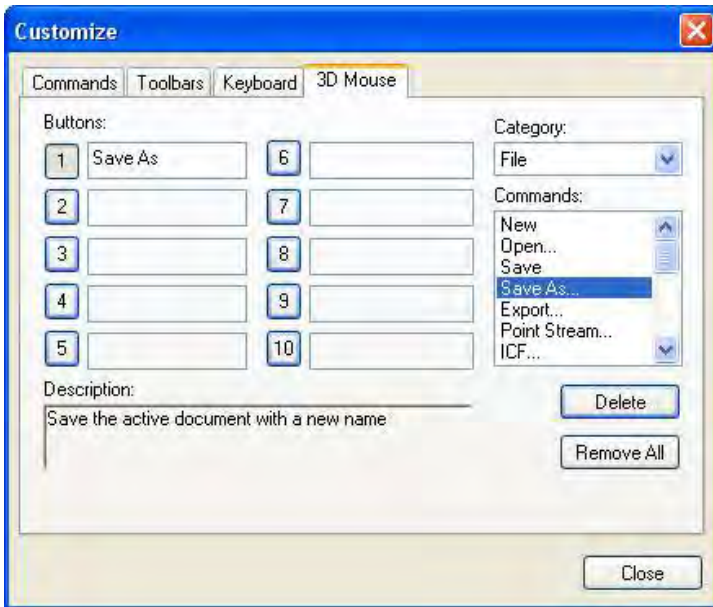


In the Keyboard tab, you can bind keys to a command to create a shortcut. First, you need to select a Category and a Command in it, then press the New Shortcut Key (for example, pressing Ctrl and X). Then when you click Assign, the shortcut key is bonded to the selected command. To take away the shortcut keys of a command, select it in the Current Keys box, and click Remove. To return everything to the original default state, click the Reset All button.

Note:

When a shortcut key is already bonded to a command, you cannot assign another command to it at the same time. It is also not possible to assign Windows level shortcuts, such as Ctrl + P.

The 3D Mouse tab



In the 3D Mouse tab, you can define the shortcut keys for the buttons of a 3D mouse. Click the number of the button and double-click a command from the list box. To remove allocated shortcuts, select the number and click the **Delete** button.

6.5. Docking Bars

You can change the behavior of a docking bar to Floating, Docking, Auto Hide, or Hide. The application includes dockable property bars for the following: Tree, Display, Console, Properties, Accuracy Analyzer(TM), Color Bar, Image Folder, Support, Help, Error List, Device Manager, Viewpoint, and Shader Control.

Showing/hiding docking bars

When a docking bar you want to see is not shown, right-click on the toolbar panel.

<input checked="" type="checkbox"/>	Tree
<input checked="" type="checkbox"/>	Display
<input type="checkbox"/>	Console
<input checked="" type="checkbox"/>	Properties
<input type="checkbox"/>	Image Folder
<input checked="" type="checkbox"/>	Support
<input checked="" type="checkbox"/>	Help
<input type="checkbox"/>	Error List
<input type="checkbox"/>	Tabular View
<input checked="" type="checkbox"/>	Report
<input checked="" type="checkbox"/>	Viewpoint
<input checked="" type="checkbox"/>	Colorbar
	Icon
	Status

<input checked="" type="checkbox"/> (Checked)	The docking bar is shown on screen.
<input type="checkbox"/> Unchecked	The docking bar is not shown.

Check desire docking bars to display them. Recheck the check box to hide the docking bar. The icon on the docking bar also hides it.

Folding/unfolding docking bars

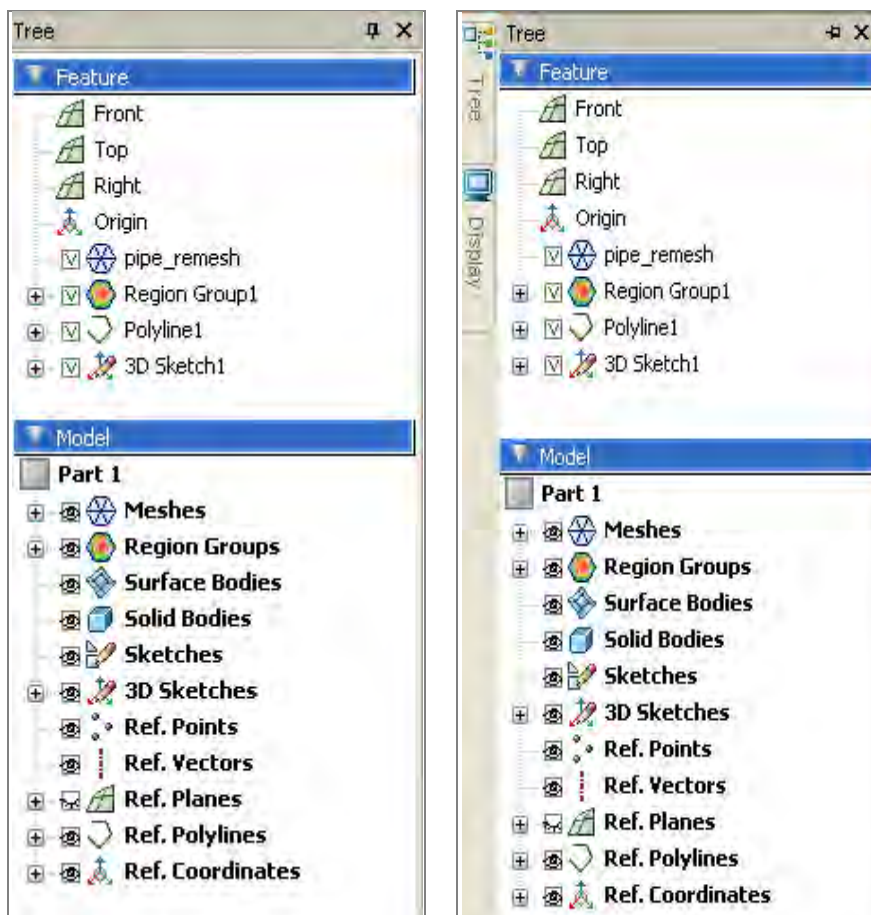
There are two ways to fold docking bars.

- F12 key

The **F12** key hides all docked bars and only shows the model view. Clicking **F12** again restores them.

- (pin) icon

Sets a window to hide automatically. The window will be shown when a mouse cursor hovers over the appropriate tabs for the window.

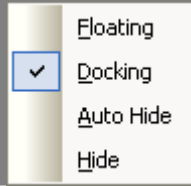


Floating/docking bars

Double-clicking on the title of a docking bar enables it to be moved. Double clicking again docks it.

Note:

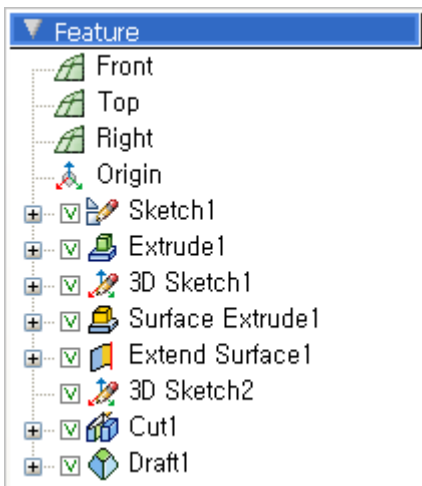
If you right-click on any of the docking bars, the following pop-up menu appears in the view.



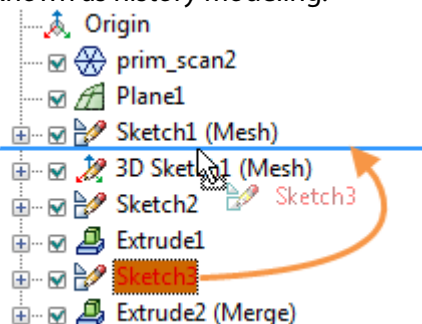
6.5.1. Tree

Feature Tree

Geomagic uses a method known as parametric history modeling. Parametric history modeling allows the creation of objects by storing the operations used to construct the geometry, as well as the order of the operations and their relationships to each other.



Whenever a feature such as a sketch, solid body, surface body, etc., is generated, it will be stacked in the Feature Tree in chronological order. The Feature Tree shows all changes and created entities in chronological order as well as the history for each change or creation. The order of features can be altered by dragging and dropping features. It is also possible to go back to a certain point in the modeling history by using the Rollback, Roll Forward, Roll To End commands. This type of modeling is known as history modeling.



Changing the feature history by dragging and dropping

Selecting several items in a tree

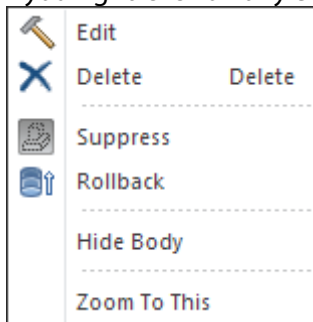
You can select a feature or multiple features by using the **Ctrl** or **Shift** buttons on the keyboard. You can then change the order of the features if desired.


Note:


You can track and select an entity by typing the name of the entity on the keyboard in the Feature Tree.


Pop-up menu


If you right-click on any entity in the Feature Tree, the following pop-up menu will be displayed.



 **Edit** – Edits data based on a selected data type. For example if you select mesh and click edit, you can edit mesh data.

 **Delete** – Deletes features

 **Suppress** - Suppresses a feature so you can work on the model with the selected feature temporarily removed.

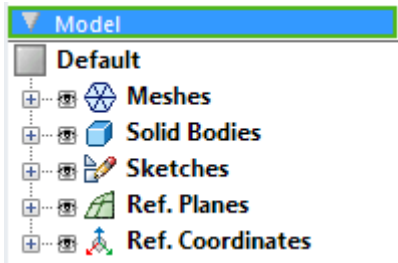
 **Rollback** - Temporarily reverts the model to an earlier state. You can add new features or edit existing features while the model is in the rollback state. When a feature is in the rollback state, any feature created will be added above the selected rollback feature. You can select Roll Forward to roll forward or select Roll To End to go to the end feature.

Hide – Hides features


Zoom To This – Zooms an entity to fit the screen

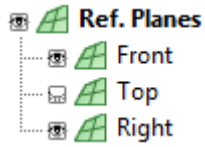
Model Tree

The Model Tree shows all created entities by categories. This pane can be used for selecting and controlling the visibility of an entity.

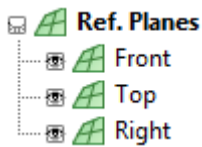


Showing/hiding entities on Model View

If the icon next to an entity appears as a closed eye , the entity will be invisible in the **Model View**. For example, in the scenario below, the Top would not be visible in the Model View.



If the parent node is made invisible, all the child entities are invisible irrespective of their status. For example see the tree below, although Front, Top and Right are set to be visible, the Ref. Planes parent node is invisible, therefore no Ref. Planes will be shown in the Model View.

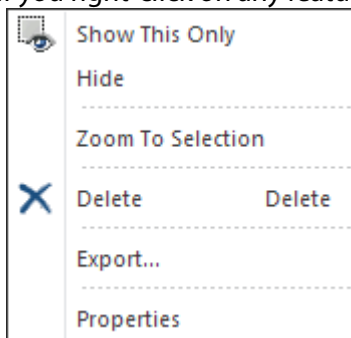



Note:

When the mouse cursor hovers over an entity in the Model View, the entity will be highlighted in the Model Tree. It is useful for recognizing a specific entity in the Model Tree when many entities exist.

Pop-up menu


If you right-click on any feature, the following menu will be displayed.



 **Show This Only** - If selected, only the feature will be shown in the viewing window

Hide – If selected, the feature will be hidden.

Zoom To Selection - If selected, the entity is zoomed to fit the screen.

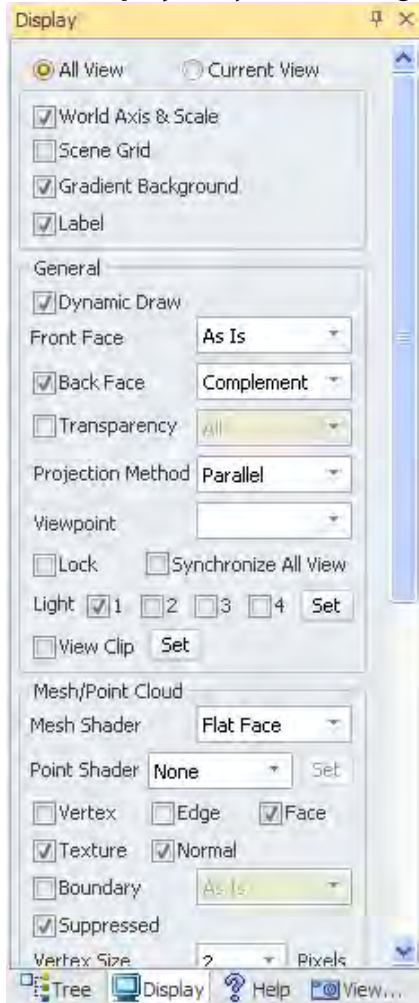
 **Delete** - If selected, the feature will be deleted.

Export - If selected, a dialog box will appear and you can specify the path and the type of file to which the feature will be exported.

Property - If selected, properties will be shown on the right hand side.

6.5.2. Display

In the **Display** tab, you can change the settings related to viewing and displaying.



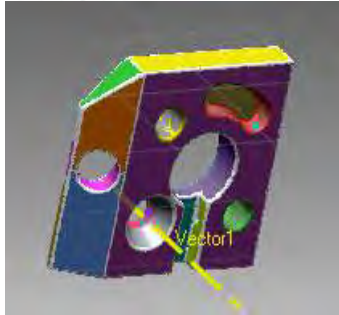
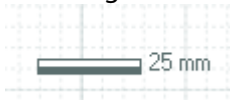
All View - If you set the viewport ([View > Viewport](#)) to display more than two viewing panes and the All View option is checked, all changes made to the viewing and display settings apply to all the views.

Current View - If you set the viewport ([View > Viewport](#)) to display more than two viewing panes and the Current View option is checked, all changes made to the viewing and display settings apply only to the current view.

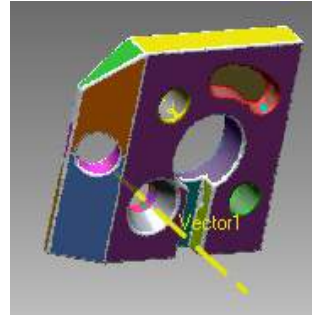
World Axis & Scale - If the **World Axis & Scale** option is checked, a world axis will be shown at bottom left corner in the **Model View**.



Scene Grid - If the **Scene Grid** option is checked, a grid will be shown in the **Model View**. It is located at the bottom of the model in the X-Y plane of the global coordinates. You can view the size of the grid as shown in the image below. This is useful for estimating the measurements of entities while sketching.



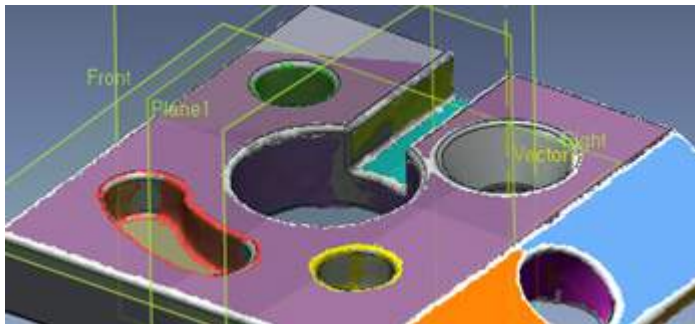
Enabled Scene Grid option



Disabled Scene Grid option

Gradient Background - If the **Gradient Background** option is checked, the background color of the workspace window will be set to a gradient.

Label - If this option is enabled, the labels of the ref. geometries such as planes and vectors are shown in the **Model View**.



General

Dynamic Draw - To simplify the model when it is in motion, check **Dynamic Draw**.

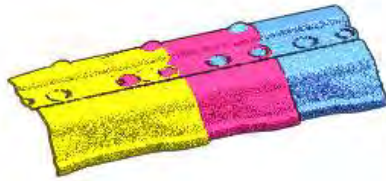


Dynamic Draw option enabled



Dynamic Draw option disabled

Front Face – Determines the surface material of meshes and bodies in the Model View. **As Is** displays each entity as a separate color, however **All The Same** displays all of the entities in gray.



As Is



All The Same

Back Face - If checked, the **Back Face Color** drop-down box will be activated. Within it, there are 3 options. If **Complement** is selected, a complementary color of the front face color will be used for back faces. If **As Is** is selected, the color of the front and back faces become the same. If **User Defined** is selected, a user defined color will be used for back faces. User Defined colors can be set in the **Display** tab in [File > Preferences](#).



Complement

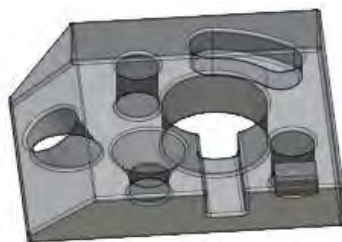


As is

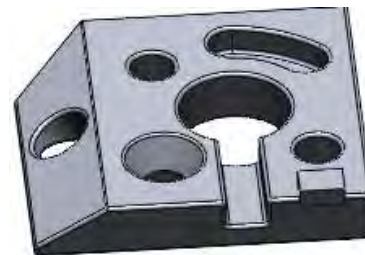


User Defined

Transparency - If checked, you can select an entity type and the selected type will be transparent in the **Model View**.



Transparency enabled



Transparency disabled

Projection Method - If **Perspective** is selected, the model is projected with parallax. In this case, the part at the foreground appears bigger, while the part at the background appears smaller. If **Parallel** is selected, the model is projected parallel to the viewing plane.

Viewpoint – You can set the viewpoint mode as Front, Back, Left, Right, Top, Bottom and Isometric. They are assigned to the following shortcut keys: Alt + 1, Alt + 2, Alt + 3, Alt + 4, Alt + 5, Alt + 6 and Alt + 7.

Lock – If checked, data cannot be rotated in the Model View.

Note:

If you cannot rotate model data, check to see if this option has been activated.

Synchronize All View - If this option is checked, all open view ports in the **Model View** will be synchronized to that in the current view.

Light - To turn Lights 1, 2, 3 or 4, on or off, check the corresponding box next to the number. If you click the **Set** button, you can change properties for lighting conditions. It can also be accessed through: [View > Light Properties](#).

View Clip – If this option is checked, you can see clipped views of model data. To set the clipping properties, click the **Set** button. It can also be accessed through: [View > View Clip](#).

Mesh/Point Cloud

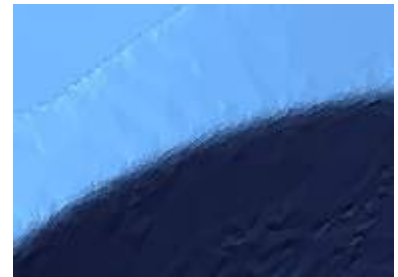
Mesh Display Mode – When **Vertex** is selected, only poly-vertices from a mesh are shaded. When **Smooth Face** is selected, poly-faces will be shaded and connected poly-faces will be blended smoothly. When **Flat Face** is selected, every triangular face will be shaded using one color and connected poly-faces will not be blended. When **Curvature** is selected, roundness will be analyzed and shown using a color range. This color range can be set from the color bar that appears in the right pane. When **Region** is select, region colors will be displayed. The mesh should have regions created in the **Region** mode in advance. When **Estimated Feature** is selected from regions, similar geometry types will have the same color. This color is editable in **Preferences**.



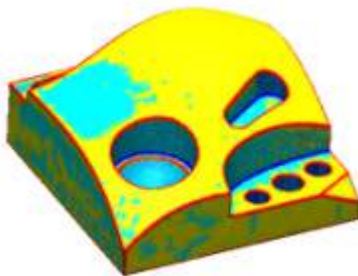
Vertex



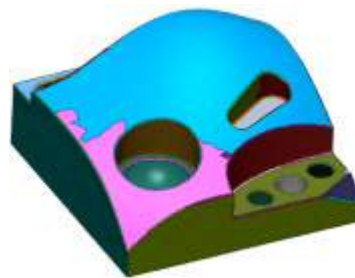
Smooth Face



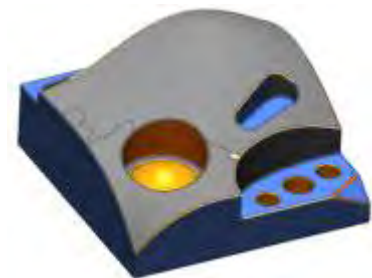
Flat Face



Curvature

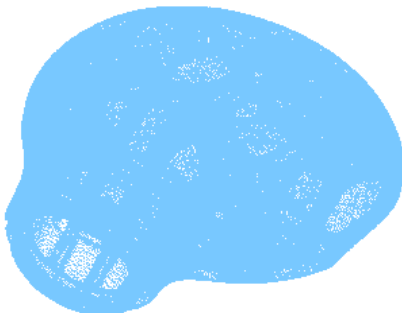


Region

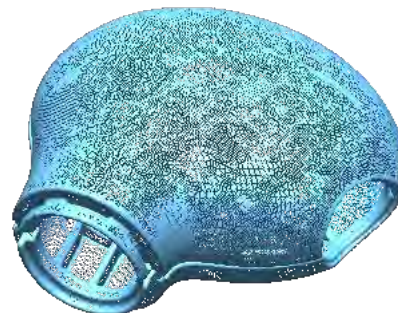


Estimated Feature

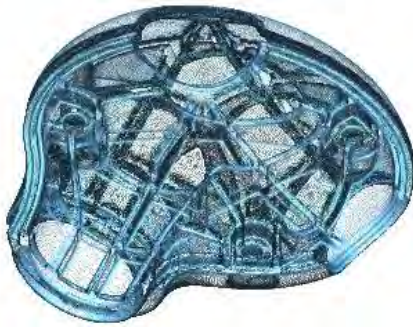
Point Shader – Defines shading methods for point clouds. These shading methods can be used to effectively view scan data.



Mesh Shader: Vertex



Normal Display: On



Depth Point Shading

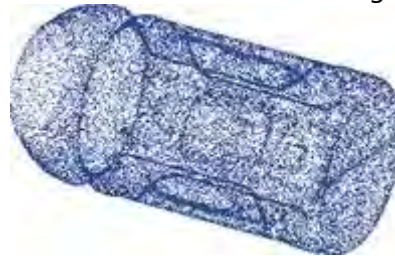


X-Ray Point Shading

None – If a point cloud has color or normal information and the Texture or Normal option is checked, the point cloud is shaded from the direction of the light source. If you turn off Texture or Normal, all points are displayed as the same color without shading.



None with Normal



None without Normal

Depth – This is the default shading method. From the origin viewpoint, the closer a point is, the darker the color of the point will be. If the point is located farther away, the color of the point will become lighter. If you click the Set button, you can define the range of depth by using the adjustable slider. If points are out of the specified range of depth, they will disappear from the Model View. The Depth function can be used to focus on a particular area by removing other unwanted areas from the range of depth.



None



Depth

When the point cloud has normal information, depth shading is available along with vertex shading. Depth shading and vertex shading display the point cloud with enhanced three-dimensional effects so that you can easily distinguish the front of the solid body from the back.



Depth Shading without Vertex Normal

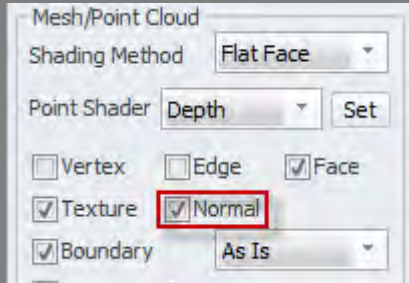


Depth Shading without Vertex Normal

Note:

To create normal information for point clouds during importing, check the Normal checkbox in the Import dialog box.

You can enable or disable vertex shading by checking or unchecking the Normal checkbox in the Display tab.



X Ray – Using this point shader setting helps to focus on an area of interest by making closer and farther areas transparent. To define the transparent areas, click the **Set** button. The **Near** and **Far** areas will be transparent and the **Middle** area will be clearly displayed. For example, if a scanner scans a building, it can be difficult to see the inside because of the outer walls. You can use the X-Ray function to make the outer walls transparent, thus letting you see inside clearly.



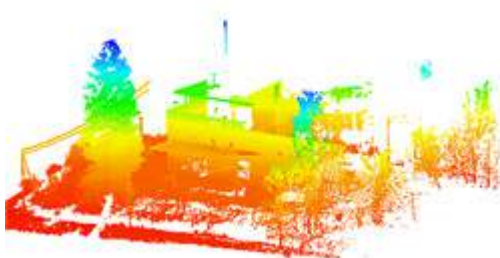
X-Ray Default



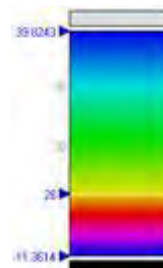
After Adjusting X-Ray Depth

The X-Ray shading method makes points transparent in a certain range. When trying to select points, points nearer to the viewing direction are selected first even if they are invisible. Therefore, this method should be only be used for data viewing purposes.

Height – With the Height option enabled, as the height increases on the Z axis, the color of data points change. Using this shading method, you can easily distinguish where the ground is as well as other height-specific features. If you click the Set button, you can modify the colors shown.



Height



Color Bar

Vertex - If the Vertex option is checked, poly-vertices are drawn.

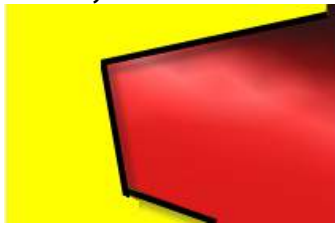
Edge - If the Edge option is checked, poly-edges are drawn.

Face - If the Face option is checked, poly-faces are drawn.

Texture - If the Texture option is checked, textures are drawn on a mesh.

Normal - If the Normal option is checked, normal information is used for shading point clouds.

Boundary – Shows or hides the boundaries of meshes. If you choose **User Defined**, you can display boundary colors as set in **Preferences**. This option is available only for open meshes.



Boundary Enabled



Boundary Disabled



As Is



User Defined

Suppressed - This option enables you to show / hide suppressed scan data.

Vertex size – Determines the display size of the vertex.



Vertex Size: 1



Vertex Size: 2

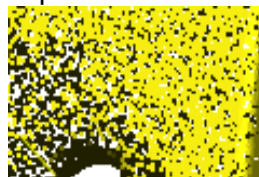
Note:

When trying to see a more detailed image of a point cloud, increasing the point size will cause the points to overlap, looking more like a mesh.

Dynamic Display Ratio - To change the amount of data displayed when a model is in motion, select a value in the **Dynamic Display Ratio** box. This option is enabled only when **Dynamic Draw** is checked.



Dynamic Display Ratio: 5%

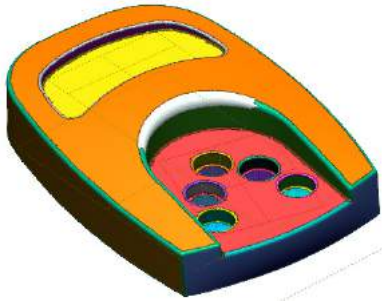


Dynamic Display Ratio: 50%

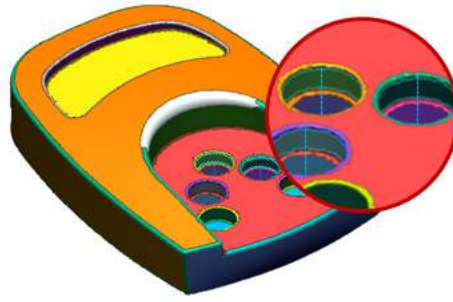
Static Display Ratio - To change the amount of the data displayed when a model is static, select a value in the **Static Display Ratio** box.

Region

Geometry Type – If **All** is selected from the drop-down box, all geometry types are displayed. If **Selected only** is selected only selected geometry types are displayed.



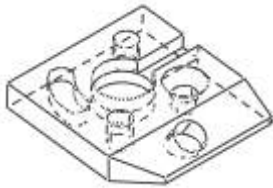
All



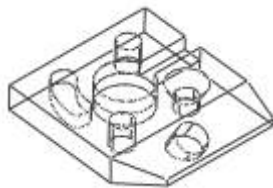
Selected Only (Cylinders)

Body

Display Mode – Determines how the body is displayed. **Wireframe** draws only edges of faces. **Hidden Line** draws all hidden lines. **Shaded** draws shaded faces. **Shaded With Edges** draws shaded faces with edges.



Hidden Line



Wireframe



Shaded



Shaded With Edge

Resolution – You can set the display resolution of bodies with this option. You can set the resolution level from 0 to 10.



Resolution: 0



Resolution: 5



Resolution: 10

Note:

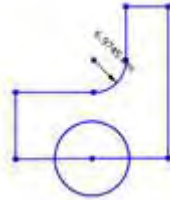
CAD objects may be displayed as a low resolution graphical mesh. If faceting occurs, then the resolution may need to be increased to display the CAD object correctly.

Sketch & 3D Sketch

Constraint – If this option is checked, sketch constraints are displayed while drawing.



Enable Constraint



Disable Constraint

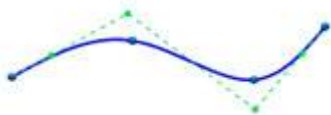
Constraint Options

	Coincidence		Parallel
	Perpendicular		Constraint Fix
	Concentric		Conversion
	Tangent		Horizontal
	Mid Point		Vertical
	Equal Length		Linear Pattern
	Equal Radius		Circular Pattern
	Cordial		Symmetry
	Collinear		Angular
	Pierce		

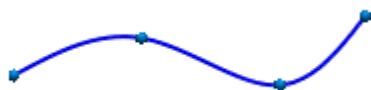
Note:

The color of the constraint icon's background will be different based on its condition. If you can only snap on the entity, the icon background color will be displayed in white . If you can snap and give a constraint to the entity, the icon background will be displayed in yellow . After creation of an entity, the icon background will be displayed in blue . If the entity is overly constrained, the background color of the icon will be displayed in red .

Control Polygon – If this option is checked, control points are displayed on splines.



3D Spline Node – If this option is checked, node points are displayed on 3D splines.



Enable 3D Spline Node

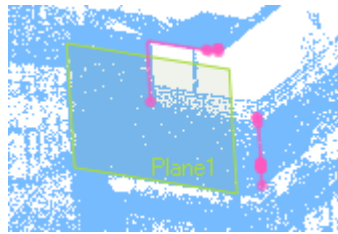


Disable 3D Spline Node

2D Section Points – If this option is checked, points which are extracted by the **Mesh Sketch** function with the **Silhouette** option are displayed.



Show 2D Section Points

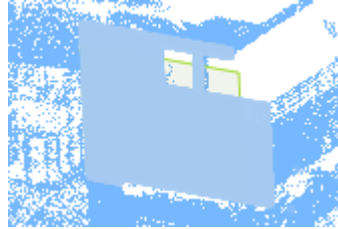


Hide 2D Section Points

2D Section Polylines – If this option is checked, polylines which are extracted by the **Mesh Sketch** function are displayed.



Show 2D Section Polylines



Hide 2D Section Polylines

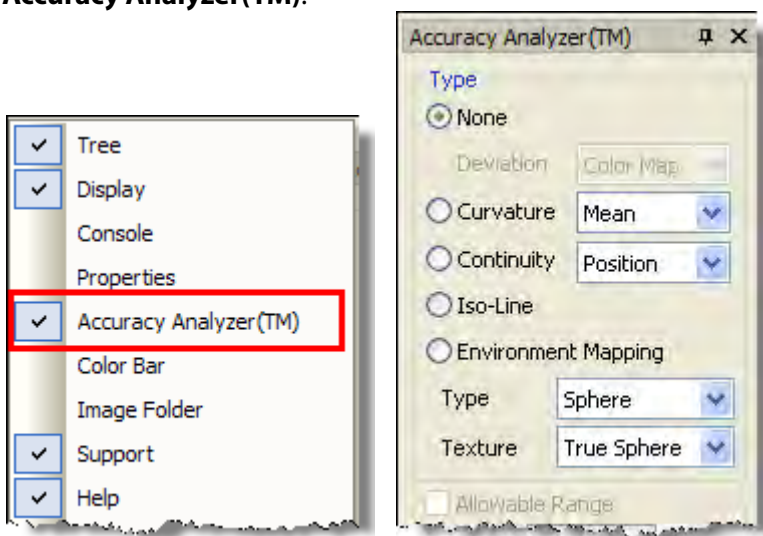
6.5.3. Accuracy Analyzer

The patent-approved **Accuracy Analyzer™** shows real time deviation analysis while you are designing, so you can ensure your finished model is faithful to the original 3D scan. It also saves time because you never have to leave the application to check your modeling accuracy.

Accuracy Analyzer(TM) works in any mode except the **Region Group** mode. You can compare designed bodies, curves, sketches or meshes with a base entity such as a base mesh or a base point cloud. The options enable you to analyze curvature, torsion, environment mapping, iso-line, and continuity to generate high quality surfaces and curves.

How to open the Accuracy Analyzer (TM) panel

If the **Accuracy Analyzer(TM)** panel is closed, use the pop-up menu on the tool palette and choose **Accuracy Analyzer(TM)**.

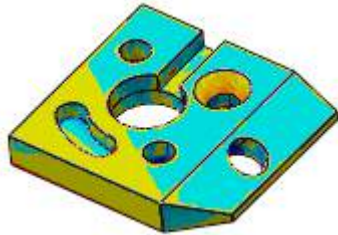


Options in Detail

Type

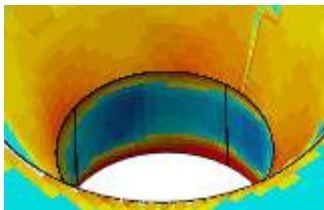
None - If None is selected, nothing is analyzed. This is the default state.

Deviation – Shows a deviation map between a created entity (body, sketch, curve or mesh) and its base (mesh or point cloud). For example, if you design a body based on scan data, you can select the **Deviation** option to check the deviation between the created body and actual scan data. You can easily confirm the accuracy of your body.

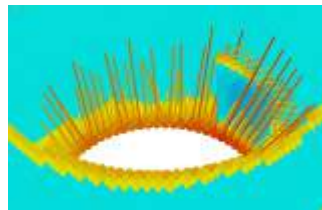


The **Deviation** has two options: **Color Map** and **Vector**.

If **Color Map** is selected, results will be displayed as a colored map on the created entity (body, sketch, modified mesh). If the **Deviation Type** is **Vector**, the results will be displayed as displacement vectors.



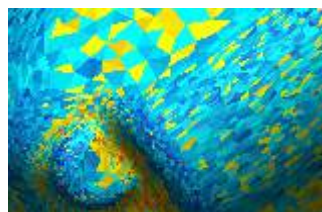
Deviation color map in the Default mode



Deviation vector in the Default mode



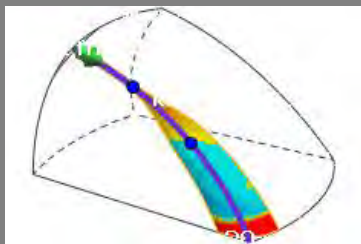
Deviation in the 3D Mesh Sketch mode



Deviation in the Mesh mode

Note:

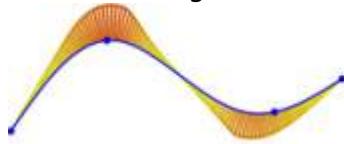
If you try to create a concave fillet and check the deviation while previewing the fillet in Default mode, the deviation may not be shown because the fillet is under the body. In this case, make the body transparent or display it as a wireframe.



If you are in **Mesh** mode, you can see the following drop down option list. You can compare edited mesh with specified mesh using an option from the list.

Compare with Original Me: ▾

Curvature - Displays the curvature flow. Curvature is the amount by which a geometric object deviates from being flat. This is available for curve, body and mesh.



Curvature on a Curve

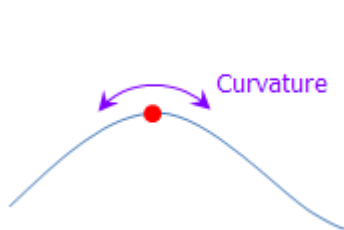


Curvature on a Body

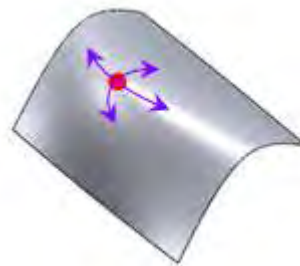


Curvature on a Mesh

The position on the 2D curve has a single curvature value. However, the position on 3D surface has many curvature values according to its sectional the direction on the position.



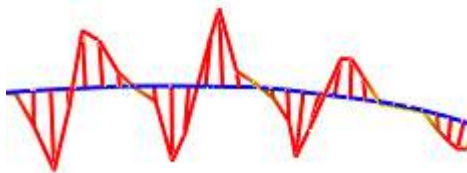
Curvature on 2D



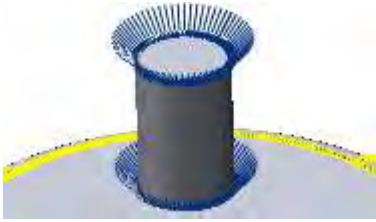
Curvature on 3D

If the **Min.** option is selected, the minimum curvature value at each position will be displayed. If the **Max.** option is selected, the maximum curvature value at each position will be displayed. If the **Mean** option is selected, the mean value of Min. and Max. Curvature value at each position will be displayed. If the **Gaussian** option is selected, the multiplication of Min. and Max. will be displayed.

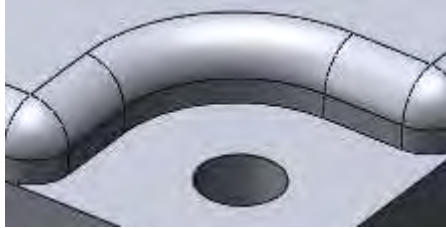
Torsion - The torsion of a curve measures how sharply it is twisting. This option is available only in the **3D Mesh Sketch** and **3D Sketch** modes.



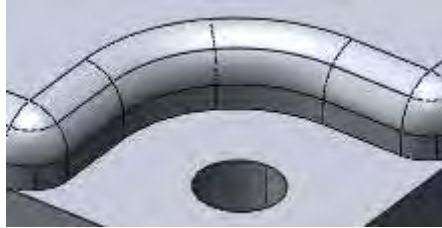
Continuity - This method is useful when inspecting how smoothly connected faces are. If this option is checked, analyzed results are displayed on the edges using color. If **Position** is selected, the positional continuity is displayed on the edges. You can check G0 continuity which defines that positions are continuous. If **Tangency** is selected, the tangential continuity is displayed on the edges. You can check G1 continuity which defines positions and face normals as continuous. If **Curvature** is selected, the curvature continuity is displayed on the edges. You can check G2 continuity which defines that positions, face normals and curvatures are all continuous.



Iso Line - If this option is enabled, Iso lines are displayed on body faces. The flow of Iso lines on a face determines its surface quality.



Disable Iso Line



Enable Iso Line

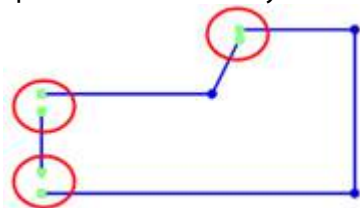
Note:

Fewer isolines will result in a smoother surface; while increasing the number of isolines could introduce noise or waviness.

Environment Mapping - Environment mapping is used to examine the quality of body faces or meshes. You can use different textures for mapping to examine surface quality. The **Sphere** type uses a single texture containing the image reflected by a ball-shaped mirror. The **Cube** type uses a single texture or six square textures containing the reflected image on six faces of a cube. Under the **Texture** option, you can choose an image which will be used for environment mapping. True Sphere shows black and white stripes and it is normally used for inspection of surface quality.




Disjoined Ends - Highlights unconnected points on a sketch as shown in the image below. This option is available only in the **Mesh Sketch** and **Sketch** modes.



Allowable Tolerance

Allowable Value - You can define the **Low**, **High** and **Color** values for the tolerance range. If you do not check the **Hide Allowable Tolerance** option, areas within the tolerance range will be colored green by default.

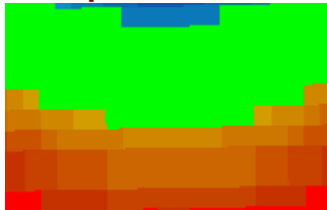
Deviation Option

Deviation Option - Deviation is calculated within specified **Max. Distance** and **Max. Angle** values. **Max. Distance** sets the deviation distance within a specified distance between a target entity and a base entity. The  icon automatically recommends a maximum distance. **Max. Angle** limits the angle deviation between the normals of target entities and a base entity at a position.

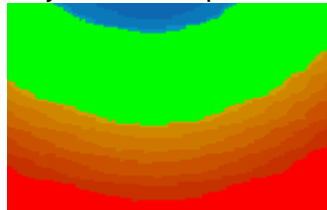
Fine Sampling – Calculates deviation more accurately but may increase calculation time.

Color Map Resolution

Color Map Resolution - You can adjust color map resolution from low to high with the slider.



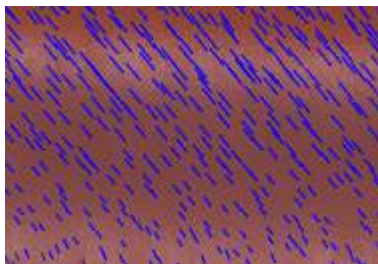
Resolution: Low



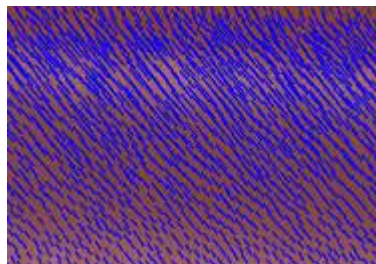
Resolution: High

Vector Resolution

Vector Resolution – Defines the amount of vectors. This option is available when **Deviation** method is set to Vector.



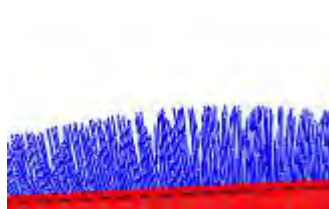
Vector Resolution: Low



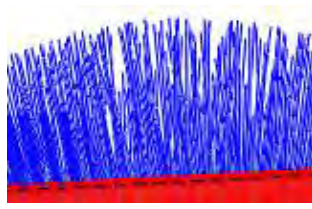
Vector Resolution: High

Vector Multiplier

Vector Multiplier - Vector multiplier is used to adjust vector length. This option is available when **Deviation** method is set to Vector.



Vector Multiplier Value: 1



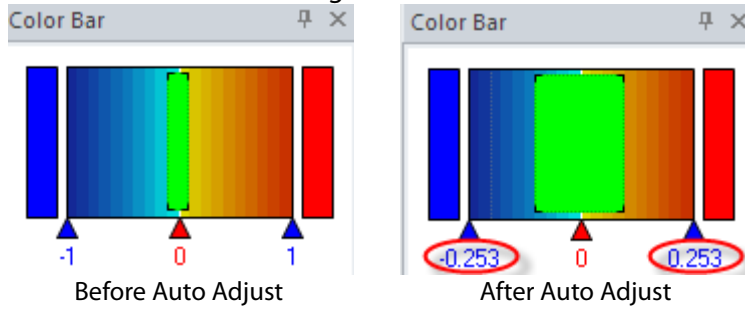
Vector Multiplier Value: 2

Auto – Automatically sets the multiplier value to the same value as the current model zooming level.

Auto Adjust Color Bar Range

Auto Adjust Color Bar Range – Automatically adjusts the minimum and maximum values of the

color bar within a suitable range.



Note:

If you have multiple solid or surface bodies and want to check the accuracy of each individually, select a body in the Model Tree and use the Accuracy Analyzer.

6.5.4. Help

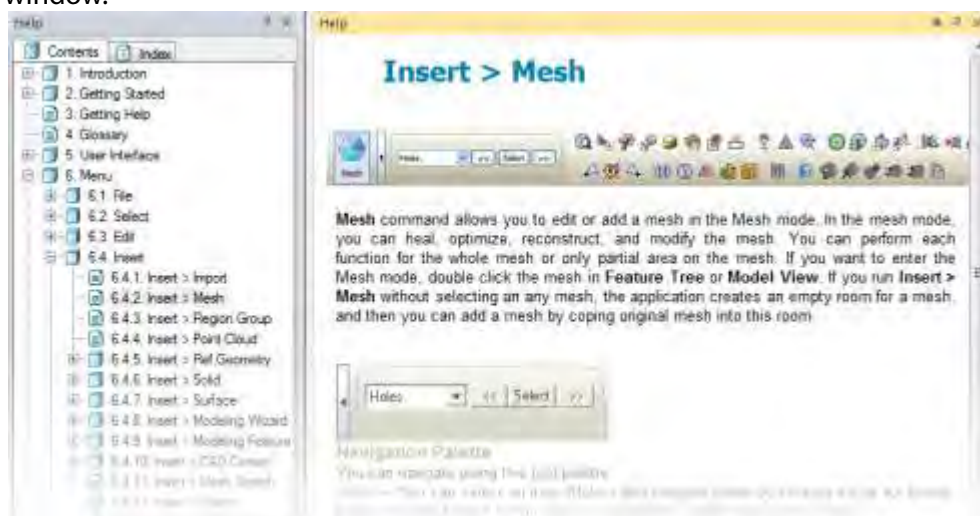
In this tab, you can find the online help file separated into two tabs for **Contents** and **Index**. In the Contents tab, you can find information on all of the commands in the application. To quickly find a command, go to the Index tab, and input key words related to it.

To see the main contents of help guide, press the F1 key while using a command or in any mode. The **F1** key by default will open the contents help guide. This will display the help document for the particular command or mode that is being used.

Note:

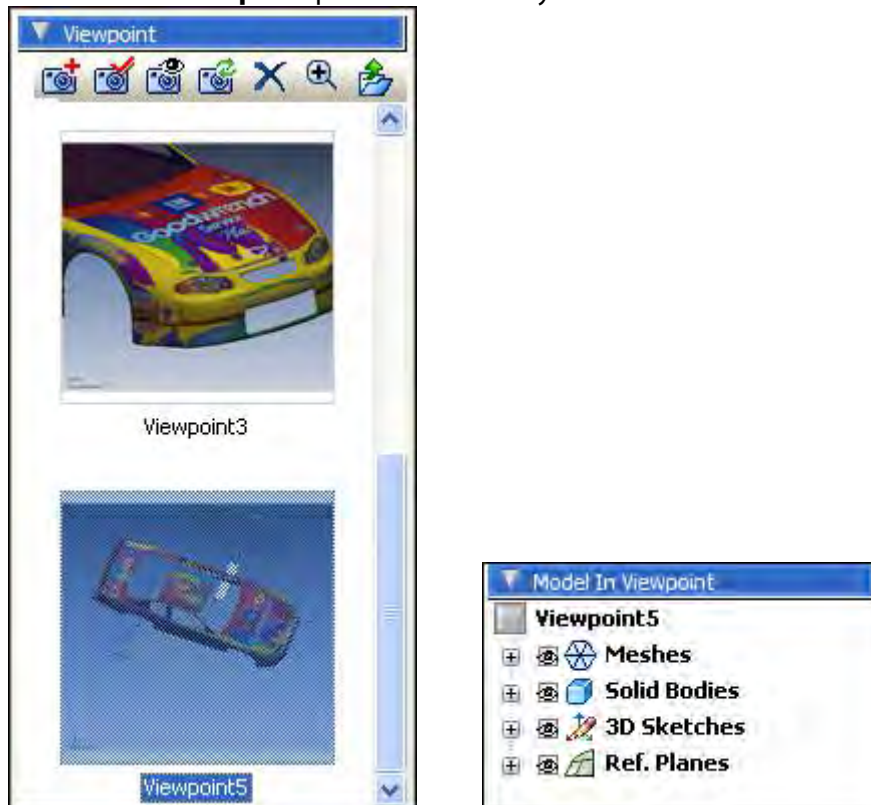
You must be in an active command or in an activate mode to use the F1 function. Otherwise, the intro document page will be displayed if the F1 function is used for the first time or the last help page viewed.

If you want to go back to the working area, click the Model View tab at the bottom of the workspace window.










6.5.5. Viewpoint

Viewpoint manages user-defined points of view with models and shown entities. It is useful for going back to certain viewpoints. When a viewpoint is selected, entities that exist in the selected viewpoint are displayed in **Model In Viewpoint**. The following images shows when Viewpoint5 is chosen in the **Viewpoint** panel that its entity state is also shown in the **Model In Viewpoint** panel.

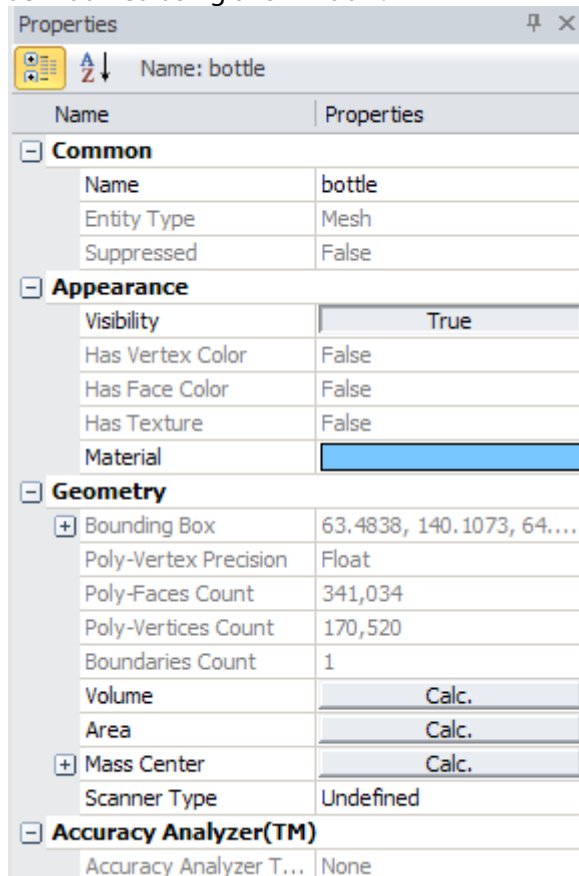


Options in Detail

-  **Add Viewpoint** - Adds a new viewpoint to capture the current view state.
-  **Apply Viewpoint** - Changes the viewpoint to a selected viewpoint. Double-clicking a viewpoint will have the same results.
-  **Show Only Selected Viewpoint** - Changes the visibility of entities to the selected viewpoint.
-  **Reassign Viewpoint** – Reassigns a selected viewpoint to the current view state.
-  **Delete Viewpoint** - Deletes the selected viewpoint.
-  **Zoom** - Determines the size of the thumbnail image.
-  **Export Viewpoint** - Exports the selected viewpoint to an image file.

6.5.6. Properties

The Properties window displays the properties of a selected object. The properties of the object can be modified using this window.



The Calc. buttons are used to calculate the volume, area and mass center of an object. The following list shows the calculation buttons available for each entity.

Mesh: Area, Volume, Mass Center (when the volume of an open mesh is measured, the title will be shown as 'Approx. Volume')

Point Cloud: Mass Center

Region: Area

Region Group: Area

Poly-Face: Area

Solid Body: Area, Volume, Mass Center

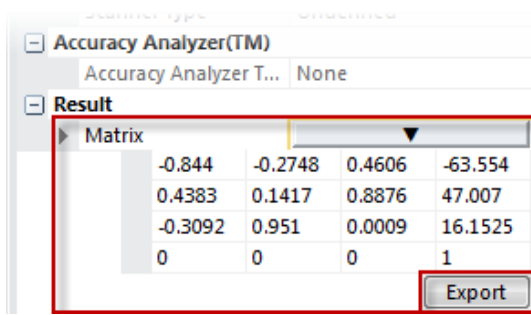
Surface Body: Area, Mass Center

Body Face: Area

When a mesh/point cloud is selected, its geometry information (scanner position, scanner direction or scanner center) can be confirmed as its scanner type in its **Properties** panel.

Scanner Type	Planar
Scanner Position	902.0632, -253.013, -19...
Scanner Direction	-0.6283, -0.7777, -0.0215

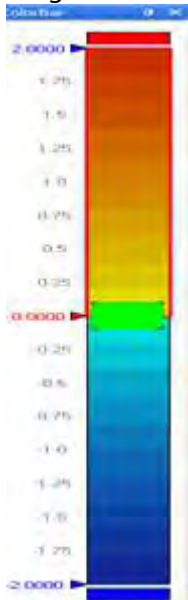
When the position of a mesh/point cloud is transformed to another position, the transformation matrix can be exported in the **.trm** file format and used for other applications by clicking the **Export** button in the **Result** section.



The exported transformation matrix can be used with other scan data files that have been scanned in the same position or to model bodies into desired positions easily. For more information, see [Tools > Scan Tools > Transform Scan Data](#).

6.5.7. Color Bar

The Color bar is used to view the deviation in created entities. The color bar looks like the following image.



Editing Color Bar

The color bar can be edited based on the current project and its tolerances.

Changing limits

A color segment is the area of the color scale between two cursors. Within a color segment, numerical values are mapped linearly to colors. They are the color scale's limits to define the value range of colors used for deviation. These limits can be modified by double-clicking the numerical values and entering new values through their related text boxes. New values are made effective by pressing the Enter key.

Splitting a color segment

A color segment can be subdivided by selecting the Split pop-up menu on the color segment. A new segment is then created on top of the current segment.

Deleting a color segment

A color segment can be deleted by selecting the Delete pop-up menu on the color segment.

Copying a color segment

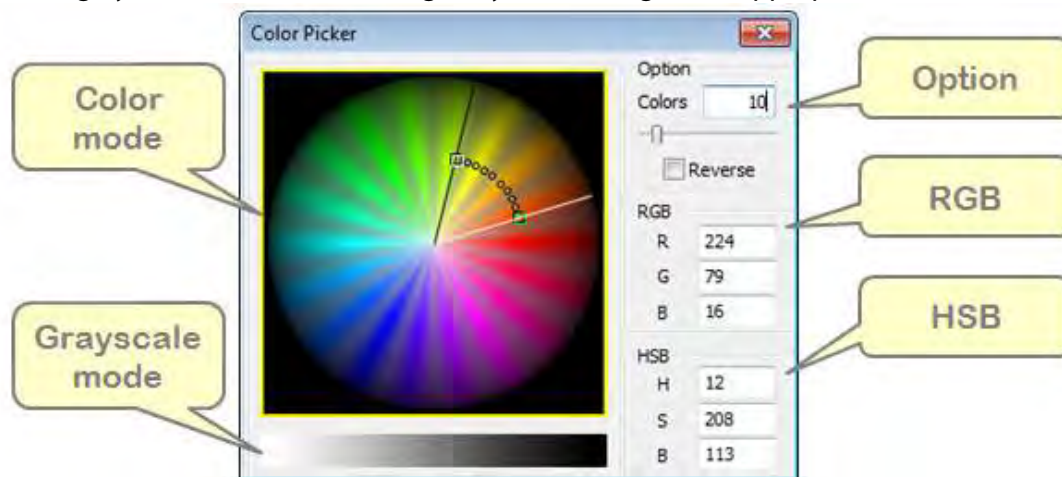
A color segment can be copied by selecting it, right-clicking, choosing Copy, and then Pasting it into another color segment. Alternatively, a color segment can be copied and pasted by using the Ctrl + C and Ctrl + V shortcut keys.

Inspect Color Picker

If you select Edit Color from the pop-up menu on a color segment or double-click it, a Color Picker dialog box will appear. Note that a tolerance color cannot be edited by the color picker.

Construction

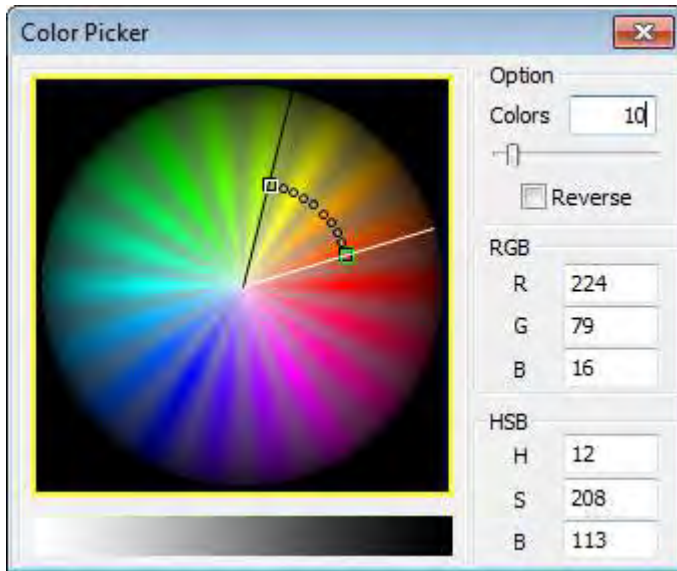
The Color Picker consists of a color mode, a grayscale mode, Option, RGB and HSB; the color mode and the grayscale mode can be changed by left-clicking in the appropriate control sector.



- **Color mode**

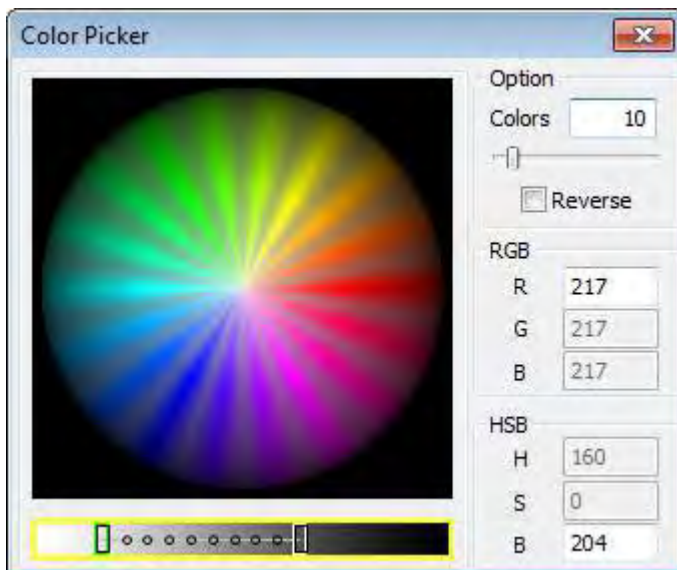
In pinwheel-shaped color mode, the Hue, Saturation, and Brightness properties are defined by moving two squares using the left mouse button and mapping them to the currently-selected segment.

The Hue property is defined by choosing a color section. The Saturation property is defined by moving the squares to the outside of the section from the center, and the Brightness property is defined by moving the squares to the inside of the section from the center.



- **Grayscale mode**

In the grayscale mode, the Brightness property is defined by positioning two squares by using the left mouse button. After positioning, the white-black gradient will be mapped to the currently-selected segment.



- **Option**

In the Option section, the editable box next to the Colors option defines the number of colors and gradients of the currently-selected segment. You can change it by typing in the box and confirming with the ENTER key or by dragging the slide bar. The maximum number is 100.

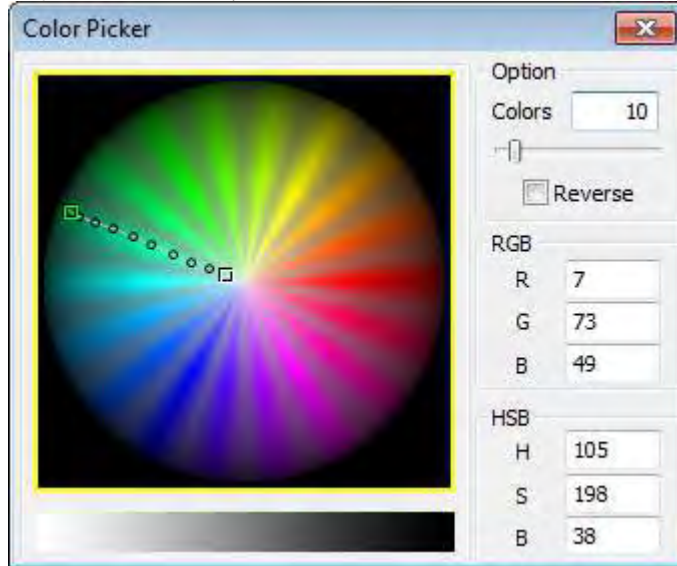
The Reverse checkbox makes the color gradient of the currently-selected segment inverse. The first and last segments of the color bar can only use one color each, so you must select them by using the square selectors in both the color and grayscale modes.

- **RGB and HSB**

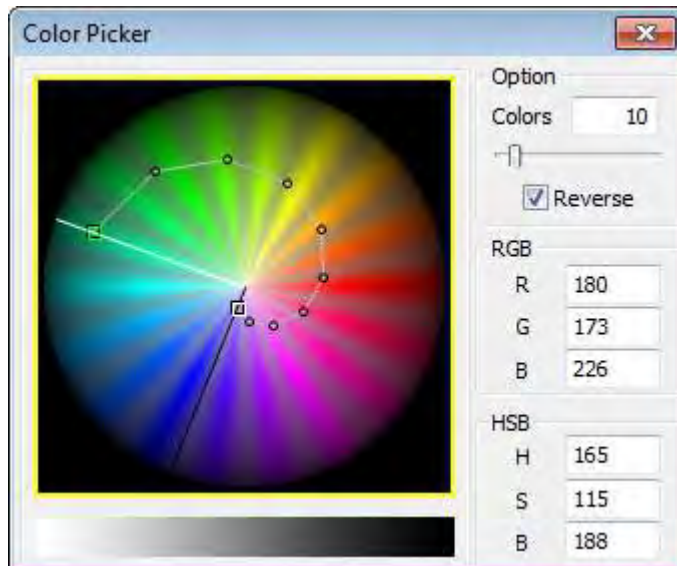
The RGB and HSB sections show the current color properties. The properties can be changed by editing the number boxes.

Making a color bar

For a single color gradient, position one of the squares to define the Hue and Saturation properties. If you move the mouse cursor around the line of the selected color, the other square will be automatically snapped to the line. Finally, position the snapped square on the line to define the Brightness property. Both squares can then be moved along the created line to fine tune if necessary.



For the fixed brightness gradient, first position a square to define the Hue and Saturation properties and position the other square to define the Brightness property. If you position the other square while holding down the Shift key, the color can be changed while keeping the brightness at a fixed value.



6.5.8. Support

The **Online Support Page** of the **Support** panel provides direct access to Geomagic's web-based support data base. It contains useful technical tips, tutorials, FAQs and other information. Feedback and comments can be left for Geomagic products which are useful for enhancing

performance and functionality.

You can also access the **Online Support Page** via your web browser at:

<http://rapidform.zendesk.com>



Note:

If you want to see this Support page when the application starts, go to the General tab in **File > Preferences**, and set 'Show Support At Startup' to True.

Reporting a log file

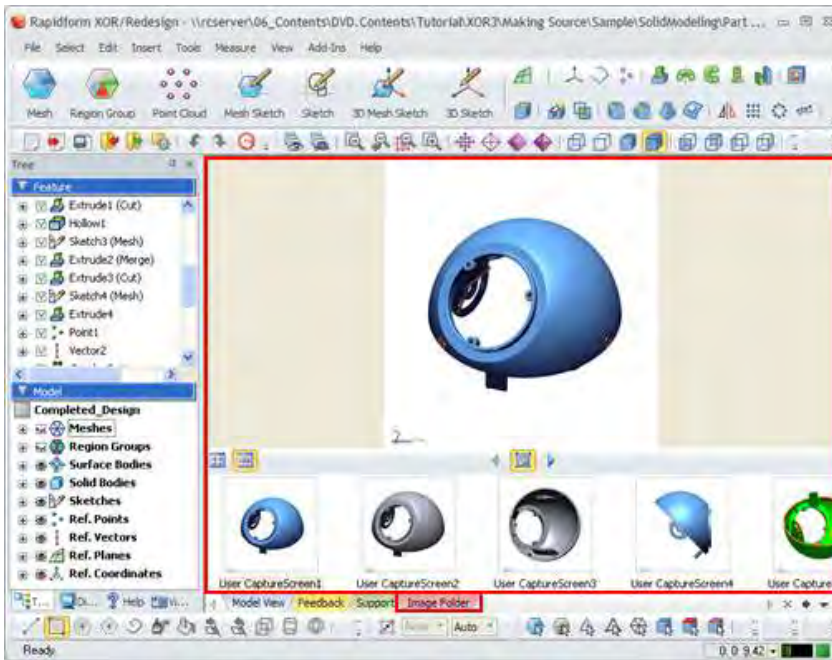
The log file records all actions while using the application. If you meet a sudden error, this file can be helpful for the developers in troubleshooting. Sometimes it is necessary to send this file with a bug report.

To generate a log file;

1. Run **Log_rfXORedesign.bat** located in the installation folder of the application.
2. Use the software and try to recreate the event that you experienced.
3. Once finished, the log file is generated in the **Logs** folder under C:\Documents and Settings*system name*\My Documents*product name*.

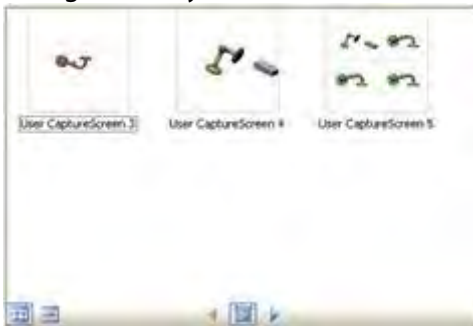
6.5.9. Image Folder

The **Image Folder** stores images that are captured in the application. You can capture images by using **File > Capture Screen**.



Display Types


In the Image Folder, you can choose between two display types; Thumbnail View and Filmstrip View.



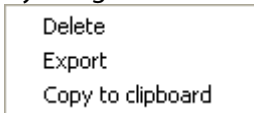
Thumbnail view



Filmstrip view

Images can be navigated through by using the forward and backward  buttons. They can also be rearranged by dragging and dropping them between other images.

If you right-click on a thumbnail, the following pop-up menu will appear.



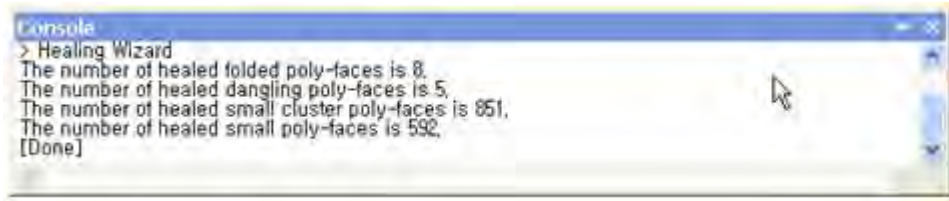
Delete – Deletes the captured image.

Export –Exports the thumbnail to another format like jpg, bmp etc.

Copy to clipboard – Copies the image to the clipboard.

6.5.10. Console

The **Console** panel shows you the detailed results of an operation. For example, the results of the **Tools > Mesh Tools > Healing Wizard** operation shows detailed information about healed poly-faces.



If you click the right mouse button on the **Console** window you can select one of the following commands:

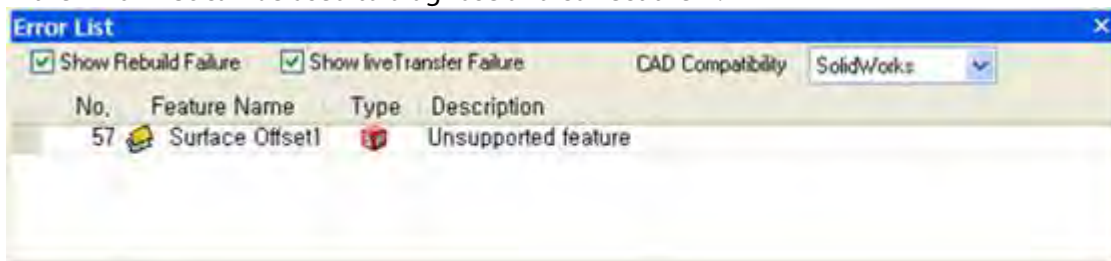
Copy Selection Only – You can copy a selected list.

Copy All – You can copy all lists.

Clear – You can clear all selections.

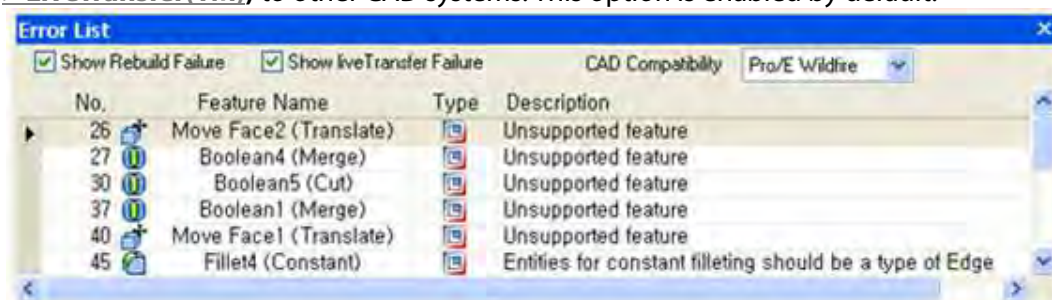
6.5.11. Error List

The **Error List** lists errors that occurred during a modeling process as well as errors that occurred while exporting models to other CAD systems. The errors can be confirmed by using the **Edit > Rebuild** command or by using the **File > LiveTransfer(TM)** command. The descriptions of the errors in the **Error List** can be used to diagnose and correct them.

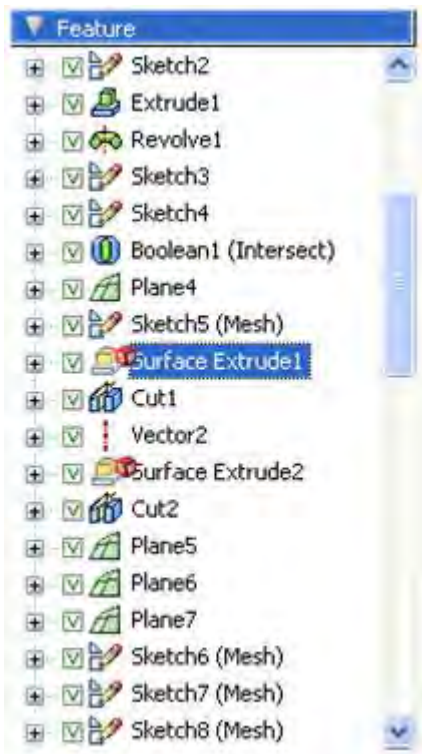


Show Rebuild Failure – Shows errors or warnings that occurred during a modeling process. This option is enabled by default.

Show Live Transfer Failure – Shows unsupported features that cannot be directly transferred (**File > LiveTransfer(TM)**) to other CAD systems. This option is enabled by default.



These errors are marked in the **Feature** tree:



CAD Compatibility – Select one of the applications to check for errors when performing the LiveTransfer operation to selected applications.

Note:

When the “Show LiveTransfer Failure” option is on and “CAD Compatibility” is chosen, you can check for failures while adding or editing features. If there is an error, you might get a dialogue box stating the error, such as the one shown below:



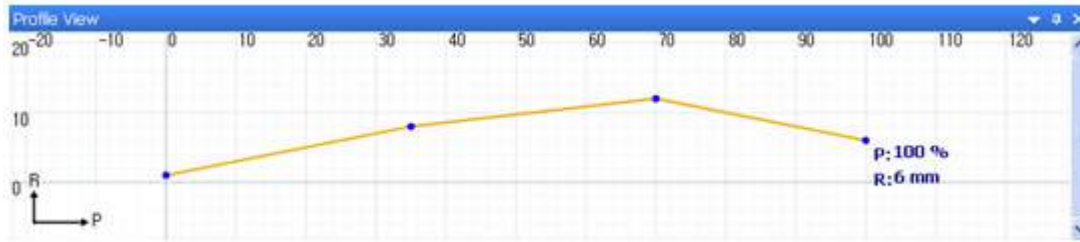
6.5.12. Profile View

The **Profile View** enables you to interactively edit the modeling parameters of an executed command. The **Profile View** appears when you run the **Fillet** (Variable and Conic Option), **Helix** and **Spiral** commands.

Editing Parameters

You can see a line or connected lines in the Profile View and edit them by adding and manipulating

control points or manually input values by clicking the value directly. To add a control point, click a point on the line. To remove a control point, select the control point and use the Delete key or drag it outside of the line. You can also Undo (Ctrl + Z) or Redo (Ctrl + Y) an operation in the Profile View.



Control Viewing


Zoom – Scroll the mouse wheel.

Pan – Use the LMB, or the Scroll Bar.

Scale (Y Direction) – Use Shift + LMB or move the top or bottom edges of the scroll bar.



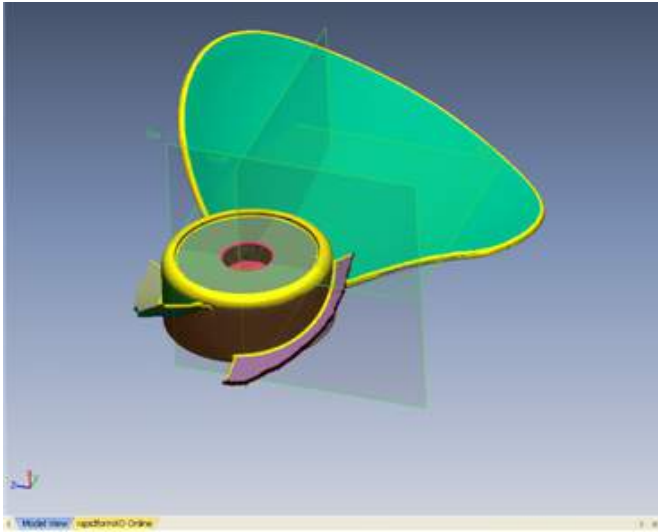
Note:

If you click the down arrow  button at the right top corner of the Profile View, you can access additional functions.

- Zoom Fit
- Fit Graph To Mesh
- Reset

6.6. Viewing Window

The **View Window** is the main workspace. A model is displayed and manipulated in this area. It supports multiple views, which can include 4 different views in four layouts of 3 views and two layouts composed of 2 views. At the bottom of the view window, there are several Tabs such as **Model View, Support, Help, Image Folder**, etc.



6.7. Dialog Tree

When a command name including an ellipsis“...” is selected, a corresponding dialog tree appears in the viewing window. According to prerequisite conditions, some controls might be enabled or disabled.

Common Style Command


The following is common format of dialog tree.




Expand or Collapse

The drop down arrow Icon or the **Tilde (~)** button on the keyboard can be used to expand or collapse a dialog tree.




Auto Hide

If the pin button  is enabled, whenever the user moves the mouse cursor away from the dialog tree, it is automatically collapsed. When the mouse cursor is near the dialog tree it is expanded. This behavior is useful as you can maximize available space to view the model. If the button is disabled, the dialog tree is always expanded irrespective of the mouse position.



Reuse Command

The **Don't Quit Command With OK Button**  preserve the opened command even you click the OK button so that you can continuously use that command.

Preview



The **Preview** button  enables you to preview the result of an operation before you apply the command to a model. If it is clicked, the **Accept** button  and **Restore** button  are enabled.



OK

When you directly click the **OK** button , the command is accepted and you quit the command. Under some commands it is automatically changed to the **Break** button  so that you can stop the command while it is being performed. Also you can use the ESC key to stop the process. While the command is being performed, the dialog tree is disabled and you cannot modify any options or parameters.

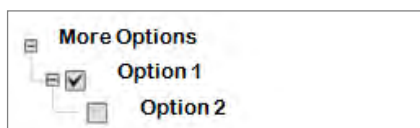
Stage Style Command






Click the **Next Stage** button  to go to the next phase of the procedure. Click the **Previous Stage** button  to go to the previous phase again.

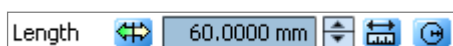
Under some stage style commands, the **Apply** button  is enabled and this button just applies the sub-method under the command. If you click the **OK** button  on the top level, settings in each stage will be applied and you quit the command.




Folded Style



The plus and minus icons,  and , allow the dialog tree to be folded or unfolded or to control various parameters. Click the plus icon  in front of More Options to access more options.

Input Values Manually




Click the **Flip Direction** icon  to flip the direction of a parameter. Click the **Measure Distance** icon  to measure distance and input an alternate value if necessary. Click the **Measure Radius** icon  to measure radius and input an alternate value if necessary. The Measure Distance and Measure Radius commands are the same as **Measure > Distance** and **Measure > Radius** in the top menu. To select a position on a line or plane, hold the **Ctrl** key and pick a point. To average a measured value, use the **Shift** key and measure.

Input Values Automatically




When the value is set to **Auto**, the application automatically estimates the adaptable value considering of the model size and the noise. If you set to “0”, it will be the same as **Auto**.

When you click the magic wand **Estimate** icon , the input value will be recommended by the application.

Selection





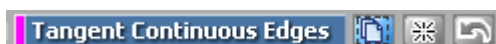
The yellow boxed button **Reference** is currently activated button that is ready to accept your selected entities. If you change the selection on another button, click another button and this button will be activated.

To reset all selected entities, click the **Select None** button . To cancel the selected entity one by one, click the **Deselect Last** button .

Add and Remove



Click the **Add** icon  to add an additional instance of controls. Click the **Remove** icon  to remove an instance of controls.



Click the **All** icon  to select all entities.

Operation



The Edit Control box for a floating-point number allows you to input a numerical value or a numerical formula. The available symbols and functions are as follows:

- A numerical formula that starts with an equal “=”.
- plus(+), minus(-), multiplication(*), division(/), power(^), factorial(!), square root(sqrt), absolute value(fabs), trigonometrical function(sin, cos, tan, sec, cosec, cot, sinh, cosh, tanh, asin, acos, atan), exponential(exp), or logarithmic(log, ln) function.
- No distinction between capital letters and lowercase letters.
- Spacing is ignored.

6.8. Monitoring

6.8.1. Status Bar



The **Status Bar** shows an explanation of a selected command and the progress of an executed command.

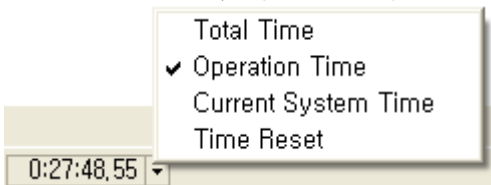
6.8.2. Progress Bar



Progress of the current operation is shown in the **Progress Bar**. The progress will be displayed as a numerical percentage.


6.8.3. Time Indicator

You can check the **Total Time**, **Operation Time**, **Current System Time**, as well as **Reset Time for the application** by right-clicking on the time indicator.



The total time is the time spent in a session. Operation Time is the time spent on one specific operation. Current System Time is the computer clock time. If Reset Time is clicked, the time is changed to 0:0:0.

6.9. Esc Key

To cancel a current operation, press the Esc key. It is same as clicking the **Break** button . The Esc key cancels the command execution and deselects all the currently selected entities.

6.10. Mouse Control

The mouse is the main device for controlling the application. Clicking buttons or objects with or without certain keyboard “modifier” keys alters the way you create, select, move, edit or view models on the screen. For simplicity, we have used specific instruction conventions within this manual:

Left-click means press and release the Left mouse button.




Middle-click means press and release the middle mouse button.

Right-click means press and release the right mouse button.

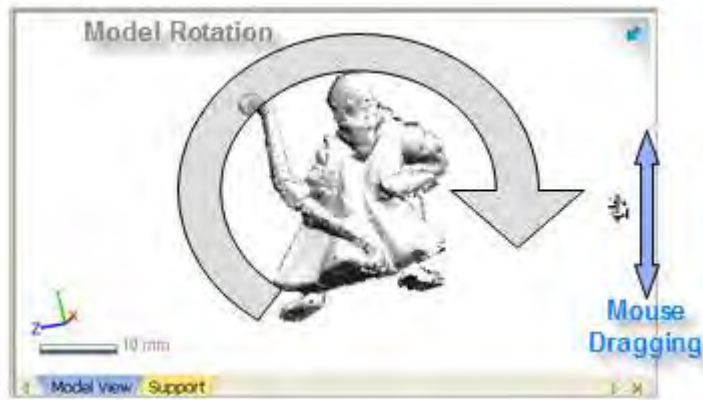
Double-click means rapidly click the left mouse button twice.

Drag means click and hold down the left mouse button and move the mouse while continuing to hold the button down.

The mouse actions and mouse action modifiers appear in the following table.

Mouse Button	Action	Usage
Left 	Left-click	- selects a single item, button or icon - run menus and select options
	Double-click	- open the properties of the node in the model. - make a docking window float, or vice versa
	Shift + Left-click	- select multiple items
	Ctrl + Left-click	- deselect items
	Drag	- rotate (in viewing mode after middle-clicking) - select multiple items - move toolbar, window, dialog tree or dialog box
	Shift + Drag	- zoom (in viewing mode after middle-clicking)
	Ctrl + Drag	- pan (in viewing mode after middle-clicking)
Middle 	Middle-click	- run the Pause command for switching selection mode to viewing mode, or vice versa
	Scroll wheel	- dynamic zoom in/out
Right 	Right-click	- access pop-up menus which contain a context-sensitive set of frequently used functions
	Drag	- rotate
	Left + Right-click + Drag	- pan
	Ctrl + Drag	- pan

If your mouse position is near a border in the **Model View** and the model can be rotated, the cursor changes to the rotate icon; 



6.11. Shortcut Keys

The shortcut keys provide quick access to the most commonly used commands. Keyboard shortcuts offer an alternative method of performing actions for users who prefer the keyboard to the mouse. Some of the commands in the application have default shortcut keys. You can define your own shortcut keys using the Customize menu on the toolbar.

The following is a list of default shortcut keys in the application.

File menu	Shortcut Key
New	Ctrl + N
Open	Ctrl + O
Save	Ctrl + S
Print	Ctrl + P

Select menu	Shortcut Key
All	Shift + A, Ctrl + A
None	Shift + N, Ctrl + D
Inverse	Shift + I, Ctrl + Shift + I

Edit menu	Shortcut Key
Undo	Ctrl + Z
Redo	Ctrl + Y
Repeat Last Command	Ctrl + Space
Cut	Ctrl + X
Copy	Ctrl + C
Paste	Ctrl + V
Delete	Delete


View menu	Shortcut Key
Zoom Area	Shift + Z
Zoom Previous Area	Shift + X
Zoom Fit	Ctrl + F
Zoom PIP	Ctrl + W
Mesh Display Mode > Point Set	F5
Mesh Display Mode > Wireframe	F6
Mesh Display Mode > Shaded	F7
Mesh Display Mode > Shaded	F8
Mesh Display Mode > Region	F9
View > Viewpoint > Front	Alt + 1
View > Viewpoint > Back	Alt + 2
View > Viewpoint > Left	Alt + 3
View > Viewpoint > Right	Alt + 4
View > Viewpoint > Top	Alt + 5
View > Viewpoint > Bottom	Alt + 6
View > Viewpoint > Isometric	Alt + 7
View > Viewpoint > Align View	Ctrl + Shift + A
View > Visibility > Meshes	Ctrl + 1
View > Visibility > Regions	Ctrl + 2
View > Visibility > Point Cloud	Ctrl + 3
View > Visibility > Surface Bodies	Ctrl + 4
View > Visibility > Solid Bodies	Ctrl + 5
View > Visibility > Sketches	Ctrl + 6
View > Visibility > 3D Sketches	Ctrl + 7
View > Visibility > Points	Ctrl + 8
View > Visibility > Vectors	Ctrl + 9
View > Visibility > Planes	Ctrl + 0
View > Visibility > Smart Visibility	Alt + Space

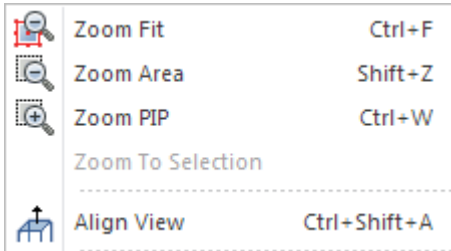
Etc	Shortcut Key
Auto Hide Docking Bar	F12
Help Context	F1
Break Command	Esc
Apply Command	Return
Repeat Latest Command	Ctrl + Space
Shrink Dialog Tree	` (Single Quote)

6.12. Pop-up Menus

In the application view, if you right-click you can see access pop-up menus. The pop-up menu that appears is based on the mode and current view.

Viewing mode ()

If you are currently in the viewing mode () and right-click the mouse, you can see the following menu.



Zoom Fit - If selected, fits the model to the entire viewing window.


Zoom Area - If selected, zooms the viewing area in on a selected rectangular region.

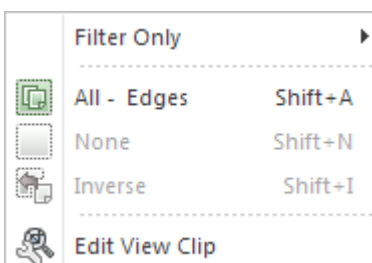
Zoom PIP – If selected, enlarges a view at the position of the mouse cursor .

Zoom To Selection – If selected, selected entities are fitted in the graphic area.

Align View – If selected, you can align the view respective to the selected planar entity.

Selection mode ()

If you are currently in the selection mode () and if you right-click in an empty area, you will see the following pop-up menu.



Filter Only – You can directly access to **Select Filter** and limits the entity type.

All – (Entity Type) - If selected, all entities of the current type will be selected.

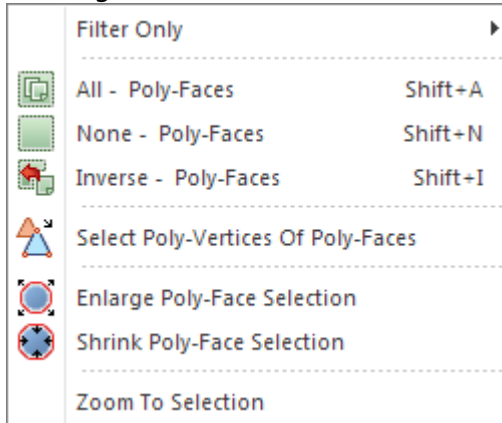
None - If selected, all selections will be released.

Inverse – If selected, current selection of the current entity type will be inverted.

Edit View Clip – You can directly access to [View > View Clip](#).

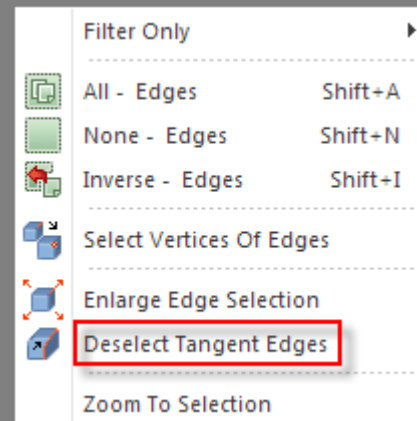
After Entity Selection

However, if you select a certain entity and access the pop-up menu, the command list as shown in the image below will be seen. The following image is the menu for Poly-Faces.



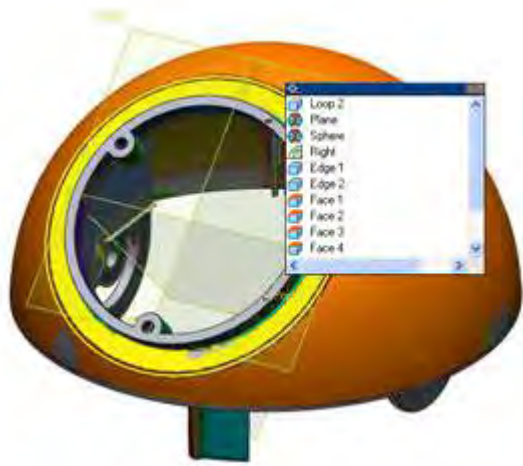
Note:

If you hold the **Ctrl** button and access a pop-up menu for the selection of an edge, you can see the **Deselect Tangent Edges** command instead of **Select Tangent Edges**.



Selection on overlapped entities

When there are overlapping entities in the Model View, it could be difficult to select an entity. In such cases, if you hold a left-click on an entity and release, the pop-up menu can be accessed.



6.13. 3D Mouse

The application supports a 3D hardware device that can help in the modeling process. (3Dconnexion, <http://www.3dconnexion.com>)

A 3D Mouse is a device similar to a joystick that allows intricate adjustments of camera views and models without repeatedly stopping to change direction, zoom, or rotate. It also provides movement controls with 6 degrees of freedom. Tilt up or down to move a model up or down. Tilt left or right to move a model left or right. Pull towards you or push away to move a model nearer or farther away. Orientate a model on the screen by simply twisting in any direction to rotate it around the X, Y or Z axes (pitch, roll, and yaw).

The Customize dialog box can be opened by using the **RMB** (Right Mouse Button) on the toolbar. Commands can be assigned to each button on the device,



Configurations will be saved at the following location; C:\Documents and Settings\User

Note:

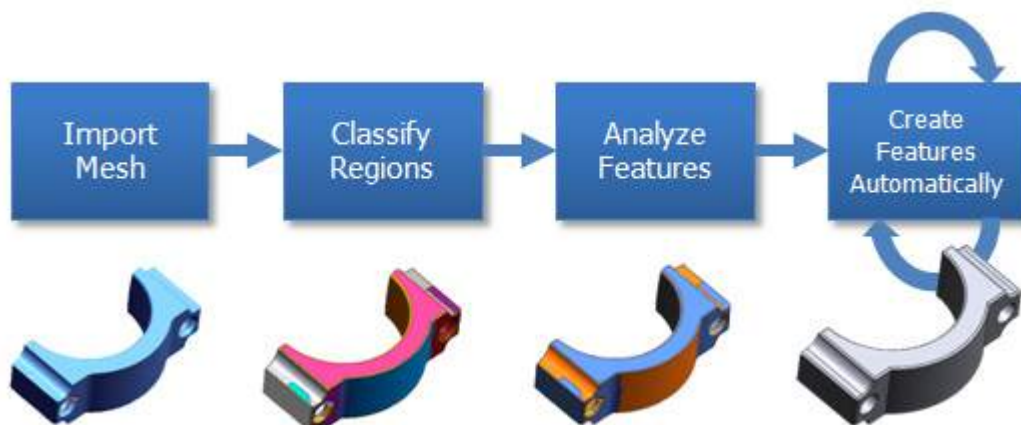
Depending on the device and the pre-set settings of some buttons, you may not be able to customize them.

6.14. Quick Modeling

Quick modeling is a semi-automated modeling process where the application analyzes geometric features and creates extrusions and revolutions from a mesh. Using this information a user can then decide to add new solid or surface bodies or cut existing bodies.

A feature consists of regions, sketches, axes of extrusion or revolution, extrusion distances, or revolution angles. Geomagic Design X can find all elements of a feature automatically and generate it as a solid or surface body. The elements of a feature are registered as a normal sketch and body feature in the Feature tree, so they can be modified easily at any time.

Quick Modeling Workflow



- **Import Mesh:**
Imports scan data which must be a mesh
- **Classify Region:**
Applies auto segmentation to classify regions
- **Analyze Features:**
Analyzes features internally after region grouping
- **Create Feature by User Choice:**

Automatically creates extrusions, revolutions, fillet features and mesh fit surfaces from a selected area of a region. By adding or removing features details of a part can be created. Combined with other general surface or solid tools, a complete part can be made.

How to Access the Quick Modeling UI

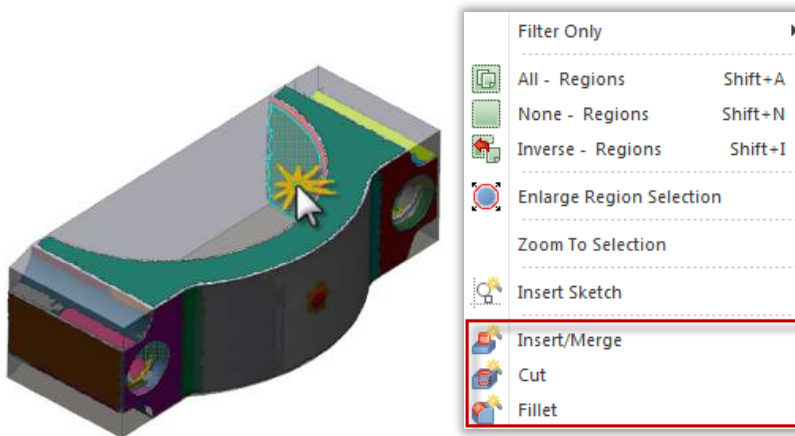
General modeling processes can be accessed by targeted a region and using the pop-up menu. An appropriate wizard function will automatically initialize and the application will also intelligently provide sub-operations such as the merge or cut commands if there are existing bodies.

The following functions are available:

Insert/Merge – Creates a new solid or surface body or merges existing bodies

Cut – Removes a feature from existing features

Fillet – Applies filleting to features

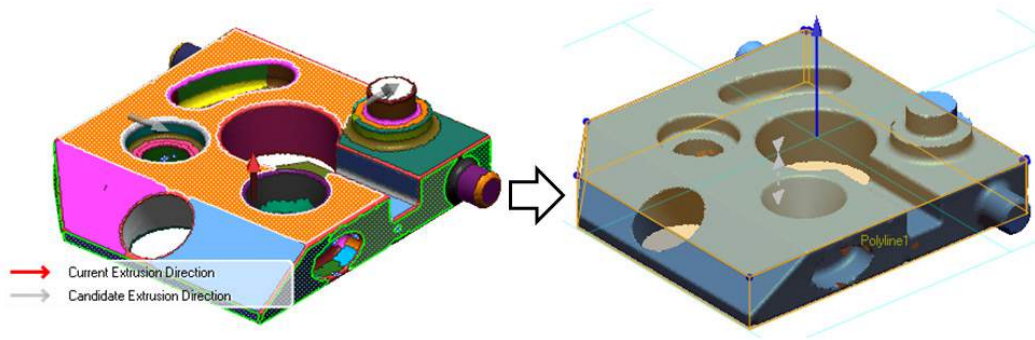


How to Recognize Feature Shapes and Create a Solid or Surface Body

The application will choose and run a Modeling Wizard based on the feature shapes analyzed from selected regions.

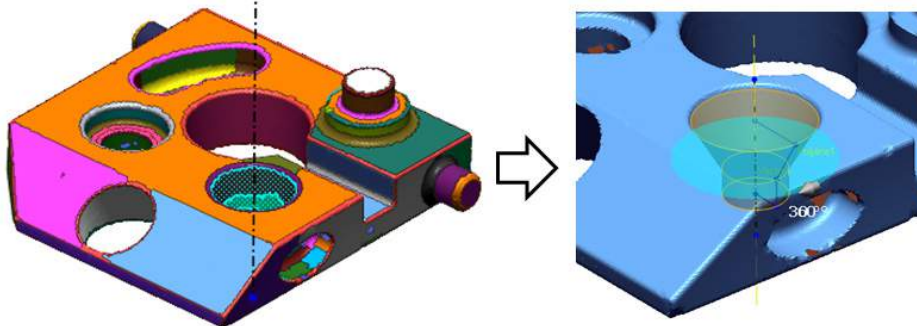
Extruded Feature Shape

When an extruded feature shape is recognized from selected regions, the **Extrusion Wizard** will automatically run and it will find target regions on all sides. A proper extrusion direction will be recommended from extrusion direction candidates. For more information, see [Insert > Modeling Wizard > Extrusion Wizard](#).



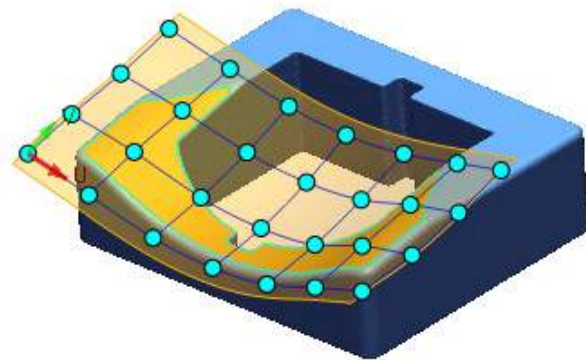
Revolved Feature Shape

When a revolved feature shape is recognized from selected regions, the **Revolution Wizard** will automatically run and find target regions. A proper revolution axis will be automatically extracted from the target regions. For more information, see [Insert > Modeling Wizard > Revolution Wizard](#).



Freeform Feature Shape

When a freeform feature shape is recognized from selected regions, the **Mesh Fit** command will automatically run and find target regions for creating features. For more information, see [Insert > Surface > Mesh Fit](#).



7. Menu

The topmost menus in Geomagic Design X are: **File, Select, Edit, Insert, Tools, Measure, View, Add-Ins** and **Help**.

Click on one of the menus. You may notice some of the menu choices are grayed-out and unselectable rather than in black and selectable. There are two possible reasons for this:

- That particular command is not available for the current operation.
- The appropriate prerequisite operations have not been performed. It's sometimes necessary that you perform operations in a certain order.

7.1. File

The **File** menu is the first menu on the menu bar and includes commands to open new or existing files, import additional files into the application, save files, and take screen captures of models and the application. In the **File** menu, the default model settings and global options that affect all the objects in the viewing window can be altered. The **Exit** command, which closes the application, is also located in the **File** menu.

7.1.1. File > New

What

The **New File** command creates a new file to use within the application.

Why

The **New File** command is useful for:

- Creating a new file for a new project

How to Use

How to create a new file

1. Go to **File > New**, click the icon in the toolbar, or use the **Ctrl + N** keyboard shortcut.
2. A save file confirmation dialog box will appear for any existing open models that have been changed. A save confirmation box will not appear for models that had only simple changes made to it such as a view rotation.

3. Click **Yes** to save any open models. To proceed without saving any open models, click **No**. To cancel the **New File** operation, click **Cancel**.

See Also

[File > Exit](#)

7.1.2. File > Open

What

The **Open File** command opens an existing xrl file.

Why

The **Open File** command is useful for:

- Opening files native to the application
- Opening files compatible with the application

How to Use

How to open a file

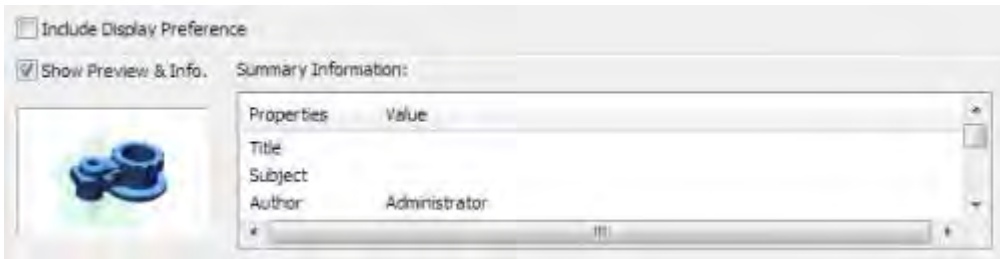
1. Go to **File > Open** or click the icon in the toolbar, or use the **Ctrl + O** keyboard shortcut.

Note:

The application cannot open an .xrl file that was created in a newer version of the software.

2. A save file confirmation dialog box will appear for any existing open models that have been changed. A save confirmation box will not appear for models that had only simple changes made to it such as a view rotation.
3. Click **Yes** to save any open models. To proceed without saving any open models, click **No**. To cancel the **New File** operation, click **Cancel**.
4. The Open dialog box will appear. Select the path of the file in the **File Name** box.
5. Click **Open** to open the selected file or click **Cancel** to quit the command.

Options in Detail



Include Display Preference – Includes the display preferences stored with an .xrl file that can also be accessed by going to **File > Preferences**.

Show Preview & Info. – Shows a thumbnail preview of the last saved view of a selected .xrl file.

Summary Information – Displays all entity information for a model being opened.

Note:

The summary information for a file can be set in the **File Summary** information box.

See Also

[File > Save](#)

[File > Save As](#)

[File > Recent Files](#)

[File > Exit](#)

7.1.3. File > Save

What

The **Save File** command saves an existing xrl file.

Why

The **Save File** command is useful for:

- Saving files native to the application
- Saving files compatible with the application

How to Use

How to save a file

1. Go to **File > Save** or click the icon in the toolbar, or use the **Ctrl + S** keyboard shortcut.

Note:

To save a copy of the current model use the **File > Save As** command.

2. A dialog box will appear for any existing open models that are a format other than the .xrl format. This dialog box will not appear if the model was originally created in the .xrl.

Note:

The directory of the last saved file will be remembered and shown when the Save File command is executed.

3. Input a name for the file and click **Save** to save the model or click **Cancel** to quit the command.

Note:

If a command is not finished, a rollback is being executed, or the model is in any mode, a prompt box will appear asking if the changes for that command want to be discarded. Finish the current command, rollback or exit the current mode to save, or click **Cancel** to cancel the save command.

See Also

[File > Save As](#)

7.1.4. File > Save As

What

The **Save As** command saves a copy of the existing .xrl file.

Why

The **Save As** command is useful for:

- Saving a file without closing it
- Saving a file as a different file type

How to Use

How to save a file using the Save As command

1. Go to **File > Save As** or click the icon in the toolbar.

Note:

To save the current model without a copy use the **File > Save** command.

2. A save as dialog box will appear
3. Input a name for the file and click **Save** to save the model or click **Cancel** to quit the command.

Note:

The directory of the last saved file will be remembered and shown when the Save File command is executed.

See Also

[File > Save](#)

7.1.5. File > TeamPlatform

What

The **TeamPlatform** command allows users to upload and download .xrl files to TeamPlatform, Geomagic's web collaboration system.

Why

The **TeamPlatform** command is useful for:

- Saving a project to the TeamPlatform system for sharing with other team members
- Allowing collaboration on projects independent of time and space restrictions

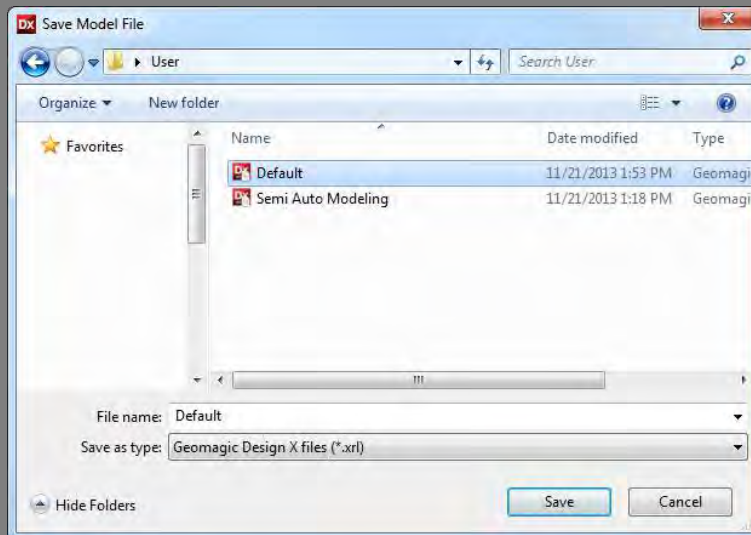
How to Use

How to upload a project to TeamPlatform

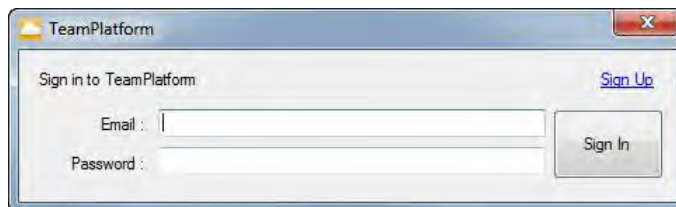
1. Go to **File > TeamPlatform > Upload**.

Note:

To upload a file to TeamPlatform it must first be saved to the local PC. If a project is being uploaded for the first time a Save Model File dialog box will be displayed.



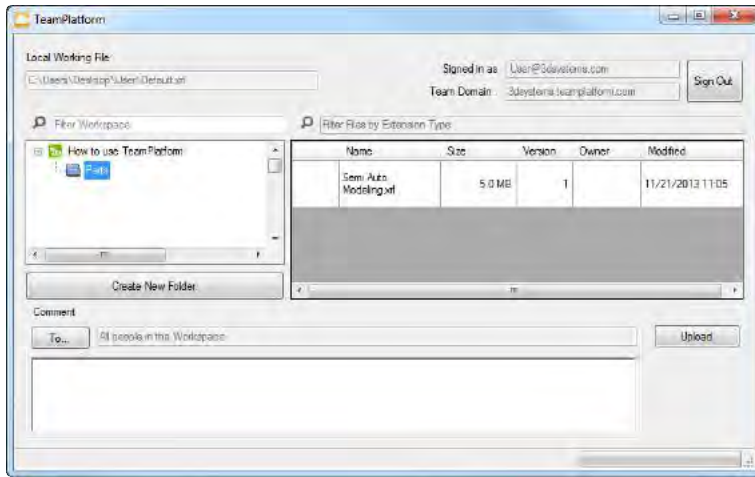
2. Enter a valid email address and password then click the **Sign In** button.



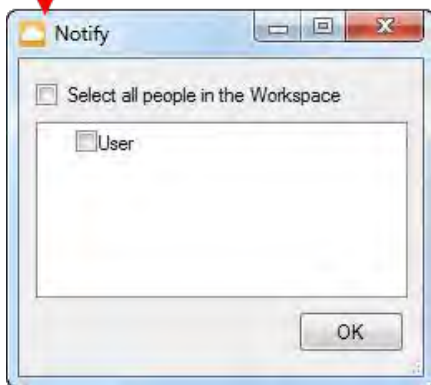
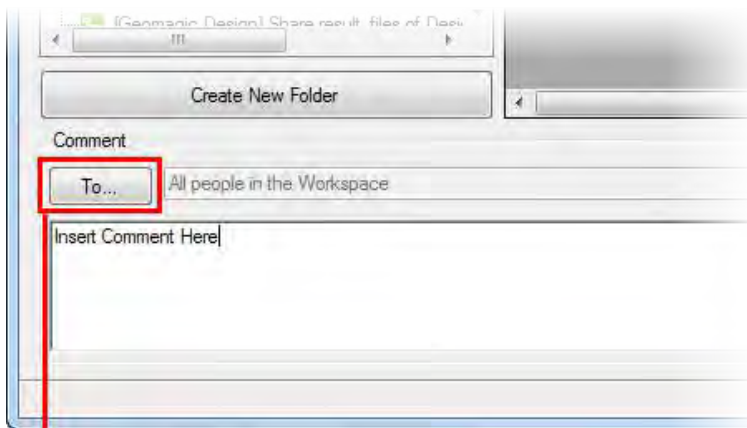
Note:

Click the Sign Up link to create a new account if you do not have an existing account.

3. Select a workspace to upload the file to after logging in. Navigate available workspaces and their subfolders using the left-hand navigation pane.



4. If necessary, click the **To...** button and select workspace members to send a comment to and input a comment in the comment box.



Note:

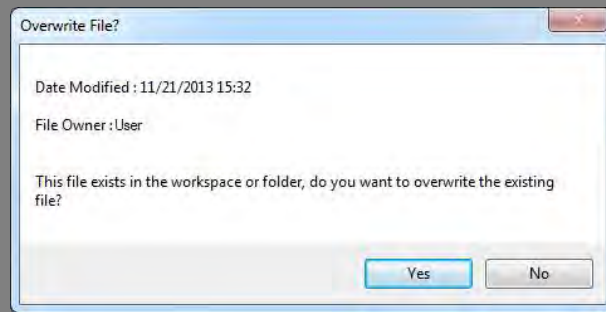
By default, all people in the workspace will receive any comments made.

5. Click the **Upload** button to upload the project file to the selected workspace. The **Cancel** button can be used to stop an upload.

Note:

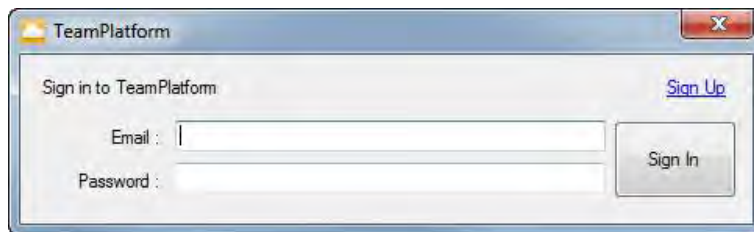
If the same file already exists in the TeamPlatform workspace and you try to upload the file

from another location a warning dialog box will be displayed.



How to download a file from TeamPlatform and open it in the application

1. Go to **File > TeamPlatform > Download**.
2. Enter a valid email address and password then click **Sign In**.



Note:

If it is the first time signing into TeamPlatform, click the Sign Up link to create a new account.

3. After logging in a dialog box will be displayed for selecting a Local Working Folder and parts from a workspace. Select a Local Working Folder.



Note:

All files in the Local Working Folder will have an accompanying .TP file with the same name. They are used to maintain TeamPlatform association and if a project file is moved to another location it is important to also move the associated .TP file with it.

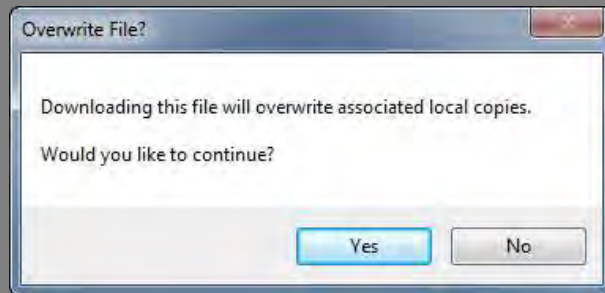
4. After selecting a Local Working Folder, navigate to the TeamPlatform workspace on the left pane of the dialog box and select a file to download.



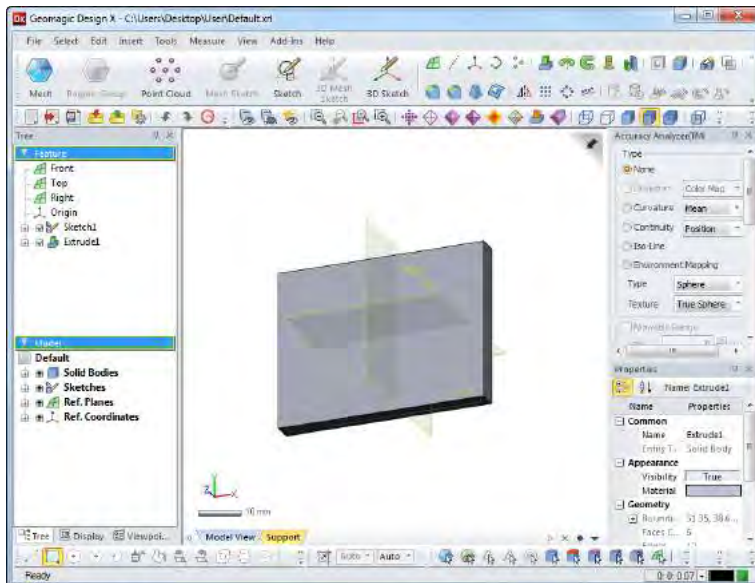
5. Click the **Download** button.

Note:

If a file with the same name exists in the specified local folder, a dialog box asking to overwrite the local copy will be displayed.



6. The file will be saved in the Local Working Folder then opened in the application.



See Also

- [File > Save](#)
- [File > Save As](#)

7.1.6. File > Export

What

The **Export** command saves selected entities in various formats.

Why

The **Export** command is useful for:

- Exporting a file into a different file type
- Exporting a file into another program

How to Use

How to export a file

1. Go to **File > Export** or click the icon in the toolbar. The application will show the default extension automatically in the **Export** dialog box which can be changed. Entities can be first selected then exported.

Note:

Supported extensions for selected entities are listed in the following table:

Export File		Entity								
Extension	Format	Point Cloud	Mesh	Body	2D/3D Sketch	Ref. Geometry	Color Vertex	Color Face	Texture Vertex	Texture Face
xdl	XO Model	Yes	Yes	Yes	Yes	Yes	Yes			
mdl	Rapidform Model	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes
xpc	Rapidform Extreme Point	Yes								
pts	Geomagic Point	Yes	Yes				Yes			
fcs	Geomagic Polygon		Yes				Yes	Yes		
icf	INUS Compression File	Yes	Yes				Yes	Yes	Yes	Yes
asc	Ascii Points	Yes	Yes				Yes			
stl	Binary STL		Yes							
stl	Ascii STL		Yes							
obj	OBJ		Yes				Yes		Yes	Yes

dxfl	AutoCAD DXF	Yes	Yes		Yes	Yes (Ref. Polyline Only)				
3ds	3D Studio	Yes	Yes						Yes	
wrl	VRML 1.0	Yes	Yes						Yes	Yes
wrl	VRML 97	Yes	Yes				Yes		Yes	Yes
ply	CyberWare Binary	Yes	Yes				Yes			
ply	CyberWare Ascii	Yes	Yes				Yes			
ptc	Kubit	Yes	Yes				Yes			
ptc	Leica	Yes	Yes				Yes			
igs	IGES			Yes	Yes	Yes				
stp	STEP			Yes	Yes	Yes				
x_t	Parasolid Text			Yes	Yes					
X_b	Parasolid Binary			Yes	Yes					
sat	ACIS Text			Yes	Yes	Yes				
sab	ACIS Binary			Yes	Yes	Yes				
model	CATIA V4			Yes	Yes	Yes				
catpart	CATIA V5			Yes	Yes	Yes				

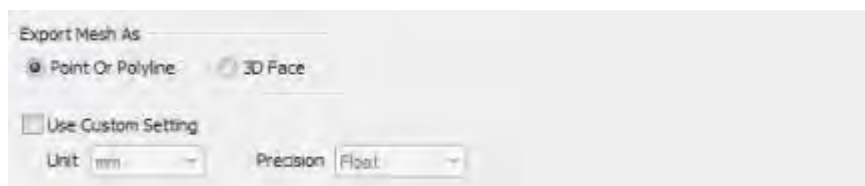
Note:

To export the file formats listed in the table below a Geomagic Exchange™ License must be installed:

CAD	File Extension	Default Export Version	Comments
CATIA V4	.model	CATIA 4.1.9	
CATIA V5	.CATPart, .CATProduct	CATIA V5 R6	Unsupported assembly non-ASCII character in the file name

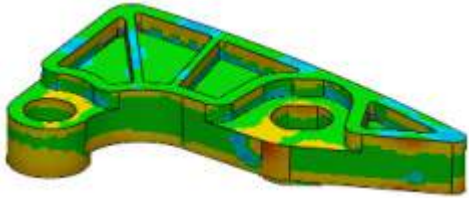
2. Click the **OK** button to export the selected entity or click the **Cancel** button to quit the command.

Options in Detail



Export Accuracy Analyzer Color Map Of Body As Mesh Format – Exports a body as a colored mesh. This option is available in the Export dialog tree when the Accuracy Analyzer™ is activated and a body has been selected. This option will not appear from the pop-up menu on the **Model**

tree.



Point or Polyline – Creates a poly-face mesh for AutoCAD. This option is available when exporting a mesh in the .dxf format.

3D Face – Creates a 3D face mesh for AutoCAD. This option is available when exporting a mesh in the .dxf format.

Use Custom Setting – Enables the **Unit** and **Precision** options.

Unit – Defines a measurement unit for the entity to be exported.

Precision – Sets the precision for exported files. This option is enabled when the precision of an imported point cloud or mesh is Double and exported in the ASCII format.

Export Each Mesh As Separate File – Saves several meshes as individual files instead of a single file.

Use Mesh Name – Saves each file with the name of the mesh.

ICF Geometry Compression Ratio – Defines the resolution of a saved model. When **Lossless** is selected, geometry will be preserved as it is, but this can increase the size of a file.

See Also

[File > Save](#)

[File > Save As](#)

7.1.7. File > Publish

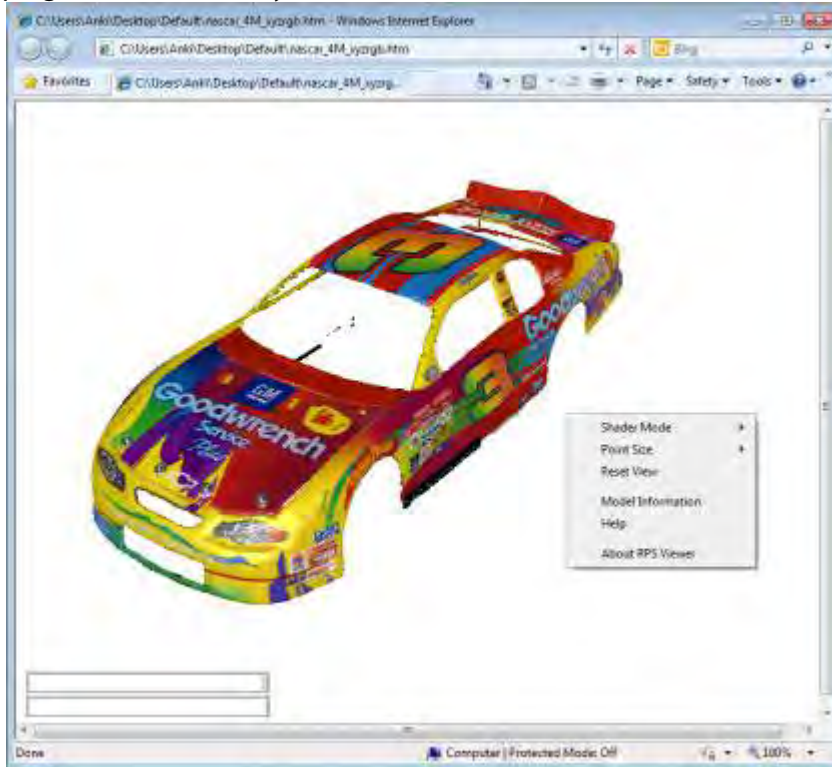
The **Publish** menu publishes mesh through Internet Explorer by using the Point Stream and ICF publishing commands.

File > Publish > Point Stream

What

The **Publish Point Stream** command publishes selected meshes and point clouds as a stream of points that can be viewed using the Internet Explorer browser. The RPS internet explorer viewer

plug-in is automatically installed when a PC is connected to the internet.



Why

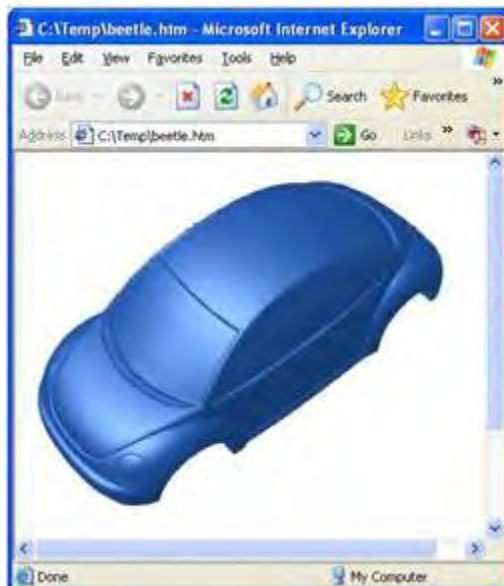
The **Publish Point Stream** command is useful for:

- Publishing a file into a Point Stream to be viewed in an Internet Explorer web browser

How to Use

A. How to publish a file as a point stream

1. Go to **File > Publish > Point Stream** and select a mesh or a point cloud to publish.



2. Click the **OK** button to publish the mesh. An .html and .rps file will be created that is viewable in an Internet Explorer browser window. The same mouse and keyboard controls used to rotate, pan and zoom in the application are also used in the browser.
3. To see the saved file, open the .html file.

If this file is being used for the first time a dialog box used to install the RPS viewer plug-in will be displayed. Click **Install** to install and proceed.

Note:

This plug-in is not supported on 64-bit Internet Explorer.

4. To control viewing:
Rotate: Left mouse button
Pan: Ctrl + Left mouse button
Zoom: Shift + Left mouse button or mouse wheel

B. How to share a generated file

Uploading on a website

The simplest way is to upload both the RPS and HTML files to a webhost, usually with an FTP client, in the same folder. Then link to the HTML file's location

To store the RPS and HTML file in a different folder, open the HTML file with a text editor and change the RPSFilePath parameter's value to correct location. (Relative path from the directory of the HTML file is in).

Inserting into another html file

To insert the RPS view of your model into another web page, just copy tags everything in the HTML file from "<OBJECT>" to "</OBJECT>", and paste them into another HTML source file.

Options in Detail

Plug-in – Shows the download path for the plug-in.

Version – Shows the current version of the RPS viewer.

Width and Height – Shows the width and height of the viewing area of a file.

HTML File Path – Specifies the name and directory of a HTML document to be generated.

Background – Specifies the background color of the RPS viewer. The RGB value can be between 0 and 255. The **Use Texture** option assigns a specific image as the background.



Without Texture



With Texture

Antialiasing – Enables or disables anti-aliasing when a model is in a static state in the RPS viewer.

Zoom Limit – Controls the range of zoom-in. **No Limit** allows infinite zoom, **Bounding Sphere** allows zooming up to the boundary of a boundary sphere, **Nearest Distances** allows zooming to a nearest distance and **Limit Factor** scales zooming distance for the **Bounding Sphere** and **Nearest Distance** options.

Launch Internet Explorer – Specifies if a .html file will be opened in Internet Explorer upon publishing.

Using the Point Stream Viewer

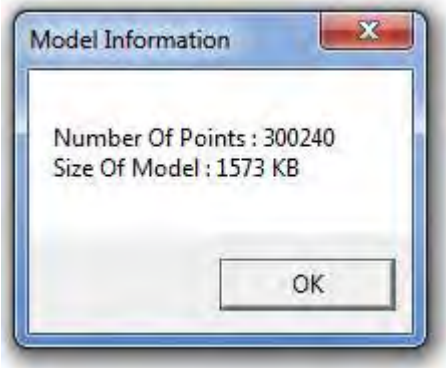


Shader Mode – Sets various shader mode settings. For more information, see [Display tab](#) page.

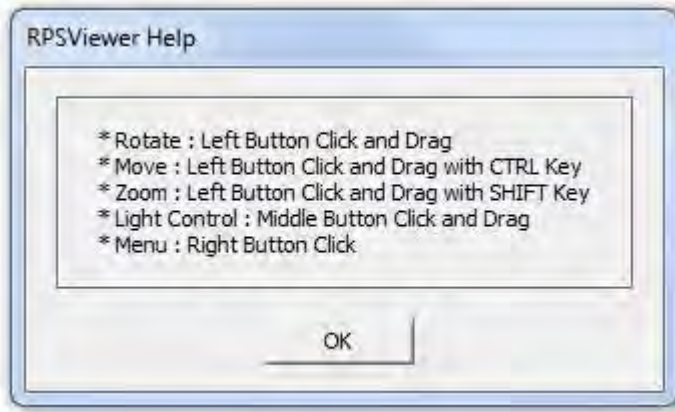
Point Size – Defines the drawing size of points.

Reset View – Resets the view to the default view.

Model Information – Displays the **Number Of Points** and **Size Of Model**.



Help – Displays the help panel.



About RPS Viewer – Displays the RPS Viewer information.

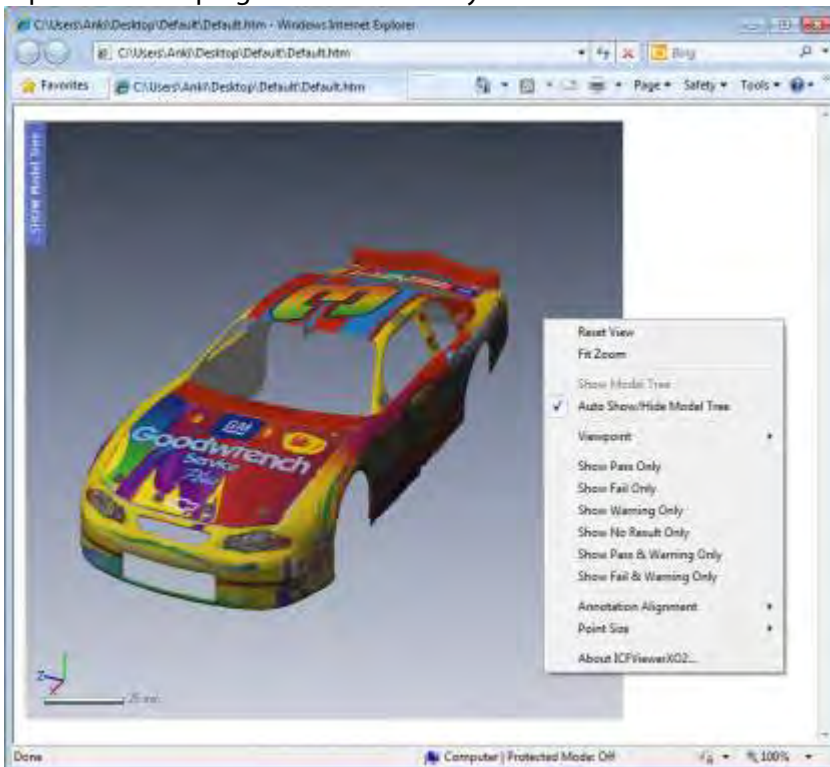
See Also

[Tools > Publish > ICF](#)

File > Publish > ICF

What

The **Publish ICF** command publishes selected entities (point cloud, mesh, surface or solid body) in the .icf file format, enabling them to be viewed by the Internet Explorer browser. The ICF internet explorer viewer plug-in is automatically installed when a PC is connected to the internet.



Why

The **Publish ICF** command is useful for:

- Publishing a file into a .icf file to be viewed in the Internet Explorer web browser
- Measuring distances on a model in a web browser

How to Use

A. How to publish a file as an .ICF file

1. Go to **File > Publish > ICF** and select point cloud, mesh, surface or solid body to publish.



2. Click the **OK** button to publish the mesh. A .html and .icf file will be created that is viewable in an Internet Explorer browser window. The same mouse and keyboard controls used to rotate, pan and zoom in the application are also used in the browser.

Display properties of the web browser can be altered by changing the .html file that is created.

```
<HTML><BODY><P>  
<OBJECT ID="ICFViewerXO2"  
  CODEBASE="http://www.rapidform.com/icfviewerxo2#VERSION=1,0,0,16"  
  CLASSID=clsid:E2B55A3D-1A41-4F58-A675-FFD34F45CBA0  
  HEIGHT=600 WIDTH=600>  
<PARAM NAME="IcfPath" VALUE="Sample Model.icf">  
<PARAM NAME="EnableDynamicDraw" VALUE="true">  
<PARAM NAME="DynamicDisplayRatio" VALUE="25">  
<PARAM NAME="StaticDisplayRatio" VALUE="100">  
<PARAM NAME="ShowColorBar" VALUE="false">  
</OBJECT>  
</P></BODY></HTML>
```

The following table shows the parameter list you can use in the html code.

PARAM NAME	VALUE	Default Value
IcfPath	Location of ICF file	
EnableDynamicDr	"true" or "false"	"true"

aw		
DynamicDisplayRatio	"100", "50", "25", "10", "5", "1", "0.5", "0.1", "auto" (identical with the value of xo)	"25"
StaticDisplayRatio	"100", "50", "25", "10", "5", "1", "0.5", "0.1" (identical with the value of xo)	"100"
SHowColorBar	"true" or "false"	"true"

3. To see the saved file, open the .html file.

If this file is being used for the first time a dialog box used to install the ICF viewer plug-in will appear. Click **Install** to install and proceed.

Note:

This plug-in is not supported on 64-bit Internet Explorer.

4. To control viewing:
 Rotate: Right mouse button
 Pan: Right + Left mouse button
 Zoom: Right + Left mouse button or mouse wheel

B. How to share a generated file

Uploading on a website

The simplest way is to upload both the ICF and HTML files to a webhost, usually with an FTP client, in the same folder. Then link to the HTML file's location

To store the ICF and HTML file in a different folder, open the HTML file with a text editor and change the "IcfPath" parameter's value to correct location. (Relative path from the directory of the HTML file is in).

Inserting into another html file

To insert the ICF view of your model into another web page, just copy tags everything in the HTML file from "<OBJECT>" to "</OBJECT>", and paste them into another HTML source file.

Options in Detail

Plug-in – Shows the location of the plug-in codebase.

Version – Shows the current version of the .icf viewer.

Width and Height – Shows the width and height of the viewing area of the file.

Fit Screen Size to Window on Resize – The viewer size is fully fit to the Explorer window size. When you adjust the Explorer window size, the viewer size is adjusted according to the Explorer window size.

HTML File Path – Specifies the name and directory of a HTML document to be generated.

Enable Dynamic Draw – If checked, the **Dynamic Display Ratio** will be enabled.

Dynamic Display Ratio – Changes the amount points drawn while a model is in motion.

Static Display Ratio – Changes the amount points drawn while a model is static.

Geometry Compression Ratio – Defines the resolution of the saved model. If **Lossless** is set, the geometry will be preserved as it is. However its file size will be much bigger than the reset settings.

Launch Internet Explorer – Specifies if the .icf file will be opened in Internet Explorer upon publishing.

Using the ICF Viewer



Reset View – Resets the view to the default view.

Fit Zoom – Fits the view to the screen.

Show Model Tree – Shows the model tree.

Auto Show / Hide Model Tree – Shows the collapsible model tree at left side. When you hover mouse cursor on the Show Model Tree, the tree appears.

Viewpoint – Changes the viewing direction of the model.

Point Size – Defines the drawing size of points.

About ICFViewerXO2 Control – Displays the ICF Viewer information.

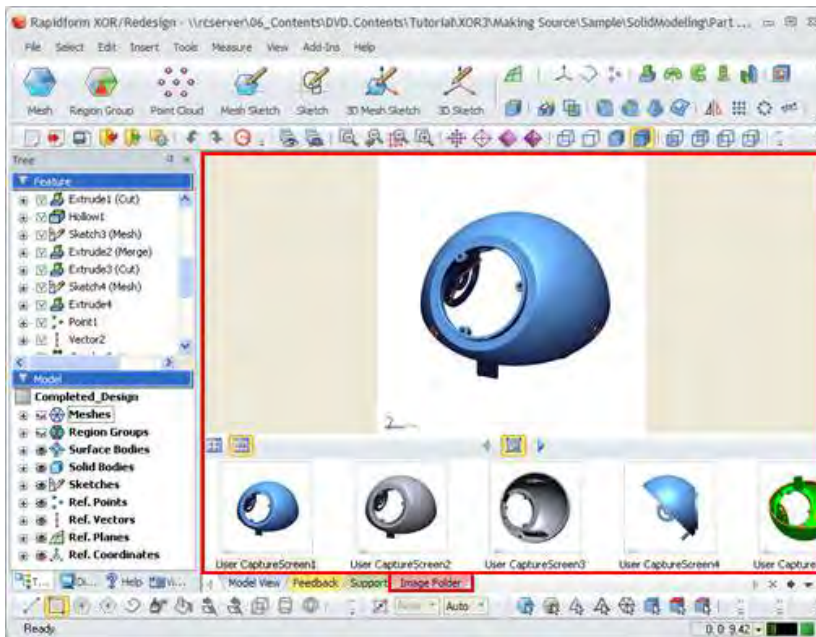
See Also

[File > Publish > Point Stream](#)

7.1.8. File > Capture Screen

What

The **Capture Screen** command creates a screen shot using a variety of methods. The captured images are registered in the Image folder.



Why

The **Capture Screen** command is useful for:

- Capturing a model in its current state
- Capturing an image of a model to use in another program

How to Use

How to capture a file

1. Go to **File > Capture Screen** or click the icon in the toolbar. The **Capture Screen** dialog box will appear.

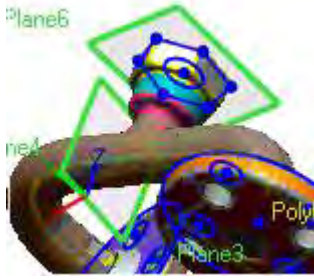
Note:

The captured images saved to the Image Folder can be set to be shown as thumbnails or filmstrip view.

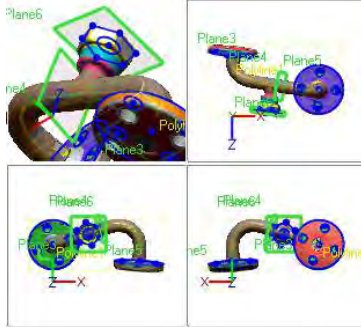
2. Select **Current View** as the **Target** method.
3. Click the **OK** button to save the image to the Image Folder.

Options in Detail

Current View – Captures the current view.



Entire Model View – Captures the entire model view.



Mouse Dragging Area – Captures the area defined by a mouse drag.

Selected Window – Captures a selected window.

Application Window – Captures an entire application window.

White Background – Sets the background of the capture to white and is available if the **Target** is **Current View**, **Entire Model View**, or **Mouse Dragging Area**.

Transparent Background – Sets the background of the capture to transparent and is available if the **Target** is **Current View**, **Entire Model View**, or **Mouse Dragging Area**. Transparent background is preserved when the exported image format is .tiff or .png.

Custom Resolution – Sets a custom resolution with values between 20 and 8192 and is available if the **Target** is **Current View**, **Entire Model View**, or **Mouse Dragging Area**.

Show Dialog Tree – Captures the dialog tree and is available if the **Target** is **Current View**, **Entire Model View**, or **Mouse Dragging Area**.

Copy To Windows Clipboard – Copies a captured image to the windows clipboard, allowing pasting into other applications.

See Also

[Docking Bars > Image Folder](#)

7.1.9. File > Print

What

The **Print** command prints the active document.

Why

The **Print** command is useful for:

- Printing a current project

How to Use

How to print a file

1. Go to **File > Print** or click the icon in the toolbar, or use the **Ctrl + P** keyboard shortcut.
2. Select the **Printer**, **Print range** and **Number of copies**.
3. Click **OK** to complete the command.

See Also

[File > Print Preview](#)

[File > Print Setup](#)

7.1.10. File > Print Preview

What

The **Print Preview** command previews an active document before printing.

Why

The **Print Preview** command is useful for:

- Previewing what a print-out will look like before printing

How to Use

How to print preview a file

1. Go to **File > Print Preview** or click the icon in the toolbar.

2. A preview of the active document will be shown.
3. Click **Print** to print the document or **Close** to exit without printing.

See Also

[File > Print](#)

[File > Print Setup](#)

7.1.11. File > Print Setup

What

The **Print Setup** command sets printer settings.

Why

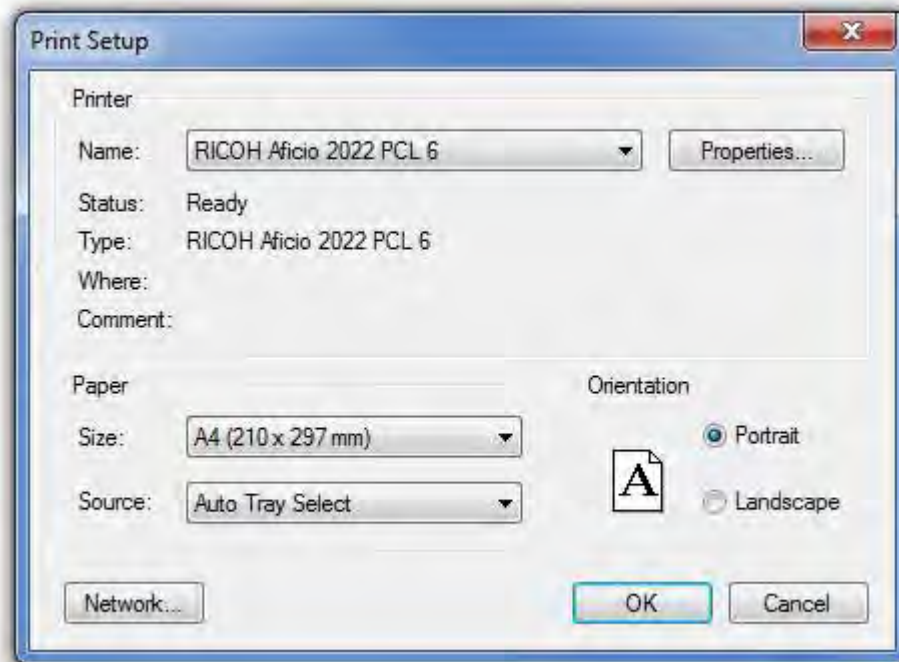
The **Print Setup** command is useful for:

- Changing printers
- Changing printer properties
- Changing paper size

How to Use

How to use print setup

1. Go to **File > Print Setup** or click the icon in the toolbar.
2. A preview of the active document will be shown.



3. Change settings and click **OK** to accept or **Cancel** to exit **Print Setup**.

See Also

[File > Print](#)

[File > Print Preview](#)

7.1.12. File > LiveTransfer(TM)

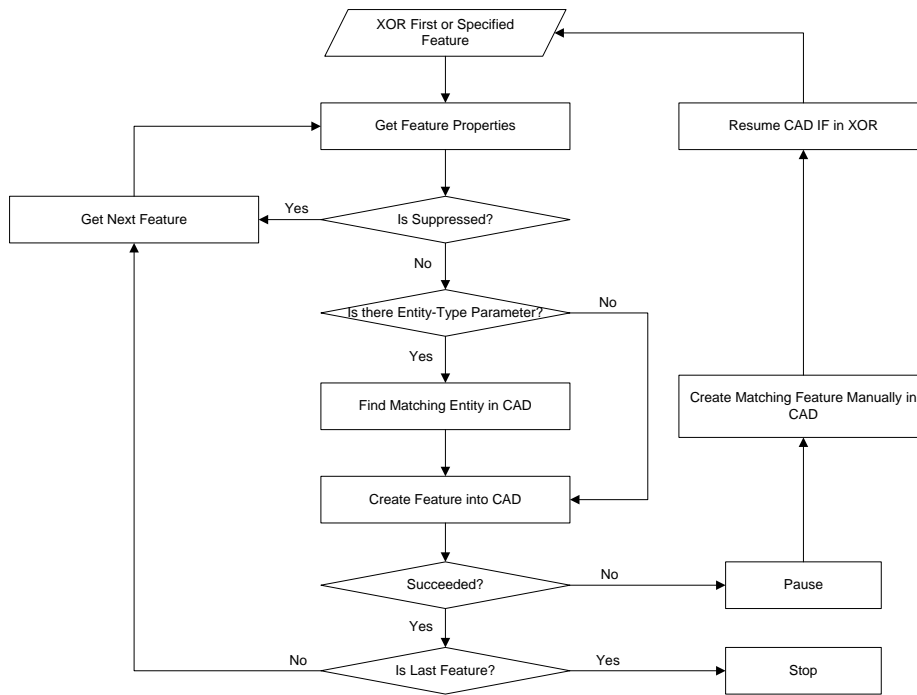
What

The **LiveTransfer™** command directly exports whole or partial data created in the application to target CAD systems using the API (Application Programming Interface) for many different CAD systems.

The **LiveTransfer™** command transfers the geometries, topologies, modeling histories, and parameters of a modeling job to a CAD system that can then be edited.

If features cannot be transferred into a CAD system while using the **LiveTransfer™** command, the transfer will pause so that the features can be manually added into the CAD system. The transfer can be resumed afterward.

The following figure shows the work flow of the **LiveTransfer™** function.



Note:

Setting the **Preferred CAD** as one of the following CAD systems: (SolidWorks, Siemens NX, Creo(Pro/E), AutoCAD, Inventor), or checking the **Show LiveTransfer™** option in the **Error** list will allow the real-time analysis of feature compatibility.

Why

The **LiveTransfer™** command is useful for:

- Transferring data within the application to CAD systems
- Transferring data easily using native CAD API's

Summary of CAD Compatibility in LiveTransfer™

Geomagic Design X LiveTransfer™ supports **SolidWorks 2006~2013, SIEMENS NX4~8, Pro/E Wildfire3.0~5.0, Creo Parametric 1.0~2.0, Creo Elements/Pro 5.0, AutoCAD 2007~2011 and Inventor 2010~2013.**

Table Legend:

Yes: fully supported

Possible: supported, but not all options

Object: supported, but non-parametric as imported dummy feature

Not Now: not support now, but will be supported in a future release

No: downstream CAD software does not support the related API

APPLICATION		SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Sketch	2D (Mesh) Sketch	Possible	Possible	Possible	Possible	Possible
	3D (Mesh) Sketch	Object	Object	Object	Object	Object
Solid	Extrude	Possible	Possible	Possible	Possible	Possible
	Revolve	Possible	Possible	Possible	Possible	Possible
	Loft	Possible	Possible	Object	Possible or	Possible or

					Object	Object
	<u>Sweep</u>	Possible	Possible	Object	Possible or Object	Possible or Object
	<u>Cut</u>	Possible	Possible	Possible	Possible	Possible
	<u>Hollow</u>	Yes	Yes	Yes	No	Possible
	<u>Offset</u>	No	Yes	No	No	Yes
	<u>Thicken</u>	Yes	Possible	Possible	No	Yes
	<u>Boolean</u>	Possible	Possible	No	Possible	Possible
Surface	<u>Surface Extrude</u>	Possible	Possible	Possible	Possible	Possible
	<u>Surface Revolve</u>	Possible	Possible	Possible	Possible	Possible
	<u>Surface Loft</u>	Possible	Possible	Object	Possible or Object	Possible or Object
	<u>Surface Sweep</u>	Possible	Possible	Object	Possible or Object	Possible or Object
	<u>Reverse Normal</u>	No	Yes	No	No	Object
	<u>Sew</u>	Possible	Possible	No	No	Possible
	<u>Trim</u>	No	Possible	Possible	No	Yes
	<u>Untrim</u>	Possible	No	No	No	No
	<u>Trim & Merge</u>	Object	No	No	No	No
	<u>Surface Offset</u>	No	Possible	Object	No	Object
	<u>Extend Surface</u>	Object	Object	Object	Object	Object
	<u>Auto Surfacing</u>	Object	Object	Object	Object	Object
	<u>Fit Boundary</u>	Object	Object	Object	Object	Object
	<u>Mesh Fit</u>	Object	Object	Object	Object	Object
<u>Fill Face</u>	No	No	No	No	Possible	
Modeling Feature	<u>Fillet</u>	Possible	Possible	Possible	Possible	Possible
	<u>Chamfer</u>	Possible	Possible	Possible	Possible	Possible
	<u>Draft</u>	Possible	Possible	Possible	No	Possible
	<u>Mirror</u>	Possible	Possible	Object	Object	Possible
	<u>Delete Face</u>	Possible	No	No	No	Possible
	<u>Move Face</u>	Possible	Possible	No	No	Possible
	<u>Refit Face</u>	No	No	No	No	No
	<u>Replace Face</u>	Possible	Possible	Possible	No	Object
	<u>Linear Pattern</u>	Possible	Object	Object	Object	Possible
	<u>Circular Pattern</u>	Possible	Object	Object	Object	Possible
	<u>Curve Pattern</u>	Possible	Object	Object	Object	Object
	<u>Transform Body</u>	Possible	No	No	No	Possible
	<u>Duplicate Body</u>	Yes	Object	No	No	Object
	<u>Remove Body</u>	Yes	No	No	No	Yes
	<u>Split Face</u>	Possible	Possible	No	No	Possible
	<u>Helix</u>	Object	Object	No	No	Object
<u>Spiral</u>	Object	Object	No	No	Object	
<u>Emboss</u>	Possible	Possible	No	No	Possible	
Reference Geometry	<u>Ref. Point</u>	No	Possible or Object	Possible or Object	No	Object
	<u>Ref. Vector</u>	Possible or Object	Possible or Object	Possible or Object	No	Possible or Object
	<u>Ref. Plane</u>	Possible or Object	Possible or Object	Possible or Object	No	Possible or Object
	<u>Ref. Coordinate</u>	No	Possible or Object	Possible or Object	No	Possible or Object
	<u>Ref. Polyline</u>	No	No	No	No	No
Object Body	Object	Object	Object	Object	Object	

Details of CAD Compatibility

Table Legend:

Yes: supported

Possible: supported, but not exact same parameters

Object: supported, but non-parametric as imported dummy feature

Note:

Sketch/Mesh Sketch features are transferred as parametric features, however, “Imported” entities inside features are non-parametric. When the “Region” option is an “Object” in the table, the feature will be transferred as a parametric feature, but only the Region option will be non-parametric.

Not now: not support now, but will be supported in a future release

No: downstream CAD software does not support the related API

Sketch/Mesh Sketch

APPLICATION		SolidWorks 2006 +	Siemens NX 4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Base Plane	Ref. Plane	Yes	Yes	Yes	Yes	Yes
	Face	Yes	Yes	Yes	Yes	Yes
	Sketch Chain	Object	Object	Object	Yes	Object
	Curve	Object	Object	Object	Yes	Object
	Sketch	Object	Object	Object	Yes	Object
	3D Sketch	Object	Object	Object	Yes	Object
	Edge	Object	Object	Object	Yes	Object
	Loop	Object	Object	Object	Yes	Object
Region	Object	Object	Object	Yes	Object	
Entity	Point	Yes	Yes	Yes	Yes	Yes
	Line	Yes	Yes	Yes	Yes	Yes
	Arc	Yes	Yes	Possible	Yes	Yes
	Circle	Yes	Yes	Yes	Yes	Yes
	Ellipse	Yes	Yes	Possible	Yes	Yes
	Partial Ellipse	Yes	Yes	Possible	Yes	Yes
	Interpolated Spline	Object	Object	Possible	Object	Object
	Canonical Spline	Yes	Yes	Possible	Yes	Object
	Text	Object	Object	Possible	Object	Object
	Fit Spline	Object	Object	Possible	Object	Object
Parabola	Object	Object	Object	Object	Object	
Property	“Constructive” option	Yes	Yes	No	No	Yes
Constraint	Coincidence	Yes	Possible	No	No	Yes
	Concentric	Yes	Possible	No	No	Yes
	Perpendicular	Yes	Yes	No	No	Yes
	Parallel	Yes	Yes	No	No	Yes
	Tangent	Yes	No	No	No	Yes
	Equal Length	Yes	Yes	No	No	Yes
	Equal Radius	Yes	Yes	No	No	Yes
	Equal Curvature	No	No	No	No	No
	Coradial	Yes	Possible	No	No	Yes
	Colinear	Yes	Yes	No	No	Yes
	Fix	Yes	Possible	No	No	No
	Horizontal	Yes	Yes	No	No	Yes
	Vertical	Yes	Yes	No	No	Yes
	Linear Aligned Dimension	Yes	Yes	Yes	No	Yes
	Linear Horizontal Dimension	Yes	Yes	Yes	No	Yes
	Linear Vertical Dimension	Yes	Yes	Yes	No	Yes
	Radial Dimension	Yes	Yes	Yes	No	Yes
Angular Dimension	Yes	Possible	Yes	No	No	
Mid Point	No	Possible	No	No	No	

	Normal	No	No	No	No	No
	Conversion	No	No	No	No	No
	Equal Distance	No	No	No	No	No
	Linear Pattern	No	No	No	No	Object
	Circular Pattern	No	No	No	No	Object
	Symmetry	No	No	No	No	Object
	Offset	No	No	No	No	Object
	Removed Original Offset	No	No	No	No	Object

3D Sketch/3D Mesh Sketch

APPLICATION	SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Curve	Object	Object	Object	Yes	Object

Extrude

APPLICATION		SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +	
Profiles	Sketch	Yes	Yes	Yes	Yes	Yes	
	3D Sketch	No	No	No	No	No	
	Whole Sketch	Yes	Yes	Yes	Yes	Yes	
	Part Of Sketch	No	No	No	No	No	
	Multi Profiles	Yes	Yes	Yes	Yes	Yes	
Custom Direction	Ref. Vector	Yes	Yes	No	Possible	Possible	
	Ref. Plane	No	Yes	No	Possible	Possible	
	Sketch Chain	No	No	No	Possible	Possible	
	Curve	No	Yes	No	Possible	Possible	
	Sketch	No	Yes	No	Possible	Possible	
	Region	No	No	No	Possible	Possible	
	3D Sketch	No	Yes	No	Possible	Possible	
	Edge	Yes	Yes	No	Possible	Possible	
	Loop	No	No	No	Possible	Possible	
Face	No	Yes	No	Possible	Possible		
Blind	Length(Flip)	Yes	Yes	Yes	Yes	Yes	
	Draft(Angle/Flip)	Yes	Yes	No	Yes	Yes	
Through All	No Option	Yes	Possible	Yes	Yes	Yes	
	Draft(Angle/Flip)	Yes	Yes	No	Yes	Yes	
Up To Vertex	Vertex	Ref. Point	No	Possible	No	Possible	Possible
		Ref. Coordinate	No	Possible	No	Possible	Possible
		Curve Point	No	Possible	No	Possible	Possible
		Body Vertex	Yes	Possible	Possible	Possible	Possible
	Offset(Dist/Flip)	No	Yes	No	Possible	Possible	
	Draft(Angle/Flip)	Yes	Yes	No	Possible	Yes	
Up To Region (Trim with ...)	Region	Object	Object	Object	No	Object	
	Offset(Dist/Flip)	Yes	No	No	No	No	
	Trans(Dist/Flip)	Yes	No	No	No	No	
	Draft(Angle/Flip)	Yes	Yes	No	No	Yes	
Up To Region (Max.Mean.Min.)	Min, Max, Mean	Possible	Possible	Possible	No	Possible	
	Draft(Angle/Flip)	Yes	Yes	No	No	Yes	
Up To Surface	Surface	Ref. Plane	Yes	Yes	Yes	No	Yes
		Face	Yes	Yes	Yes	No	Yes
	Offset(Dist/Flip)	Yes	No	No	No	No	
	Trans(Dist/Flip)	Yes	No	No	No	No	
	Draft(Angle/Flip)	Yes	Yes	No	No	Yes	
Up To Body	Body	Surface Body	Yes	Yes	No	No	Yes
		Solid Body	Yes	Yes	No	No	Yes
	Draft(Angle/Flip)	Yes	Yes	No	No	Yes	
Mid Plane	Length(Flip)	Yes	Possible	Yes	Yes	Yes	

	Draft(Angle/Flip)	Yes	Yes	No	Yes	Yes
Result Operations	Solid	Yes	Yes	Possible	Yes	Yes
	Cut	Yes	Possible	Yes	Possible	Yes
	Merge	Yes	Possible	Yes	Possible	Yes
	Opposite	Yes	Possible	Yes	Yes	Yes

Revolve

APPLICATION		SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Profiles	Sketch	Yes	Yes	Yes	Yes	Yes
	3D Sketch	No	No	No	No	No
	Whole Sketch	Yes	Yes	Yes	Yes	Yes
	Part Of Sketch	No	No	No	No	No
	Multi Profiles	Yes	No	Yes	Yes	Yes
Axis	Ref. Vector	Yes	Yes	Yes	Possible	Yes
	Curve	Yes	Yes	Yes	Possible	Yes
	Region	No	Object	No	Possible	No
	Edge	Yes	Yes	Yes	Possible	No
	Face	No	Yes	No	Possible	No
One Direction	Angle(Flip)	Yes	Yes	Yes	Yes	Yes
Mid Plane	Angle(Flip)	Yes	Yes	Yes	Yes	Yes
Both Directions	Angle(Flip)	Yes	Yes	Yes	Yes	Yes
	Opposite Angle	Yes	Yes	Yes	Yes	Yes
Result Operations	Solid	Yes	Yes	Possible	Yes	Yes
	Cut	Yes	Possible	Yes	Possible	Yes
	Merge	Yes	Possible	Yes	Possible	Yes

Loft

Not supported in NX4 and Pro/E 3.0+.

APPLICATION		SolidWorks 2006 +	SIEMENS NX5 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Profiles	Composite Profile	No	No	No	No	No
	Sketch	Yes	Yes	No	Yes	Yes
	3D Sketch	Yes	Yes	No	Yes	Yes
	Whole Sketch	Yes	Yes	No	Yes	Yes
	Part Of Sketch	No	No	No	No	No
	Edge	Not now	Not now	No	No	No
	Loop	No	No	No	No	No
	Face	Not now	Not now	No	No	Yes
Constraint	None	Yes	Yes	No	Yes	Yes
	Direction Vector	Not now	No	No	No	No
	Normal To Profile	Not now	No	No	No	Possible
	Tangency To Face	No	No	No	No	Yes
	Curvature to Face	No	No	No	No	Yes
Guide Curves	Composite Profile	No	No	No	No	No
	Sketch	Yes	Yes	No	Yes	Yes
	3D Sketch	Yes	Yes	No	Yes	Yes
	Whole Sketch	Yes	Yes	No	Yes	Yes
	Part Of Sketch	No	No	No	No	No
	Edge	Not now	No	No	No	No
Merge Tangent Faces		No	No	No	No	Yes
Close Loft		Yes	No	No	No	Yes
Result Operations	Solid	Yes	Yes	No	Yes	Yes
	Cut	Yes	Possible	No	Possible	Yes
	Merge	Yes	Possible	No	Possible	Yes

Sweep

Not supported in NX4 and Pro/E 3.0+.

APPLICATION		SolidWorks 2006 +	SIEMENS NX5 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
-------------	--	----------------------	------------------	-----------------------	-------------------	--------------------

Profiles	Composite Profile	No	No	No	No	No
	Sketch	Yes	Yes	No	Yes	Yes
	3D Sketch	Yes	Yes	No	Yes	No
	Whole Sketch	Yes	Yes	No	Yes	Yes
	Part Of Sketch	No	No	No	No	No
	Edge	Not now	Not now	No	No	No
	Loop	No	No	No	No	No
Path	Composite Profile	No	No	No	No	No
	Sketch	Yes	Yes	No	Yes	Yes
	3D Sketch	Yes	Yes	No	Yes	Yes
	Whole Sketch	Yes	Yes	No	Yes	Yes
	Part Of Sketch	No	No	No	No	No
	Edge	Not now	Not now	No	No	No
Method	Follow Path	Yes	Yes	No	Yes	Yes
	Keep Normal Constant	Not now	No	No	No	Yes
	Follow Path And First Guide Curve	Not now	No	No	No	Yes
	Follow First And Second Guide Curves	Not now	No	No	No	No
	Twist Along Path	Not now	No	No	No	No
	Twist Along Path With Normal Constant	Not now	No	No	No	No
Path Alignment Type	None	Yes	Yes	No	Yes	Yes
	Direction Vector	Not now	No	No	No	No
Guide Curves	Composite Profile	Not now	No	No	No	No
	Sketch Chain	Not now	No	No	No	Possible
	Curve	Not now	No	No	No	Yes
	Sketch	Not now	No	No	No	Yes
	Sketch Loop	Not now	No	No	No	Possible
	3D Sketch	Not now	No	No	No	Yes
	Edge	Not now	No	No	No	No
Improve Curvature Flow Along Sweep Path	No	No	No	No	No	
Merge Tangent Faces	Not now	No	No	No	No	
Result Operations	Solid	Yes	Yes	No	Yes	Yes
	Cut	Yes	Possible	No	Possible	Possible
	Merge	Yes	Possible	No	Possible	Possible

Cut

APPLICATION		SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Tool Entities	Region	Object	Object	Object	Not Now	Object
	Ref. Plane	Yes	Yes	Yes	Not Now	Yes
	Sketch Chain	No	No	No	No	No
	Curve	No	No	No	No	No
	Sketch	No	No	No	No	No
	3D Sketch	No	No	No	No	No
	Edge	No	No	No	No	No
	Surface Body	Yes	Yes	Yes	Yes	Yes
	Multiple Tool Entities	No	Yes	No	No	Yes
Target Bodies	Solid Body	Yes	Yes	Possible	Yes	Yes

Hollow

APPLICATION		SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Body	Solid Body	Yes	Yes	Yes	No	Yes
Depth		Yes	Yes	Yes	No	Yes
Hollow Outward		Yes	Yes	Yes	No	Yes
Remove Faces	Faces	Yes	Yes	Yes	No	Yes
Multi-Thickness Faces	Face	Yes	Yes	Yes	No	No
	Thickness	Yes	Yes	Yes	No	No

Offset

APPLICATION	SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Solid Bodies	No	Yes	No	No	Yes
Offset Distance(Flip)	No	Yes	No	No	Yes

Thicken

APPLICATION		SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Surface Body		Yes	Yes	Yes	No	Yes
Thickness		Yes	Yes	Yes	No	Yes
Direction	Direction1	Yes	Yes	Yes	No	Yes
	Direction2	Yes	Yes	Yes	No	Yes
	Both	Yes	Yes	Yes	No	Yes
Result Operator	Solid	Yes	Possible	Possible	No	Yes
	Cut	Yes	Possible	Yes	No	Yes
	Merge	Yes	Possible	Yes	No	Yes

Boolean

APPLICATION			SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Merge	Tool Bodies	Surface Body	No	No	No	No	No
		Solid Body	Yes	Possible	No	Possible	Yes
Cut	Tool Bodies	Surface Body	No	No	No	No	No
		Solid Body	Yes	Possible	No	Possible	Yes
	Target Bodies	Solid Body	Yes	Possible	No	Possible	Yes
	Multiple Target Bodies		No	No	No	No	No
Intersect	Tool Bodies	Surface Body	No	No	No	No	No
		Solid Body	Yes	Possible	No	Possible	Yes
	Target Body	Solid Body	Yes	Possible	No	Possible	Yes

Surface Extrude

APPLICATION		SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +	
Profiles	Sketch	Yes	Yes	Yes	Yes	Yes	
	3D Sketch	No	No	No	Yes	No	
	Whole Sketch	Yes	Yes	Yes	Yes	Yes	
	Part Of Sketch	No	No	No	No	No	
	Multi Profiles	Yes	Yes	No	Yes	No	
Custom Direction	Ref. Vector	No	Yes	No	Possible	Possible	
	Ref. Plane	No	Yes	No	Possible	Possible	
	Sketch Chain	No	No	No	Possible	Possible	
	Curve	No	Yes	No	Possible	Possible	
	Sketch	No	Yes	No	Possible	Possible	
	Region	No	No	No	Possible	Possible	
	3D Sketch	No	Yes	No	Possible	Possible	
	Edge	No	Yes	No	Possible	Possible	
	Loop	No	No	No	Possible	Possible	
	Face	No	Yes	No	Possible	Possible	
Blind	Length(Flip)	Yes	Yes	Yes	Yes	Yes	
	Draft(Angle/Flip)	Yes	Yes	No	Yes	Yes	
Through All	No Option	Yes	Possible	No	Yes	Object	
	Draft(Angle/Flip)	Yes	Yes	No	Yes	Yes	
Up To Vertex	Vertex	Ref. Point	No	Possible	No	Possible	Possible
		Ref. Coordinate	No	Possible	No	Possible	Possible
		Curve Point	No	Possible	No	Possible	Possible
		Body Vertex	Yes	Possible	Possible	Possible	Possible
	Offset(Dist/Flip)	No	Yes	No	Possible	Possible	
	Draft(Angle/Flip)	Yes	Yes	No	Possible	Yes	
Mid Plane	Length(Flip)	Yes	Possible	Yes	Yes	Yes	

	Draft(Angle/Flip)	Possible	Yes	No	Yes	Yes
Opposite		Yes	Possible	Yes	Possible	Yes

Surface Revolve

APPLICATION		SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Profiles	Sketch	Yes	Yes	Yes	Yes	Yes
	3D Sketch	No	No	No	No	No
	Whole Sketch	Yes	Yes	Yes	Yes	Yes
	Part Of Sketch	No	No	No	No	No
	Multi Profiles	Yes	No	No	Yes	Yes
Axis	Ref. Vector	Yes	Yes	Yes	Possible	Yes
	Curve	Yes	Yes	Yes	Possible	Yes
	Region	No	No	No	Possible	No
	Edge	Yes	Yes	Yes	Possible	No
	Face	No	Yes	No	Possible	No
One Direction	Angle(Flip)	Yes	Yes	Yes	Yes	Yes
Mid Plane	Angle(Flip)	Yes	Yes	Yes	Yes	Yes
Both Directions	Angle(Flip)	Yes	Yes	Yes	Yes	Yes
	Opposite Angle	Yes	Yes	Yes	Yes	Yes

Surface Loft

Not supported in NX4 and Pro/E 3.0+.

APPLICATION		SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Profiles	Composite Profile	No	No	No	No	No
	Sketch	Yes	Yes	No	Yes	Yes
	3D Sketch	Yes	Yes	No	Yes	Yes
	Whole Sketch	Yes	Yes	No	Yes	Yes
	Part Of Sketch	No	No	No	No	No
	Edge	Not now	Not now	No	No	Yes
	Loop	No	No	No	No	No
	Face	Not now	Not now	No	No	No
Constraint	None	Yes	Yes	No	Yes	Yes
	Direction Vector	Not now	No	No	No	No
	Normal To Profile	Not now	No	No	No	No
	Tangency To Face	No	No	No	No	Possible
	Curvature to Face	No	No	No	No	Possible
Guide Curves	Composite Profile	No	No	No	No	No
	Sketch	Yes	Yes	No	Yes	Yes
	3D Sketch	Yes	Yes	No	Yes	Yes
	Whole Sketch	Yes	Yes	No	Yes	Yes
	Part Of Sketch	No	No	No	No	No
	Edge	Not now	No	No	No	No
Merge Tangent Faces		No	No	No	No	Yes

Surface Sweep

Not supported in NX4 and Pro/E 3.0+.

APPLICATION		SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Profiles	Composite Profile	No	No	No	No	No
	Sketch	Yes	Yes	No	Yes	Yes
	3D Sketch	Yes	Yes	No	Yes	No
	Whole Sketch	Yes	Yes	No	Yes	Yes
	Part Of Sketch	No	No	No	No	No
	Edge	Not now	Not now	No	No	No
	Loop	No	No	No	No	No
Path	Composite Profile	No	No	No	No	No
	Sketch	Yes	Yes	No	Yes	Yes
	3D Sketch	Yes	Yes	No	Yes	Yes

	Whole Sketch	Yes	Yes	No	Yes	Yes
	Part Of Sketch	No	No	No	No	No
	Edge	Not now	Not now	No	No	No
Method	Follow Path	Yes	Yes	No	Yes	Possible
	Keep Normal Constant	Not now	No	No	No	Yes
	Follow Path And First Guide Curve	Not now	No	No	No	Possible
	Follow First And Second Guide Curves	Not now	No	No	No	No
	Twist Along Path	Not now	No	No	No	No
	Twist Along Path With Normal Constant	Not now	No	No	No	No
Path Alignment Type	None	Yes	Yes	No	Yes	Yes
	Direction Vector	Not now	No	No	No	No
Guide Curves	Composite Profile	Not now	No	No	No	No
	Sketch Chain	Not now	No	No	No	Possible
	Curve	Not now	No	No	No	Yes
	Sketch	Not now	No	No	No	Yes
	Sketch Loop	Not now	No	No	No	Possible
	3D Sketch	Not now	No	No	No	Yes
	Edge	Not now	No	No	No	No
Improve Curvature Flow Along Sweep Path		No	No	No	No	No
Merge Tangent Faces		Not now	No	No	No	No

Reverse Normal

APPLICATION	SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Surface Bodies	No	Yes	No	No	Object

Sew

APPLICATION	SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Surface Bodies	Yes	Yes	No	No	Yes
Continuity Constraint	No	No	No	No	No

Trim

APPLICATION		SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Tool	Sketch Chain	No	No	No	No	Yes
	Ref. Plane	No	Yes	Yes		Yes
	Region	No	Object	Object		Object
	Surface Body	No	Yes	Yes		Yes
	Solid Body	No	No	No		Yes
Target	Surface Body	No	Yes	Yes	No	Yes

Untrim

APPLICATION		SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Tool	Edge	Yes	No	No	No	No
	Loop	Yes	No	No		No
	Face	No	No	No		No

Surface Offset

APPLICATION		SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Faces	Face	No	Yes	Object	No	Object
	Surface Body	No	Yes	Object	No	Object
Offset Distance(Flip)		No	Yes	Object	No	Object
More Option	Make Side Wall	No	No	Object	No	No
	Remove Original Faces	No	No	Object	No	No

Fillet

APPLICATION			SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +	
Constant Fillet	Entities	Edge	Yes	Yes	Yes	Yes	Yes	
		Loop	Not now	Not now	Not now	Not now	Not now	
		Face	Yes	Yes	No	No	Yes	
	Radius	Yes	Yes	Yes	Yes	Yes		
	Tangent Propagation	Yes	Yes	Yes	No	Yes		
	Multiple Set	Yes	Yes	Yes	Yes	Possible		
Variable Fillet	Entities	Edge	Yes	Yes	Yes	No	Yes	
	Radius	Possible	Yes	Yes	Yes		Yes	
	Tangent Propagation	Yes	Yes	No	No		No	
	Transition Method	Smooth Transition	Yes	No	No		No	Yes
		Linear Transition	Yes	Yes	No		No	Yes
Conic Fillet	No	No	No	No	No	No		
Face Fillet	Faces/Faces	Face	Yes	Yes	Yes	No	Yes	
	Radius	Yes	Yes	Yes	Yes		Yes	
	Hold Line	Edge	No	Yes	No		No	No
		Curve	No	Yes	No		No	No
		Sketch Chain	No	No	No		No	No
		Asymmetric	No	Possible	No		No	No
	Help Point	Ref. Point	No	No	No		No	No
		Curve Point	No	No	No		No	No
		Vertex	Yes	No	No		No	No
	Tangent Propagation	Yes	Yes	Yes	Yes		No	
	Constant Curvature	Yes	No	No	No		No	
	Constant Width	Yes	No	No	No		No	
	Conic	Radius2	No	Yes	No		No	No
Rho		No	Yes	No	No	No		
Full Face Fillet	Fillet Entity Set	Left	Yes	No	No	No	Yes	
		Right	Yes	No	No		Yes	
		Center	Yes	No	No		Yes	

Chamfer

APPLICATION		SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Entities	Edge	Yes	Yes	Yes	Yes	Yes
	Loop	Not now	Not now	Not now	Not now	Not now
	Face	Yes	Yes	No	No	No
	Multiple Entities	No	No	No	No	No
Angle Distance &	Distance	Yes	Yes	Yes	No	Yes
	Angle	Yes	Yes	Yes		Yes
	Flip Direction	Yes	Yes	No		No
	Tangent Propagation	Yes	Yes	Yes		Yes
Distance & Distance	Distance	Yes	Yes	Yes	Yes	Yes
	Distance2	Yes	Yes	Yes	Yes	Yes
	Flip Direction	Yes	Yes	No	Yes	Yes
	Tangent Propagation	Yes	Yes	Yes	No	Yes

Draft

APPLICATION			SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Neutral Plane	Neutral Plane	Ref. Plane	No	Yes	Yes	No	Yes
		Region	No	No	No	No	No
		Face	Yes	Yes	Yes	No	Yes
	Draft Faces	Yes	Yes	Yes	No	Yes	
	Angle(Flip)	Yes	Yes	Yes	No	Possible	

Parting Line Draft	Pull Direction	Ref. Vector	Not Now	Yes	Yes	No	Yes
		Ref. Plane	Not Now	Yes	No	No	Yes
		Sketch Chain	Not Now	No	No	No	No
		Curve	Not Now	Yes	No	No	No
		Sketch	Not Now	Yes	No	No	No
		Region	Not Now	No	No	No	No
		3D Sketch	Not Now	Yes	No	No	No
		Edge	Not Now	Yes	Yes	No	Yes
		Loop	Not Now	No	No	No	No
		Face	Not Now	Yes	No	No	
Parting Lines	Edge	Not Now	Yes	Yes	No	Yes	
Angle		Not Now	Yes	Yes	No	Yes	
Flip Direction		Not Now	Yes	Yes	No	Yes	
Allow Reduced Angle		Not Now	No	No	No	Yes	
Step Draft	Neutral Plane	Ref. Plane	Not Now	No	No	No	No
		Region	Not Now	No	No	No	No
		Face	Not Now	No	No	No	No
	Parting Lines	Edge	Not Now	No	No	No	No
	Angle		Not Now	No	No	No	No
	Direction		Not Now	No	No	No	No
	Step Type	Tapered Steps	Not Now	No	No	No	No
Perpendicular Steps		Not Now	No	No	No	No	

Mirror

APPLICATION		SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Bodies	Surface Body	Yes	Yes	Object	Object	No
	Solid Body	Yes	Yes	Object	Object	Yes
Symmetry Plane	Ref. Plane	Yes	Yes	Object	Object	Yes
	Face	Yes	Possible	Object	Object	Yes
Trim & Merge		No	No	Object	Object	No

Delete Face

APPLICATION	SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Delete	Yes	No	No	No	Yes
Delete And Heal	No	No	No		Yes
Delete And Fill	No	No	No		No

Move Face

APPLICATION		SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +	
Translate	Faces	Yes	Yes	No	No	Yes	
	Direction	Ref. Vector	Yes	Yes	No	No	Yes
		Ref. Plane	Yes	Yes	No	No	No
		Curve	No	Possible	No	No	No
		Edge	Yes	Yes	No	No	Yes
		Face	Yes	Yes	No	No	Yes
Distance(Flip)		Yes	Yes	No	No	Yes	
Rotate	Faces	Yes	Yes	No	No	Yes	
	Axis	Ref. Vector	Yes	Yes	No	No	Yes
		Curve	No	Possible	No	No	No
		Edge	Yes	Yes	No	No	Yes
	Angle(Flip)		Yes	Yes	No	No	Yes

Replace Face

APPLICATION		SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Target Faces	Face	Yes	Yes	Yes	No	Object
Tool Body	Region	Object	Object	No	No	Object
	Surface Body	Yes	Yes	Yes	No	Object

Linear Pattern

APPLICATION		SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Bodies	Solid Body	Yes	Object	Object	Object	Yes
	Surface Body	Yes	Object	Object	Object	No
Direction	Ref. Vector	Yes	Object	Object	Object	Yes
	Ref. Plane	No	Object	Object	Object	Yes
	Sketch Chain	No	Object	Object	Object	No
	Curve	No	Object	Object	Object	No
	Sketch	No	Object	Object	Object	No
	3D Sketch	No	Object	Object	Object	No
	Edge	Yes	Object	Object	Object	Yes
	Loop	No	Object	Object	Object	No
	Region	No	Object	Object	Object	No
Face	No	Object	Object	Object	Yes	
Skip Instances		Yes	Object	Object	Object	No

Circular Pattern

APPLICATION		SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Bodies	Solid Body	Yes	Object	Object	Object	Yes
	Surface Body	Yes	Object	Object	Object	No
Rotation Axis	Ref. Vector	Yes	Object	Object	Object	Yes
	Curve	No	Object	Object	Object	No
	Edge	Yes	Object	Object	Object	Possible
	Region	No	Object	Object	Object	No
	Face	No	Object	Object	Object	No
Equal Spacing		Yes	Object	Object	Object	Yes
Rotate About Axis		True: Yes False: No	Object	Object	Object	True: Yes False: No
Skip Instances		Yes	Object	Object	Object	No

Curve Pattern

APPLICATION		SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Bodies	Solid Body	Yes	Object	Object	Object	Object
	Surface Body	Yes	Object	Object	Object	Object
Path Curve	Curve	1 Curve: Yes	Object	Object	Object	Object
	Edge	Yes	Object	Object	Object	Object
Equal Spacing		Yes	Object	Object	Object	Object
Align Method	Align To Seed	Yes	Object	Object	Object	Object
	Rotate Along Curve	Yes	Object	Object	Object	Object
Skip Instances		Yes	Object	Object	Object	Object

Transform Body

APPLICATION		SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +	
Bodies	Solid Body	Yes	No	No	No	Yes	
	Surface Body	Yes	No	No	No	Yes	
Duplicate		Yes	No	No	No	No	
Method	Rotate And Translate		Yes	No	No	No	Yes
	Scale: Scale Center	Global Origin	Yes	No	No	No	No
		Body Center	No	No	No	No	No
		Custom	No	No	No	No	No
	By Matrix		No	No	No	No	Possible
	Datum Alignment		No	No	No	No	Object
Align To Scan		No	No	No	No	Object	

Split Face

APPLICATION			SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Projection	Tool	Sketch	Yes	Yes	No	No	Yes

	Entities	3D Sketch	No	Yes	No	No	No
	Target Entities	Face	Yes	Yes	No	No	Yes
	Options	Single Direction(Flip)	Yes	Yes	No	No	No
		Bi-Direction	Yes	No	No	No	Yes
Silhouette	Pull Direction	Ref. Vector	No	No	No	No	No
		Region	No	No	No	No	No
		Ref. Plane	No	No	No	No	No
		Sketch Chain	No	No	No	No	No
		Curve	No	No	No	No	No
		Sketch	No	No	No	No	No
		3D Sketch	No	No	No	No	No
		Edge	No	No	No	No	No
		Loop	No	No	No	No	No
	Face	No	No	No	No	No	
	Target Entities	Face	No	No	No	No	No
Options	Draft Angle	No	No	No	No	No	
Intersection	Tool Entities	Face	No	Yes	No	No	No
		Surface Body	No	Yes	No	No	Yes
		Solid Body	No	Yes	No	No	Yes
	Target Entities	Face	No	Yes	No	No	Yes
		Surface Body	No	Yes	No	No	Yes
		Solid Body	No	Yes	No	No	Yes

Helix/Spiral

APPLICATION	SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Curves	Object	Object	No	No	Object

Emboss

APPLICATION		SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +	
Pad/Pocket		Yes	Yes	No	No	Yes	
Profile	Sketch	Yes	Yes	No	No	Yes	
	Sketch Chain	No	No	No	No	Yes	
	Sketch Loop	No	No	No	No	Yes	
	3D Sketch	No	Yes	No	No	No	
	Loop	No	No	No	No	No	
Target	Face	Yes	Yes	No	No	Yes	
	Solid Body	No	Yes	No	No	Yes	
	Surface Body	No	Yes	No	No	No	
End Cap	Offset Surface		Yes	Yes	No	No	Yes
	Translate Surface	Ref. Vector	No	Yes	No	No	No
		Region	No	No	No	No	No
		Ref. Plane	No	Yes	No	No	No
		Sketch Chain	No	No	No	No	No
		Curve	No	Yes	No	No	No
		Sketch	No	Yes	No	No	No
		3D Sketch	No	Yes	No	No	No
		Edge	No	Yes	No	No	No
		Loop	No	No	No	No	No
	Face	No	Yes	No	No	No	
	End Cap Surface	Region	No	Object	No	No	No
		Face	No	Yes	No	No	No
		Surface Body	No	Yes	No	No	No
Solid Body		No	Yes	No	No	No	
Side Wall	Face Normal		Yes	Yes	No	No	No
	Profile Direction		No	Yes	No	No	Yes

	Direction Vector	Ref. Vector	Yes	Yes	No	No	No
		Region	No	No	No	No	No
		Ref. Plane	No	Yes	No	No	No
		Sketch Chain	No	No	No	No	No
		Curve	No	Yes	No	No	No
		Sketch	No	Yes	No	No	No
		3D Sketch	No	Yes	No	No	No
		Edge	Yes	Yes	No	No	No
		Loop	No	No	No	No	No
Draft	Angle (Flip)	No	Yes	No	No	No	

Ref. Point

APPLICATION		SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +	
Definition	Position	No	Object	Object	No	Object	
Extract		No	Object	Object	No	Object	
Find Circle Center		No	Object	Object	No	Object	
Find Slot Center		No	Object	Object	No	Object	
Find Rectangle Center		No	Object	Object	No	Object	
Find Polygon Center		No	Object	Object	No	Object	
Find Sphere Center		No	Object	Object	No	Object	
Projection		No	Object	Object	No	Object	
Pick Multiple Points	Entities	Ref. Point	No	Object	Object	No	Object
		Curve Point	No	Object	Object	No	Object
		Poly-Vertex	No	Object	Object	No	Object
		Curve	No	Object	Object	No	Object
		Region	No	Object	Object	No	Object
		Region Group	No	Object	Object	No	Object
		Vertex	No	1 Vertex: Yes Else : Object	1 Vertex: Yes Else : Object	No	1 Vertex: Yes Else: Object
		Edge	No	Object	Object	No	Object
		Face	No	Object	Object	No	Object
N Division		No	Object	Object	No	Object	
Between 2 Points		No	Object	Object	No	Object	
Intersect 2 Lines		No	Object	Yes	No	Object	
Intersect Line and Face		No	Object	Yes	No	Object	
Intersect 3 Planes		No	Object	Yes	No	Object	

Ref. Vector

APPLICATION		SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +	
Definition	Position & Direction	Position (XYZ) Direction (+-XYZ, XYZ (value))	Object	Object	Object	No	Object
	Start & End Position	Start Position (XYZ) End Position (XYZ)	Object	Object	Object	No	Object
Extract	Entities	Poly-Vertex	Object	Object	Object	No	Object
		Ref. Polyline	Object	Object	Object	No	Object
		Curve	Object	Yes	Object	No	Yes
		Poly-Face	Object	Object	Object	No	Object
		Region	Object	Object	Object	No	Object
		Region Group	Object	Object	Object	No	Object
		Edge	1 Linear Edge : Yes Else : Object	1 Linear Edge : Yes Else : Object	1 Linear Edge : Yes Else : Object	No	1 Linear Edge : Yes Else : Object
		Face	Object	Yes	Object	No	Object
Mesh		Object	Object	Object	No	Object	

	Fitting Options		Object	Object	Object	No	Object
	Constraint Options	Project Plane On	Object	Object	Object	No	Object
Find Slot Axis			Object	Object	Object	No	Object
Find Cylinder Axis			Object	Object	Object	No	Object
Find Cone Axis			Object	Object	Object	No	Object
Projection			Object	Object	Object	No	Object
Pick Multiple Points	Entities	Ref. Point	Object	2 Point : Yes Else : Object	2 Point : Yes Else : Object	No	2 Vertex : Yes Else : Object
		Curve Point	Object	Object	Object	No	Object
		Poly-Vertex	Object	Object	Object	No	Object
		Curve	Object	Object	Object	No	Object
		Region	Object	Object	Object	No	Object
		Region Group	Object	Object	Object	No	Object
		Vertex	2 Vertex : Yes Else : Object	2 Vertex : Yes Else : Object	2 Vertex : Yes Else : Object	No	2 Vertex : Yes Else : Object
		Edge	Object	Object	Object	No	Object
		Face	Object	Object	Object	No	Object
Pick Point & Line	Entities	Ref. Point	Object	Yes	Object	No	Object
		Ref. Vector	Object	Yes	Object	No	Object
		Poly-Vertex	Object	Object	Object	No	Object
		Curve Point	Object	Object	Object	No	Object
		Ref. Plane	Object	Yes	Object	No	Object
		Curve	Object	Yes	Object	No	Object
		Sketch	Object	Possible	Object	No	Object
		Region	Object	Object	Object	No	Object
		Region Group	Object	Object	Object	No	Object
		Vertex	Object	Yes	Object	No	Object
		Edge	Object	Yes	Object	No	Object
		Face	Object	Yes	Object	No	Object
Intersect 2 Planes	Entities	Ref. Plane	Yes	Yes	Yes	No	Yes
		Sketch	Object	Object	Object	No	Object
		Region	Object	Object	Object	No	Object
		Region Group	Object	Object	Object	No	Object
		Face	Yes	Yes	Yes	No	Yes
Average			Object	Object	Object	No	Object
Cross 2 Lines			Object	Object	Object	No	Object
Revolving Axis			Object	Object	Object	No	Object
Extrusion Axis			Object	Object	Object	No	Object
Pattern Revolving Axis			Object	Object	Object	No	Object
Pattern Translation Axis			Object	Object	Object	No	Object

Ref. Plane

APPLICATION			SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Definition	Position & Normal	Position (XYZ) Normal (+- XYZ, XYZ (value))	Object	Object	Object	No	Object
	3 Positions	Position1 (XYZ) Position2 (XYZ) Position3 (XYZ)	Object	Object	Object	No	Object
	Coefficients	A (value) B (value) C (value) D (value)	Object	Object	Object	No	Object
Extract	Entities	Poly-Vertex	Object	Object	Object	No	Object
		Ref. Polyline	Object	Object	Object	No	Object
		Ref. Plane	Object	Object	Yes	No	Object
		Curve	Object	Object	Object	No	Object
		Poly-Face	Object	Object	Object	No	Object

		Region	Object	Object	Object	No	Object
		Region Group	Object	Object	Object	No	Object
		Edge	Object	Yes	Object	No	Object
		Face	1 Planar Face : Yes Else : Object	1 Planar Face : Yes Else : Object	1 Planar Face : Yes Else : Object	No	1 Planar Face : Yes Else : Object
		Mesh	Object	Object	Object	No	Object
	Fitting Options		Object	Object	Object	No	Object
	Constraint Options	Axis Constraint	Object	Object	Object	No	Object
Projection	Entities		Object	Object	Object	No	Object
Pick Multiple Points	Entities	Ref. Point	Object	3 Point : Yes Else : Object	3 Point : Yes Else : Object	No	3 Point : Yes Else : Object
		Poly-Vertex	Object	Object	Object	No	Object
		Curve Point	Object	Object	Object	No	Object
		Curve	Object	Object	Object	No	Object
		Region	Object	Object	Object	No	Object
		Region Group	Object	Object	Object	No	Object
		Vertex	Object	3 Vertex : Yes Else : Object	3 Vertex : Yes Else : Object	No	3 Vertex : Yes Else : Object
		Edge	Object	Object	Object	No	Object
		Face	Object	Object	Object	No	Object
Pick Point & Normal Axis	Entities	Ref. Point	Object	Yes	Yes	No	Object
		Curve Point	Object	Object	Object	No	Object
		Poly-Vertex	Object	Object	Object	No	Object
		Ref. Vector	Object	Yes	Yes	No	Object
		Curve	Object	Yes	Object	No	Object
		Ref. Plane	Object	Yes	Object	No	Object
		Sketch	Object	Possible	Object	No	Object
		Region	Object	Object	Object	No	Object
		Region Group	Object	Object	Object	No	Object
		Vertex	Object	Yes	Yes	No	Object
		Edge	Object	Yes	Yes	No	Object
Face	Object	Yes	Object	No	Object		
Pick Point & Coplanar Axis	Entities	Ref. Point	Object	Yes	Yes	No	Object
		Curve Point	Object	Object	Object	No	Object
		Poly-Vertex	Object	Object	Object	No	Object
		Ref. Vector	Object	Yes	Yes	No	Object
		Curve	Object	Yes	Object	No	Object
		Region	Object	Object	Object	No	Object
		Region Group	Object	Object	Object	No	Object
		Vertex	Object	Yes	Yes	No	Object
		Edge	Object	Yes	Yes	No	Object
		Face	Object	Object	Object	No	Object
N Division	Entities		Object	Object	Object	No	Object
	Pattern Option		Object	Object	Object	No	Object
Offset	Entities	Ref. Plane	Yes	Yes	Yes	No	Yes
		Sketch	Object	Object	Object	No	Object
		Region	Object	Object	Object	No	Object
		Region Group	Object	Object	Object	No	Object
		Face	Yes	Yes	Yes	No	Yes
	Offset Options	Number (Value)	Yes	Yes	Yes	No	Yes
		Distance (Value/Flip)	Yes	Yes	Yes	No	Yes
Rotation	Entities		Object	Object	Object	No	Object
	Rotate Options	Number (Value)	Object	Object	Object	No	Object
		Angle (Flip)	Object	Object	Object	No	Object
Average	Entities		Object	Object	Object	No	Object
View Direction	Entities		Object	Object	Object	No	Object
Tangent	Entities		Object	Object	Object	No	Object

Orthogonal	Entities	Object	Object	Object	No	Object
Draw Cutting	Entities	Object	Object	Object	No	Object
Mirror	Entities	Object	Object	Object	No	Object
Extreme Position	Entities	Object	Object	Object	No	Object
	Direction	Object	Object	Object	No	Object

Ref. Coordinate

APPLICATION			SolidWorks 2006 +	SIEMENS NX4 +	Pro/E Wildfire 3 +	AutoCAD 2007 +	Inventor 2010 +
Definition	Origin & Axes	Origin (XYZ) Axis 1, 2 (+- XYZ, XYZ (value)) Order	No	Object	Object	No	Object
	Origin & Rotation	Origin (XYZ) Rotation	No	Object	Object	No	Object
Extract	Entities	Poly-Vertex	No	Object	Object	No	No
		Ref. Polyline	No	Object	Object	No	No
		Curve	No	Object	Object	No	No
		Poly-Face	No	Object	Object	No	No
		Region	No	Object	Object	No	No
		Region Group	No	Object	Object	No	No
		Edge	No	Object	Object	No	No
		Face	No	Object	Object	No	No
	Mesh	No	Object	Object	No	No	
Fitting Options			No	Object	Object	No	No
Pick Multiple Points	Entities	Ref. Point	No	3 Point : Yes Else : Object	Object	No	No
		Poly-Vertex	No	Object	Object	No	No
		Curve Point	No	Object	Object	No	No
		Curve	No	Object	Object	No	No
		Region	No	Object	Object	No	No
		Region Group	No	Object	Object	No	No
		Vertex	No	3 Vertex : Yes Else : Object	Object	No	Object
		Edge	No	Object	Object	No	No
	Face	No	Object	Object	No	No	
More Option	Order	No	Possible Only XY Order	Object	No	Possible	
Pick Point & Line	Entities	Ref. Point	No	Yes	Object	No	No
		Ref. Vector	No	Yes	Object	No	No
		Poly-Vertex	No	Object	Object	No	No
		Curve Point	No	Object	Object	No	No
		Ref. Plane	No	Yes	Object	No	No
		Curve	No	Yes	Object	No	No
		Region	No	Object	Object	No	No
		Region Group	No	Object	Object	No	No
		Sketch	No	Yes	Object	No	No
		Vertex	No	Yes	Object	No	No
		Edge	No	Yes	Object	No	No
		Face	No	Yes	Object	No	No
	More Option	Order	No	Possible Only XY Order	Object	No	No
Pick Point & Coordinate	Entities	Ref. Point	No	Yes	Object	No	No
		Ref. Coordinate	No	Yes	Object	No	No
		Poly-Vertex	No	Object	Object	No	No
		Curve Point	No	Object	Object	No	No
		Curve	No	Object	Object	No	No
		Region	No	Object	Object	No	No
		Region Group	No	Object	Object	No	No

		Vertex	No	Yes	Object	No	No
		Edge	No	Object	Object	No	No
		Face	No	Object	Object	No	No
Intersect Plane	3 Entities	Ref. Plane	No	Yes	Yes	No	No
		Sketch	No	Object	Object	No	No
		Region	No	Object	Object	No	No
		Region Group	No	Object	Object	No	No
		Face	No	Yes	Yes	No	No
	Intersection Options	Check Orthogonal	No	Possible Only Orthogonal	Object	No	No
	More Options	Flip Z Flip X	No	Object	Object	No	No
Tangent	Entities	Ref. Point	No	Object	Object	No	No
		Poly-Vertex	No	Object	Object	No	No
		Curve Point	No	Object	Object	No	No
		Vertex	No	Object	Object	No	No

See Also

[File > LiveTransfer\(TM\) > To SolidWorks](#)

[File > LiveTransfer\(TM\) > To Siemens NX](#)

[File > LiveTransfer\(TM\) > To Creo\(Pro/E\)](#)

[File > LiveTransfer\(TM\) > To AutoCAD](#)

[File > LiveTransfer\(TM\) > To Inventor](#)

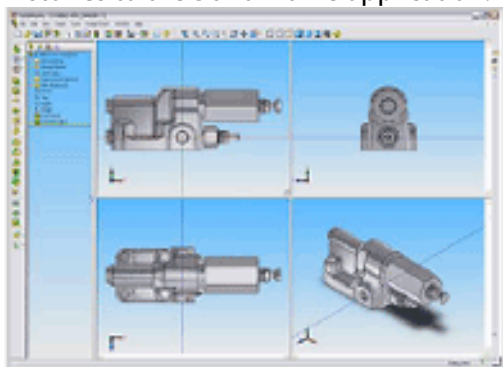
[File > LiveTransfer\(TM\) > Save As CATIA V4](#)

[File > LiveTransfer\(TM\) > Save As CATIA V5](#)

File > LiveTransfer(TM) > To SolidWorks

What

The **LiveTransfer™ To SolidWorks** command is used to directly export features and modeling histories to the **SolidWorks** application.



Why

The **LiveTransfer™ To SolidWorks** tool is useful for:

- Transferring data into the **SolidWorks** application

How to Use

How use the LiveTransfer™ To SolidWorks command

1. Go to **File > LiveTransfer(TM) > To SolidWorks**.

Note:

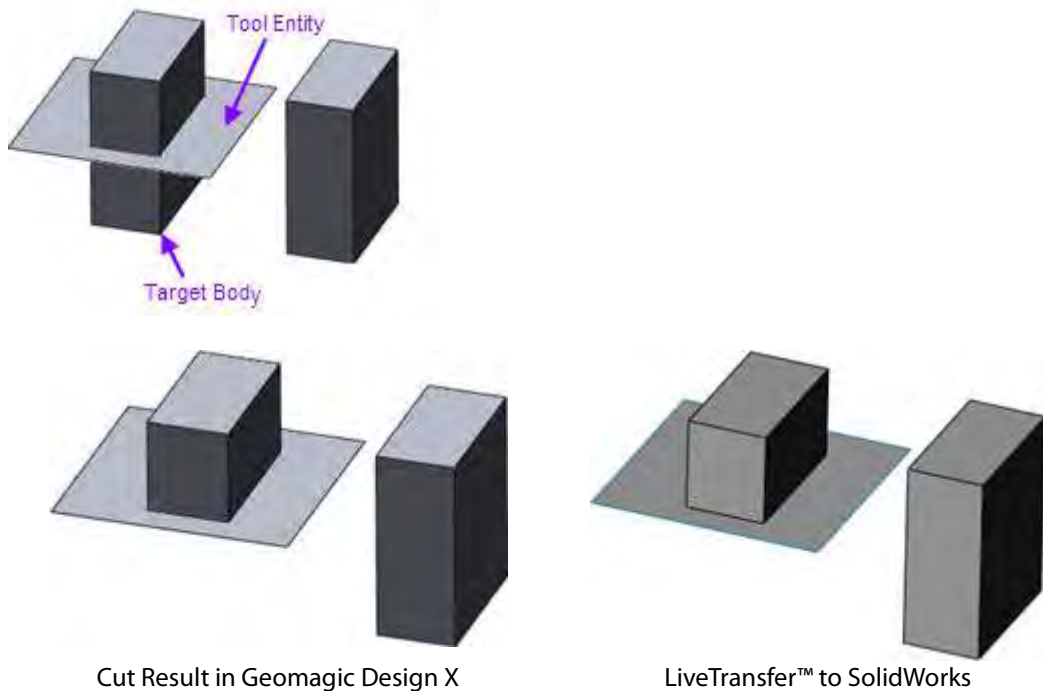
The application currently supports **SolidWorks** 2006, 2007, 2008, 2009, 2010, 2011, 2012, and 2013.

2. Go to **Start From First Feature** in the dialog tree.

Note:

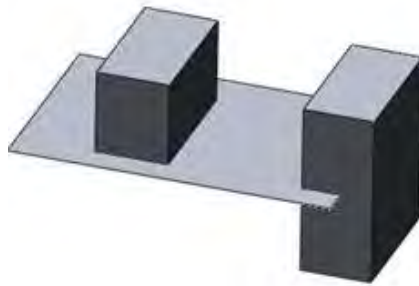
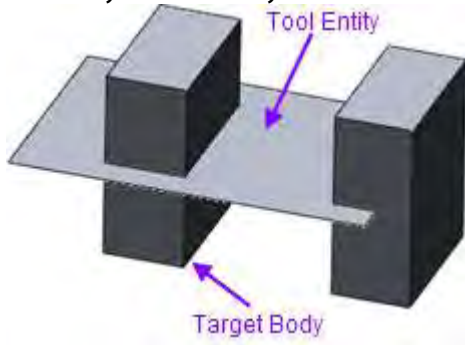
The creation method is different in **SolidWorks** and there are some limitations to the **Cut** command.

If a **Tool Entity** completely cuts a body, it will be transferred without any barriers.



However, if a **Tool Entity** completely cuts a body and crosses into another body, it may generate an error message within **SolidWorks**. To avoid this error message, uncheck the **Auto-select** option in the **Cut With Surface** command in **SolidWorks**, and then

manually cut the body.

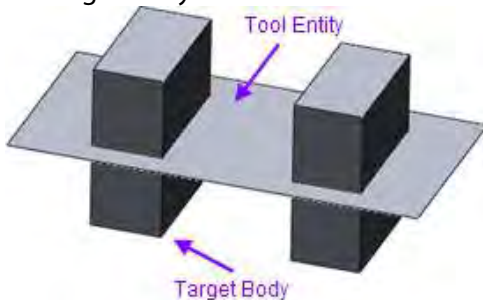


Cut Result in Geomagic Design X

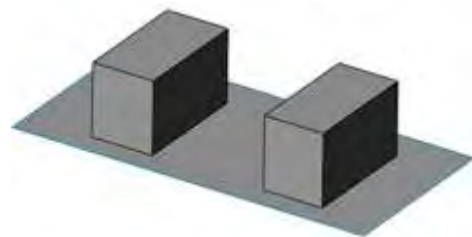


LiveTransfer™ to SolidWorks

If a **Tool Entity** completely cuts a target body and completely crosses another body, the other body will also be cut in **SolidWorks**. To avoid this situation, uncheck the **Auto-select** option in the **Cut With Surface** command in **SolidWorks**, and then manually cut the target body.

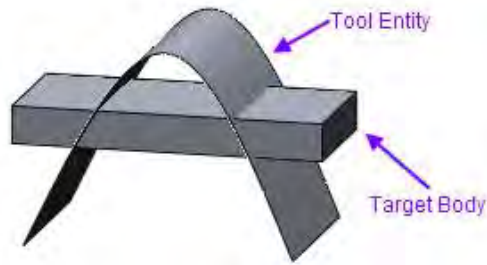


Cut Result in Geomagic Design X

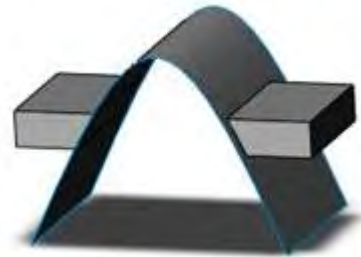


LiveTransfer to SolidWorks

If a **Tool Entity** cuts a single body and creates two or more remaining bodies, remaining bodies can be selected in the application. After using the **LiveTransfer™ To SolidWorks** command however, the remaining bodies will be determined to be in the normal direction of the Tool Entity. In this case the body needs to be cut manually in **SolidWorks**.



Cut Result in Geomagic Design X



LiveTransfer to SolidWorks

3. Click **OK** to finish the command.

Options in Detail

Methods

This tool features 3 different methods:

- **Start From First Feature**

All features starting from the first feature in the **Feature Tree** will be exported into **SolidWorks**

- **Resume From Selected Feature**

All features starting from a selected feature in the **Feature Tree** will be exported into **SolidWorks**

- **Only Selected Entities**

Only selected features will be exported into **SolidWorks**

Options

SolidWorks (Version) – Chooses a dedicated software version.

Note:

When different SolidWorks versions are installed in the same PC, the **SolidWorks (Version)** option will be displayed.

Prefix For Feature Name – Defines the prefix for a feature name.

Try To Transfer Sweep/Loft As Parametric Feature – Transfers **Sweep** and **Loft** features as parametric features into **SolidWorks**, however if the sweeping and lofting algorithms are different between the applications then the features will be transferred as dummy objects.

Feature – Selects a feature to start transferring.

Entities – Selects entities only to transfer.

Specify Face Or Plane Of Sketch Inside Of SolidWorks – Allows a **Face** or **Plane** to be selected to be used as a 2D sketch plane.

SolidWorks Option

Use Custom SolidWorks Execution File Path – Manually defines the path of a **SolidWorks** application execution file.

Pause At ...

Pause At – Pauses transfer before a selected command is performed.

See Also

[File > LiveTransfer\(TM\) > To Siemens NX](#)

[File > LiveTransfer\(TM\) > To Creo\(Pro/E\)](#)

[File > LiveTransfer\(TM\) > To AutoCAD](#)

[File > LiveTransfer\(TM\) > To Inventor](#)

[File > LiveTransfer\(TM\) > Save As CATIA V4](#)

[File > LiveTransfer\(TM\) > Save As CATIA V5](#)

File > LiveTransfer(TM) > To Siemens NX

What

The **LiveTransfer™ To Siemens NX** command is used to directly export features and modeling histories to the **Siemens NX** application.



Why

The **LiveTransfer™ To Siemens NX** tool is useful for:

- Transferring data into the **Siemens NX** application

How to Use

How use the LiveTransfer™ To Siemens NX command

1. Go to **File > LiveTransfer(TM) > To Siemens NX**.

Note:

The application currently supports **Siemens NX4, 5, 6, 7, 7.5, and 8**.

2. Choose **Start From First Feature** in the dialog tree.
3. Click **OK** to finish the command.

Options in Detail

Methods

This tool features 3 different methods such as:

- **Start From First Feature**

All features starting from the first feature in the **Feature Tree** will be exported into **Siemens NX**

- **Resume From Selected Feature**

All features starting from a selected feature in the **Feature Tree** will be exported into **Siemens NX**

- **Only Selected Entities**

Only selected features will be exported into **Siemens NX**

Options

Siemens NX (Version) – Chooses a dedicated software version.

Note:

When different Siemens NX versions are installed in the same PC, the **Siemens NX (Version)** option will be displayed.

Prefix For Feature Name – Defines the prefix for a feature name.

Try To Transfer Sweep/Loft As Parametric Feature – Transfers **Sweep** and **Loft** features as parametric features into **Siemens NX**, however if the sweeping and lofting algorithms are different between the applications then the features will be transferred as dummy objects.

Feature – Selects a feature to start transferring.

Entities – Selects entities only to transfer.

Use Selected Face Or Plane In NX For 2D Sketch Plane – Allows a **Face** or **Plane** to be selected to be used as a 2D sketch plane.

Siemens NX Option

Use Custom NX Execution File Path – Manually defines the path of a **Siemens NX** application execution file.

Pause At ...

Pause At – Pauses transfer before a selected command is performed.

See Also

[File > LiveTransfer\(TM\) > To SolidWorks](#)

[File > LiveTransfer\(TM\) > To Creo\(Pro/E\)](#)

[File > LiveTransfer\(TM\) > To AutoCAD](#)

[File > LiveTransfer\(TM\) > To Inventor](#)

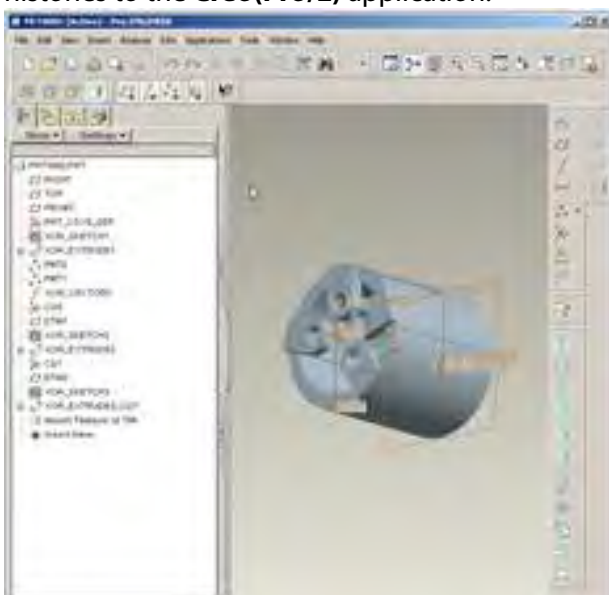
[File > LiveTransfer\(TM\) > Save As CATIA V4](#)

[File > LiveTransfer\(TM\) > Save As CATIA V5](#)

File > LiveTransfer(TM) > To Creo(Pro/E)

What

The **LiveTransfer™ To Creo(Pro/E)** command is used to directly export features and modeling histories to the **Creo(Pro/E)** application.



Why

The **LiveTransfer™ To Creo(Pro/E)** tool is useful for:

- Transferring data into the **Creo(Pro/E)** application

How to Use

How use the **LiveTransfer™ To Creo(Pro/E)** command

1. Go to **File > LiveTransfer(TM) > To Creo(Pro/E)**.

Note:

The application currently supports Pro/E Wildfire 3.0, 4.0, 5.0, Creo Elements/Pro 5.0, and Creo Parametric 1.0, 2.0.

2. Choose **Start From First Feature** in the dialog tree.
3. Click **OK** to finish the command.

Options in Detail

Methods

This tool features 3 different methods:

- **Start From First Feature**

All features starting from the first feature in the **Feature Tree** will be exported into **Creo(Pro/E)**

- **Resume From Selected Feature**

All features starting from a selected feature in the **Feature Tree** will be exported into **Creo(Pro/E)**

- **Only Selected Entities**

Only selected features will be exported into **Creo(Pro/E)**

Options

Prefix For Feature Name – Defines the prefix for a feature name.

Try To Transfer Sweep/Loft As Parametric Feature – Transfers **Sweep** and **Loft** features as parametric features into **Creo(Pro/E)**, however if the sweeping and lofting algorithms are different between the applications then the features will be transferred as dummy objects.

Feature – Selects a feature to start transferring.

Entities – Selects entities only to transfer.

Use Selected Face Or Plane In Creo(Pro/E) For 2D Sketch Plane – Allows a **Face** or **Plane** to be selected to be used as a 2D sketch plane.

Cre-(Pro/E) Options

Use Custom Creo(Pro/E) Execution File Path – Manually defines the path of a Creo(Pro/E) application execution file.

Use Creo(Pro/E) Template File – Uses a template **Creo(Pro/E)** file in the .prt format.

Pause At ...

Pause At – Pauses transfer before a selected command is performed.

See Also

[File > LiveTransfer\(TM\) > To SolidWorks](#)

[File > LiveTransfer\(TM\) > To Siemens NX](#)

[File > LiveTransfer\(TM\) > To AutoCAD](#)

[File > LiveTransfer\(TM\) > To Inventor](#)

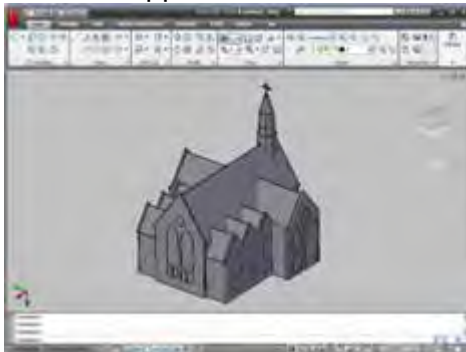
[File > LiveTransfer\(TM\) > Save As CATIA V4](#)

[File > LiveTransfer\(TM\) > Save As CATIA V5](#)

File > LiveTransfer(TM) > To AutoCAD

What

The **LiveTransfer™ To AutoCAD** command is used to directly export features and sketches to the **AutoCAD** application.



Why

The **LiveTransfer™ To AutoCAD** tool is useful for:

- Transferring data into the **AutoCAD** application

How to Use

How use the LiveTransfer™ To AutoCAD command

1. Go to **File > LiveTransfer(TM) > To AutoCAD**.

Note:

The application currently supports **AutoCAD 2007, 2008, 2009, 2010, and 2011**. **AutoCAD LT** is not supported

2. Choose **Start From First Feature** in the dialog tree.
3. Click **OK** to finish the command.

Options in Detail

Methods

This tool features 3 different methods:

- **Start From First Feature**

All features starting from the first feature in the **Feature Tree** will be exported into **AutoCAD**

- **Resume From Selected Feature**

All features starting from a selected feature in the **Feature Tree** will be exported into **AutoCAD**

- **Only Selected Entities**

Only selected features will be exported into **AutoCAD**

Options

AutoCAD (Version) – Chooses a dedicated software version.

Note:

When different AutoCAD versions are installed in the same PC, the **AutoCAD (Version)** option will be displayed.

Try To Transfer Sweep/Loft As Parametric Feature – Transfers **Sweep** and **Loft** features as parametric features into **AutoCAD**, however if the sweeping and lofting algorithms are different between the applications then the features will be transferred as dummy objects.

Feature – Selects a feature to start transferring.

Entities – Selects entities only to transfer.

AutoCAD Options

Use Custom AutoCAD Execution File Path – Manually defines the path of a **AutoCAD** application execution file.

Pause At ...

Pause At – Pauses transfer before a selected command is performed.

See Also

[File > LiveTransfer\(TM\) > To SolidWorks](#)

[File > LiveTransfer\(TM\) > To Siemens NX](#)

[File > LiveTransfer\(TM\) > To Creo\(Pro/E\)](#)

[File > LiveTransfer\(TM\) > To Inventor](#)

[File > LiveTransfer\(TM\) > Save As CATIA V4](#)

[File > LiveTransfer\(TM\) > Save As CATIA V5](#)

File > LiveTransfer(TM) > To Inventor

What

The **LiveTransfer™ To Inventor** command is used to directly export features and modeling histories to the **Inventor** application.



Why

The **LiveTransfer™ To Inventor** tool is useful for:

- Transferring data into the **Inventor** application

How to Use

How use the LiveTransfer™ To Inventor command

1. Go to **File > LiveTransfer(TM) > To Inventor**.

Note:

The application currently supports **Inventor 2010, 2011, 2012, and 2013**.

2. Choose **Start From First Feature** in the dialog tree.
3. Click **OK** to finish the command.

Options in Detail

Methods

This tool features 3 different methods:

- **Start From First Feature**
All features starting from the first feature in the **Feature Tree** will be exported into **Inventor**
 - **Resume From Selected Feature**
All features starting from a selected feature in the **Feature Tree** will be exported into **Inventor**
 - **Only Selected Entities**
Only selected features will be exported into **Inventor**
-

Options

Inventor (Version) – Chooses a dedicated software version.

Note:

When different versions of Inventor are installed in the same PC, the **Inventor (Version)** option will be displayed.

Prefix For Feature Name – Defines the prefix for a feature name.

Try To Transfer Sweep/Loft As Parametric Feature – Transfers **Sweep** and **Loft** features as parametric features into **Inventor**, however if the sweeping and lofting algorithms are different between the applications then the features will be transferred as dummy objects.

Feature – Selects a feature to start transferring.

Entities – Selects entities only to transfer.

Inventor Options

Use Custom Inventor Execution File Path – Manually defines the path of an **Inventor** application execution file.

Pause At ...

Pause At – Pauses a transfer before a selected command is performed.

See Also

[File > LiveTransfer\(TM\) > To SolidWorks](#)

[File > LiveTransfer\(TM\) > To Siemens NX](#)

[File > LiveTransfer\(TM\) > To Creo\(Pro/E\)](#)

[File > LiveTransfer\(TM\) > To AutoCAD](#)

[File > LiveTransfer\(TM\) > Save As CATIA V4](#)

[File > LiveTransfer\(TM\) > Save As CATIA V5](#)

File > LiveTransfer(TM) > To Solid Edge

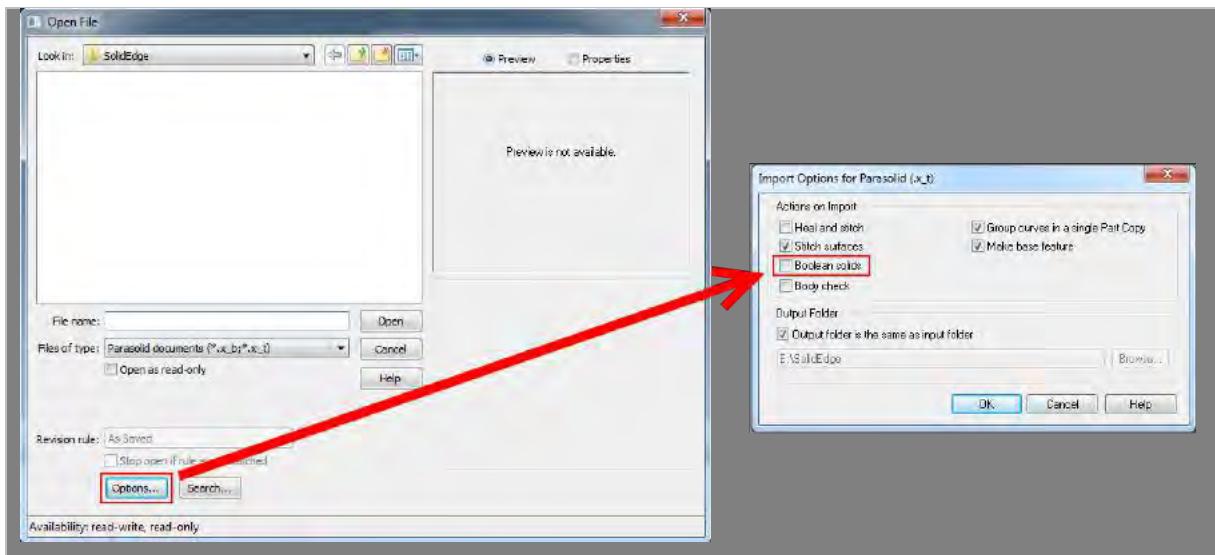
What

The **LiveTransfer™ To Solid Edge** command is used to directly export solid and surface bodies to the **Solid Edge** application.



Note:

The Boolean option in the import function of Solid Edge must be turned off before using this command if a user wants to transfer multiple bodies without the bodies getting merged.



Why

The **LiveTransfer™ To Solid Edge** tool is useful for:

- Transferring data into the **Solid Edge** application

How to Use

How use the LiveTransfer™ To Solid Edge command

1. Go to **File > LiveTransfer(TM) > To Solid Edge**.

Note:

The application currently supports **Solid Edge ST4, ST5, and ST6**. Only one version of **Solid Edge** can be installed on a computer at one time.

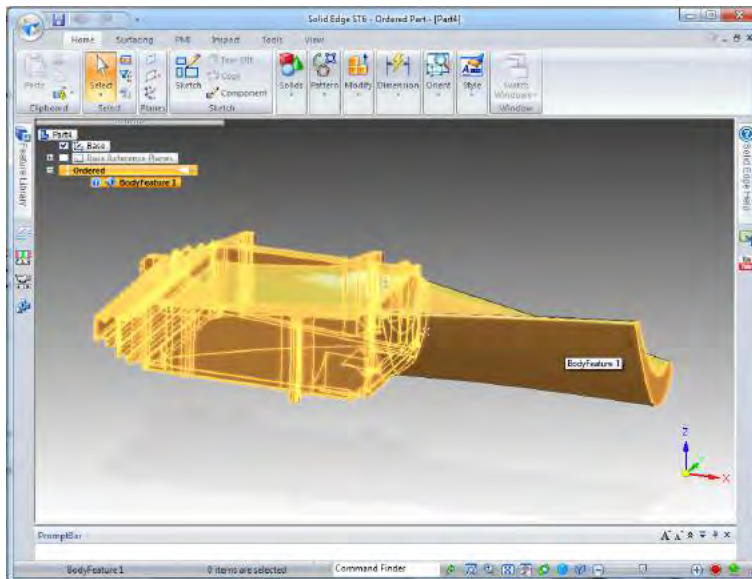
2. Select solid bodies or surface bodies as the **Entities** in the dialog tree.

Note:

Only solid and surface bodies can be transferred.



3. Click **OK** to finish the command.



Options in Detail

Options

Prefix For Feature Name – Defines the prefix for a feature name.

Entities – Selects entities only to transfer.

Solid Edge Options

Use Custom Solid Edge Execution File Path – Manually defines the path of a **Solid Edge** application execution file.

See Also

[File > LiveTransfer\(TM\) > To Siemens NX](#)

[File > LiveTransfer\(TM\) > To Creo\(Pro/E\)](#)

[File > LiveTransfer\(TM\) > To AutoCAD](#)

[File > LiveTransfer\(TM\) > To Inventor](#)

File > LiveTransfer(TM) > Save As CATIA V4

What

The **LiveTransfer™ Save As CATIA V4** command is used to save files in the **CATIA 4.1.9** format. **Sketches** and **Bodies** can be saved in the .model format.

Why

The **LiveTransfer™ Save As CATIA V4** tool is useful for:

- Saving data as the **CATIA V4** file format

How to Use

How use the LiveTransfer™ Save As CATIA V4 command

1. Go to **File > LiveTransfer(TM) > Save As CATIA V4**.

Note:

To use this command the Geomagic Exchange™ License must be purchased and installed

2. Type a File name and select **Except Hidden Entities** and **Use Custom Settings** option boxes if desired.
3. Click **Save** to finish the command.

Options in Detail

Options

Except Hidden Entities – Will not export hidden sketches and bodies.

Unit – Defines the measurement unit of the entity to be saved.

See Also

[File > LiveTransfer\(TM\) > To SolidWorks](#)

File > LiveTransfer(TM) > To Siemens NX
File > LiveTransfer(TM) > To Creo(Pro/E)
File > LiveTransfer(TM) > To AutoCAD
File > LiveTransfer(TM) > To Inventor
File > LiveTransfer(TM) > Save As CATIA V5

File > LiveTransfer(TM) > Save As CATIA V5

What

The **LiveTransfer™ Save As CATIA V5** command is used to save files in the **CATIA V5 R6** format. **Sketches** and **Bodies** can be saved in the .model format.

Why

The **LiveTransfer™ Save As CATIA V5** tool is useful for:

- Saving data as the **CATIA V5** file format

How to Use

How use the LiveTransfer™ Save As CATIA V5 command

1. Go to **File > LiveTransfer(TM) > Save As CATIA V5**.

Note:

To use this command the Geomagic Exchange™ License must be purchased and installed

2. Type a File name and select **Except Hidden Entities** and **Use Custom Settings** option boxes if desired.
3. Click **Save** to finish the command.

Options in Detail

Options

Except Hidden Entities – Will not export hidden sketches and bodies.

Unit – Defines the measurement unit of the entity to be saved.

See Also

[File > LiveTransfer\(TM\) > To SolidWorks](#)
[File > LiveTransfer\(TM\) > To Siemens NX](#)
[File > LiveTransfer\(TM\) > To Creo\(Pro/E\)](#)
[File > LiveTransfer\(TM\) > To AutoCAD](#)
[File > LiveTransfer\(TM\) > To Inventor](#)

7.1.13. File > Batch Process

What

The **Batch Process** commands execute a series of programs or “jobs” on a computer without manual intervention.

Why

The **Batch Process** commands are useful for:

- Processing jobs when computer resources are less busy
- Avoiding idling computing resources with minute-by-minute intervention and supervision
- Sharing optimized processing with others
- Fully utilizing computing resources, helping to amortize the cost of the machine

See Also

[File > Batch Process > Run](#)

File > Batch Process > Design

What

The **Batch Process Design** command is used to plan a series of commands and set parameters for a batch process.




Why

The **Batch Process Design** tool is useful for:

- Easily and intuitively designing a batch job

How to Use

How design a Batch Process

1. Go to **File > Batch Process > Design** or click the icon in the toolbar.
2. Click the **New Batch Process** button  to create a new batch job.
3. Drag and drop the **Align** icon into the right panel.

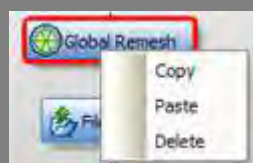


4. Add more commands using the same method. The processing order can be changed by dragging the icons in the right panel. The process will start from the top to the bottom.

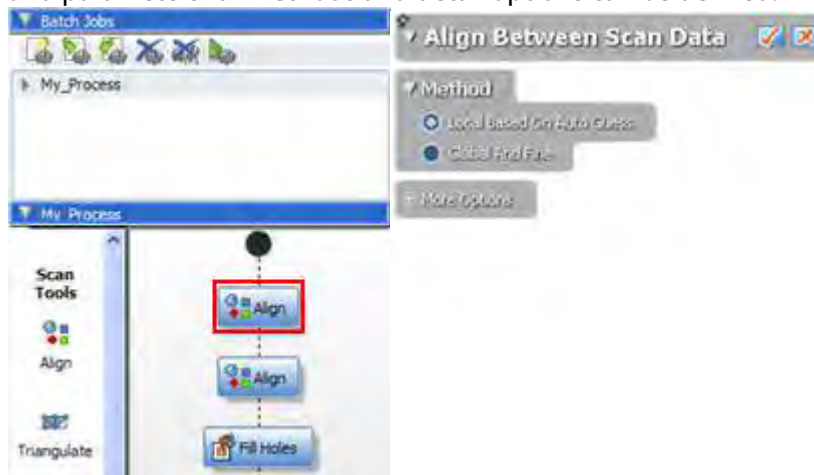


Note:

Copy, Paste, and Delete commands using the pop-up menu for each command



- Click added commands in the right panel to set its parameters. A dialog tree will appear and parameters for methods and detail options can be defined.



- The batch process automatically saves and when the **Batch Process Design** command is executed the last batch state will be show. The batch process can be exported using the **Export Batch Process** button. To execute the batch process, go to **File > Batch Process > Run** or click the **Run Batch Process** button.

The batch file is automatically saved as an .xml formatted file at the following location:
 ..\Users\User Name\Documents\Product Name\Configured\BatchProcess

How to use a batch template

When you run the **Batch Process Design** or **Run** commands, you can use or edit default batch template files.

Auto Surfacing (Mesh only, XDL out)-[General] – Generates fitting surfaces using **Auto Surface** with the **Feature Following Network** options based on a mesh and saves the final data in the .xdl format.

Data Conversion_Any Scan Format to ICF-[General] – Used to reduce the file size of point clouds or meshes. Files are saved in the .icf format.

General Mesh Generation (XDL out)-[Object Scan] – Generates a single complete mesh from non-aligned multiple meshes and saves the final data in the .xdl format.

MeshBased_1 Data Polishing (XDL out)-[Long Range] – From MeshBased_1 to 4 are continuous processes used to generate mesh from a long range scan data. This batch file removes noise in a point cloud and saves the final data in the .xdl format.

MeshBased_2 Alignment (XDL out)-[Long Range] – The second step for long range scan data processing. It aligns multiple point clouds using the **Local Based On Auto Guess** and **Global And Fine** options. The final data is saved in the .xdl format.

MeshBased_3 Triangulation (XDL out)-[Long Range] – The third step for long range scan data processing. It generates meshes using **2D Triangulation** with the **Spherical** option. The final data is saved in the .xdl format.

MeshBased_4 Single Mesh Generation (XDL out)-[Long Range] – The fourth step for long range scan data processing. It merges meshes creating a single mesh. The final data is saved in the .xdl format.

PointBased_1 Data Polishing (XDL out)-[Long Range] – PointBased_1 to 4 are continuous processes used to generate a single point cloud from long range scan data. This batch file removes noise in a point cloud and saves the final data in the .xdl format.

PointBased_2 Alignment (XDL out)-[Long Range] – The second step for long range scan data processing. It aligns multiple point clouds using the **Local Based On Auto Guess** and **Global And Fine** options. The final data is saved in the .xdl format.

PointBased_3 Single Point Cloud Generation (ICF out)-[Long Range] – The third step for long range scan data processing. It generates a single combined point cloud. The final data is saved in the .xdl format.

Watertight Mesh Generation for FEM (XDL out)-[Object Scan] – Generates a single watertight mesh from non-aligned multiple meshes and saves the final data in the .xdl format.

Options in Detail



New Batch Process – Creates a new batch process.



Import Batch Process – Imports a batch process saved in the .xml format.



Export Batch Process – Exports a batch process in the .xml format.



Delete Batch Process – Removes a selected batch process.



Delete All Batch Processes – Removes all batch processes.



Run Batch Process – Executes a selected batch process which can also be accessed by going to **File > Batch Process > Run**.

See Also

[File > Batch Process > Run](#)

File > Batch Process > Run

What

The **Run Batch Process** command is used to execute a designed batch job.



Why

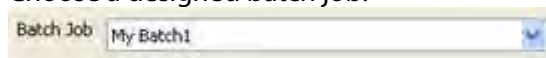
The **Run Batch Process** tool is useful for:

- Running designed batch jobs

How to Use

How run a Batch Process

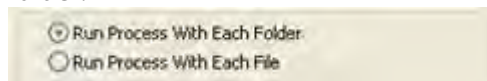
1. Go to **File > Batch Process > Run** or click the icon in the toolbar.
2. Choose a designed batch job.



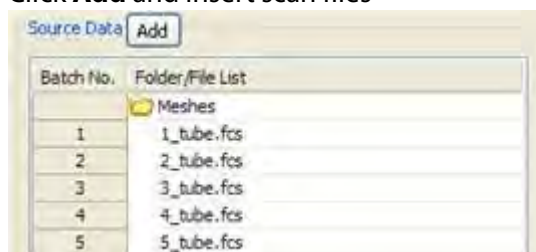
The processes for the selected job will be displayed in a dialog box.



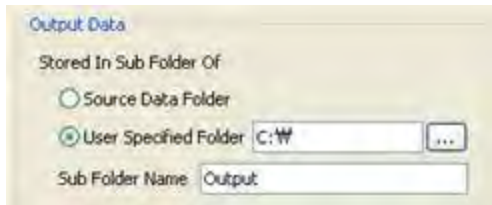
3. Choose the **Run Process With Each Folder** option to process all scan files in a specific folder.



4. Click **Add** and insert scan files



5. Define the location and folder name to output to.



6. Click the **Start Batch Process** button.

Options in Detail

Methods

Run Process With Each Folder – Runs the batch process using folders.

Run Process With Each File – Runs the batch process using individual files.

Source Data

Add – Adds source data.

Delete – Deletes source data.

Filter – Filters the source data.

Output Data

Source Data Folder – Saves output files in the same folder as the source files.

User Specified Folder – Saves output files to a user-specified folder.

Sub Folder Name – Creates a sub-folder in a user-specified folder.

Save Final Model File As XRL File – Saves output files as .xrl formatted files.

Same Name As Source Data – Saves output .xrl file with the same name as the source data file.

User Specified Name – Saves output .xrl file with a user-specified name.

Generate Log File – Creates a log file of the batch process in a .txt formatted file.

File Path – Chooses a file path for a log file.

Action When Error Occurs

Ignore Error And Continue Batching All Source Data – Continues batching if an error is detected.

Stop Batching Current Source Data And Skip to Next Source Data – Skips the current batch process if an error is detected.

Stop Entire Batch Process – Stops the entire batch process if an error is detected.

See Also

[File > Batch Process > Design](#)

7.1.14. File > Preferences

What

The **Preferences** commands manage global options that affect all objects in the viewing window. Options are saved to each .xpf file and can be used later.

Note:

After updating the application, customized settings files from older versions of the application such as the Template, MaterialInfo.xmt.rfPreference.xpf, rfSpaceball.cfg, and rfToolBarInfo.cfg can be used again. These files are located at: `..\Users\User Name\Documents\Product Name`

Why

The **Preferences** commands are useful for:

- Changing colors
- Changing line, point, and edge sizes
- Changing global tolerances
- Changing hardware settings
- Changing input/output settings
- Changing general settings

How to Use

How to set up Preferences

1.

Note:

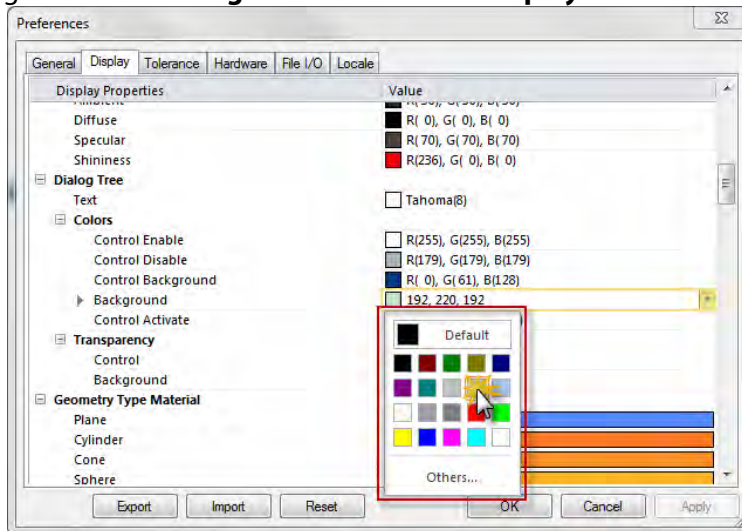
When starting the application for the first time the **Initial Setup Dialog** box will be displayed. **True** or **False** can be selected for the option to **Use Previous Version** of customized settings. To enable the option make sure the setting file of previous versions is located at: `C:\Documents and Settings\User Name\My Documents\Product Name`.

The settings for toolbars and customized toolbars can be imported, however the position of

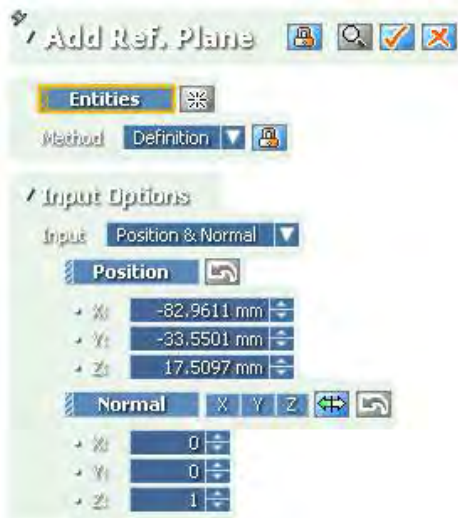
customized toolbars cannot be imported and will be in a hidden state by default.

Go to **File > Preferences**.

2. The **Preference** dialog box will be displayed. Change the **Background** color to light green in the **Dialog Tree** section in the **Display** tab.



3. Click **Apply** to apply changes without closing the dialog box. The color of the dialog tree will become green.



4. Change preferences as desired and click **OK** to accept or **Cancel** to exit **Preferences**. **Preferences** can be exported as a .xpf file by using the **Export** button, and can also be imported by using the **Import** button. The **Reset** button will return the **Default Decimal Place, Default Measure Unit, Thousand Separators** preferences as well as all entities in the display tab to their default states.

Options in Detail

General Tab

Enable Multi-thread – Enables multiple core or multiple thread enabled processors to utilize multiple threads, speeding up the modeling process.

Template XRL File – Specifies the xrl file that is used as the template.

Update Product Automatically – Updates the application automatically when the option is set to **True** and the program is launched.

Cache Folder For Unloading And Backup – Specifies a directory for temporary file caching and is located in the user's temporary directory assigned by the system environment variables by default.

Note:

When working with large files it is recommended that the cache folder is set to a separate dedicated hard disk drive. Setting the virtual memory to a separate dedicated hard disk drive can also improve performance, especially when using Microsoft Windows Vista OS.

Show Support At Startup – Show the Support tab whenever the application is launched when set to **True**.

Auto Save Reminder In Every X Minutes – Sets the amount of time in-between save dialog box reminders.

View Manipulation Style – Selects a view manipulation style (zooming, panning, and rotating) from either Rapidform, SolidWorks, Siemens NX, Creo(Pro/E), CATIA, PolyWorks, or Geomagic.

Zooming Center Of Mouse Wheel Scroll – Sets the zooming center to the position of the mouse cursor or the center of the screen.

Rotation Angle Of Down/Up Arrow Key (degree) – Sets the rotation angle in the model view when using the arrow keys on the keyboard.

Undo Depth (times) – Sets the maximum amount of actions saved for the Undo command.

Default Decimal Place – Sets the number of digits displayed after a decimal point between 1 and 10.

Default Input (Scanner) Unit – Sets a default input unit. If different unit has been defined in the files, prompt will be displayed and ask to change the unit while importing files.

Note:

The specified default scanner unit will be used for the file unit if the scanner files do not have an internal unit definition. This is different than the **Default Measure Unit** parameter.

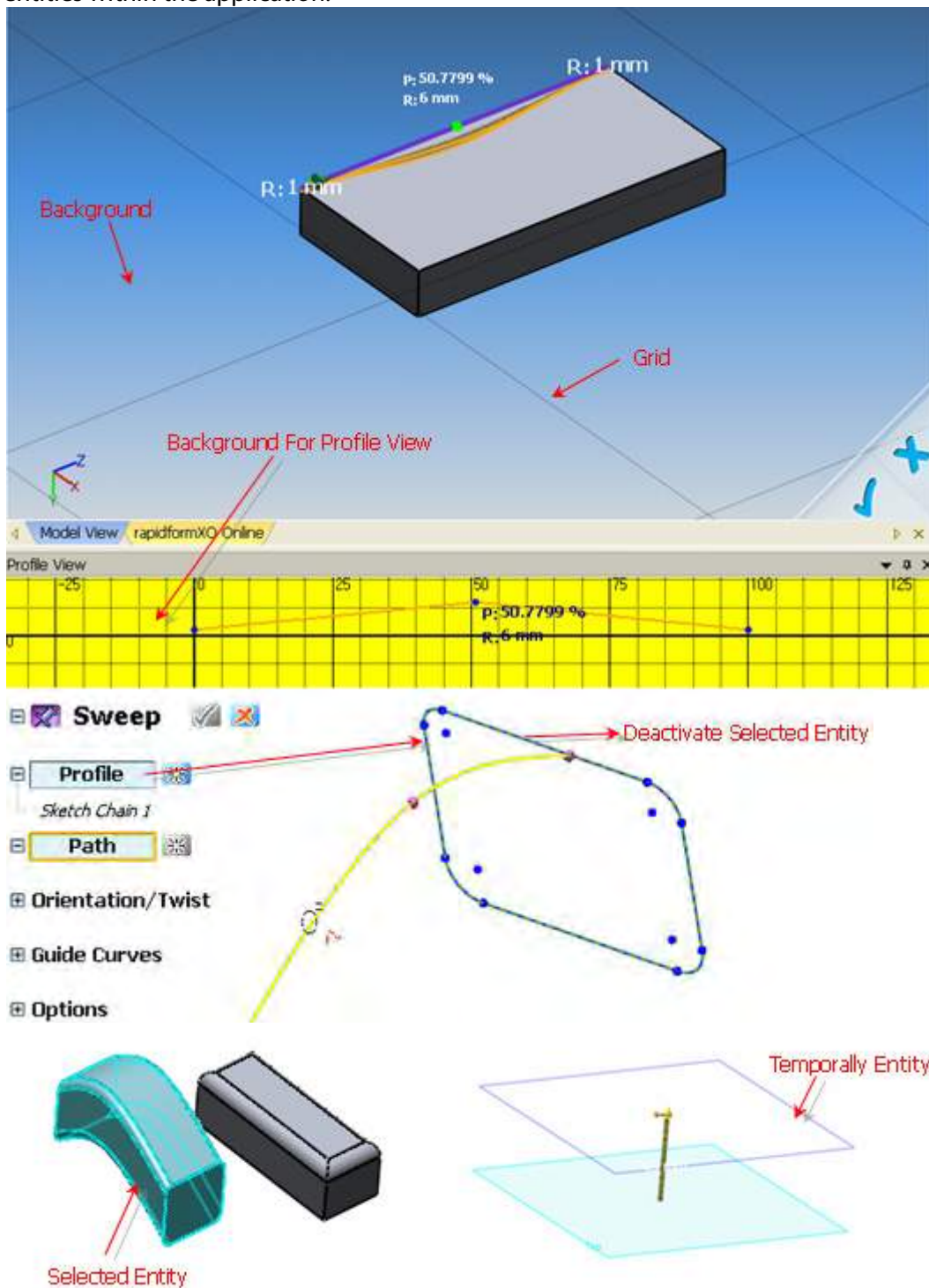
Default Measure Unit – Sets the default measurement units for models.

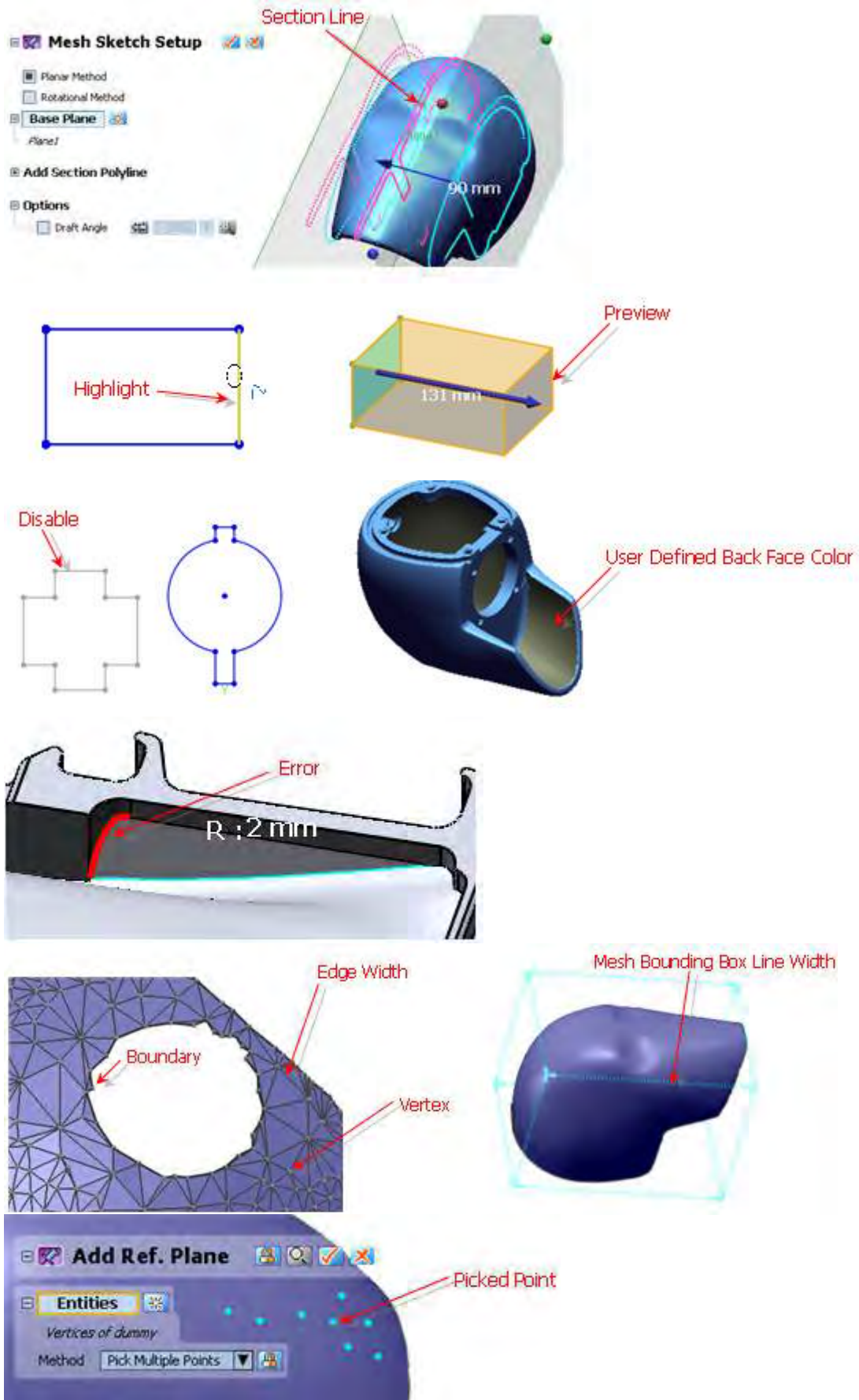
Thousand Separators – Separates thousands from hundreds for Real Value and Integer Value numbers that have four or more places to the left of the decimal separator when set to **True**.

Sound – Specifies the sounds used when command completes, breaks, or sends a message and can be either the default application sound or a user-specified sound.

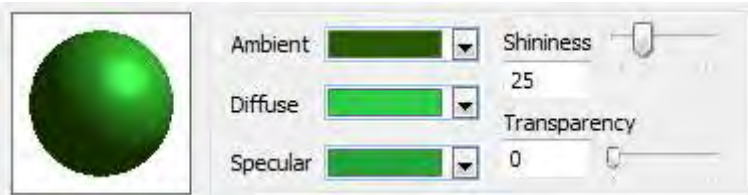
Display Tab

Draw Properties – Sets the color, size, width, backgrounds, and other draw properties for entities within the application.

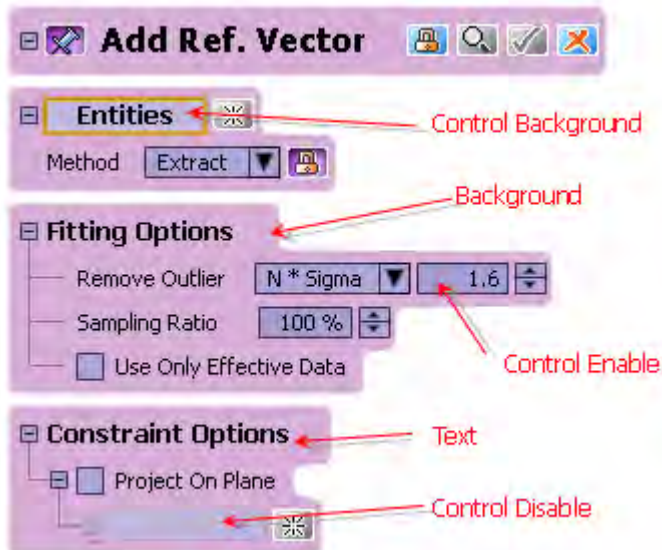




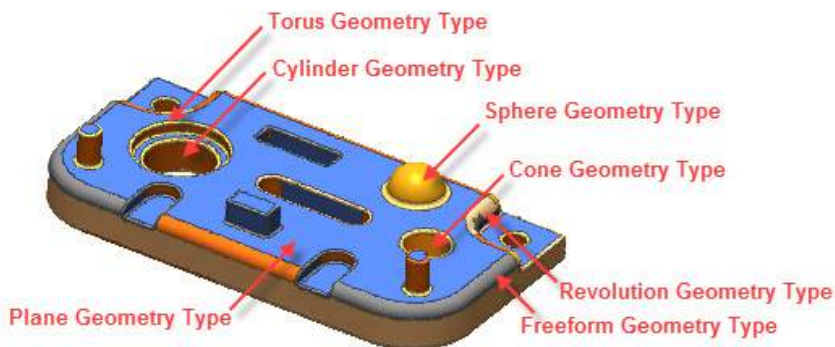
Color Mesh – Sets the color of mesh face with Ambient, Diffuse, Specular, and Shininess.



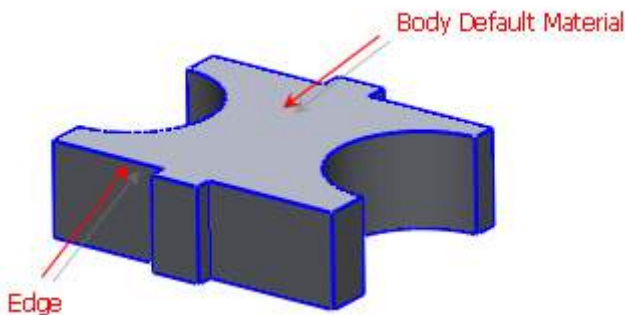
Dialog Tree – Sets the color, font, and transparency for dialog trees.



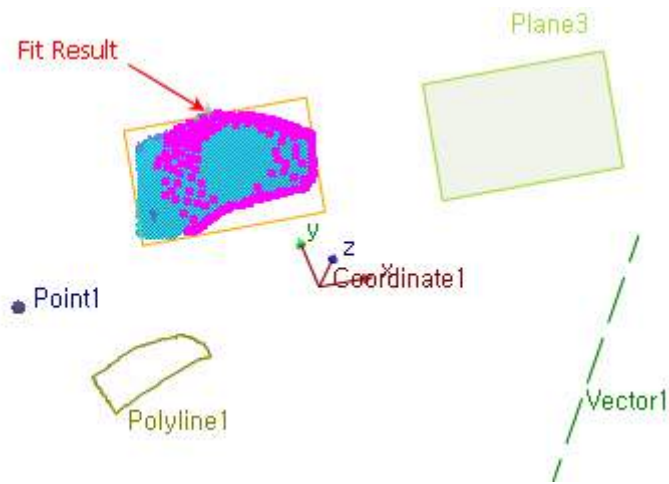
Geometry Type Material – Sets the default colors for geometry type materials.



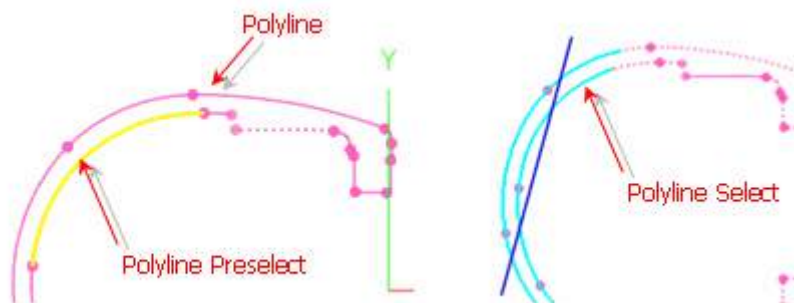
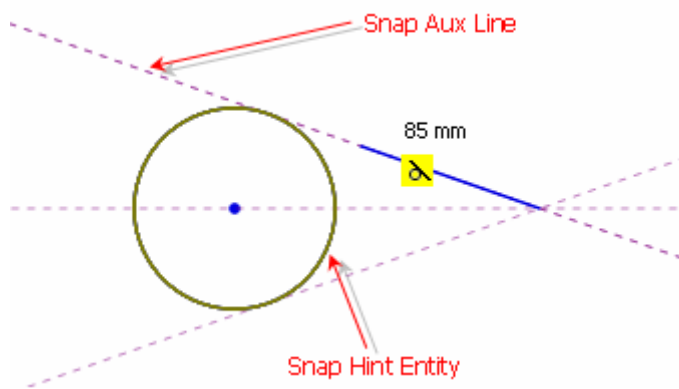
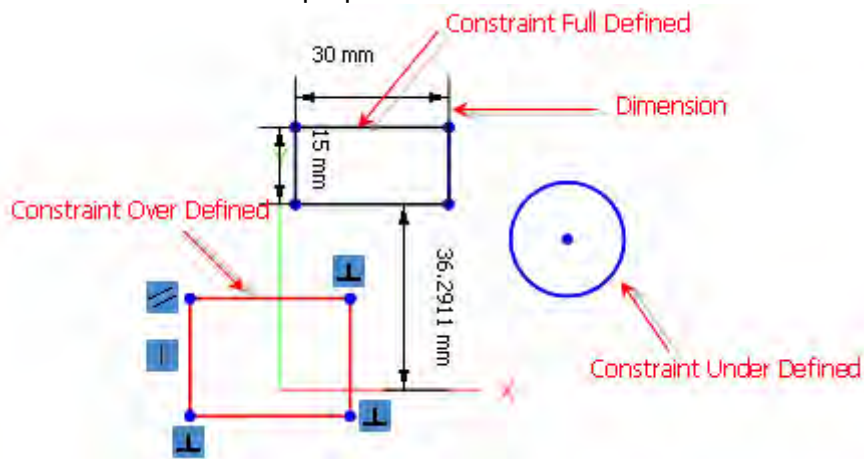
Body – Sets the default body material colors.

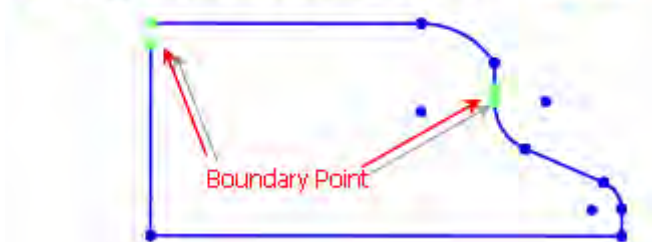
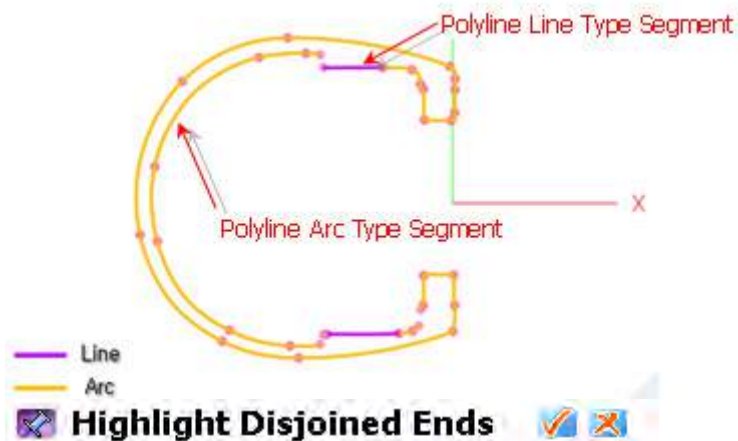


Ref. Geometry – Sets the default properties for ref. geometry.

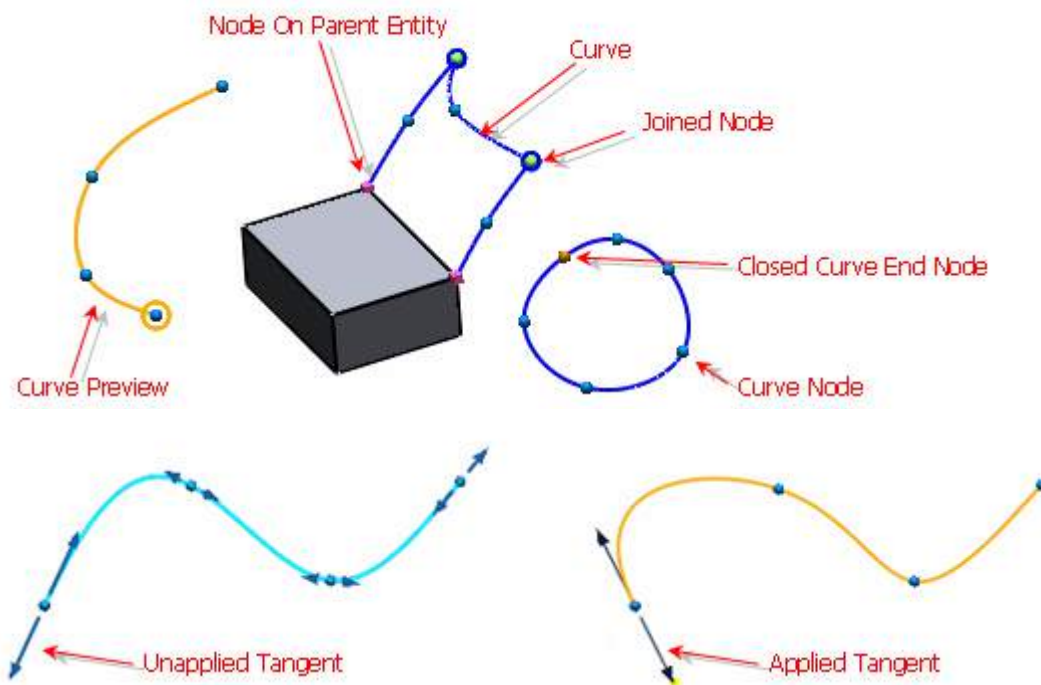


Sketch – Sets the default properties for sketches.

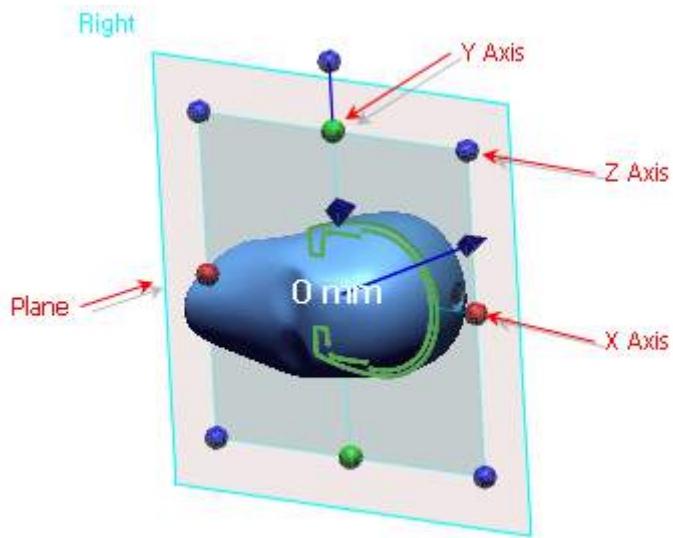




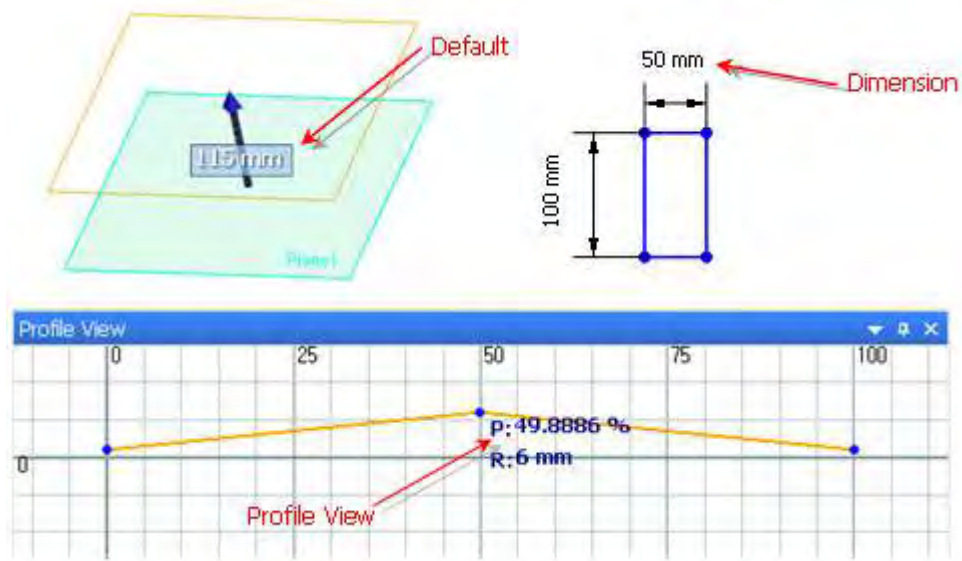
3D Sketch – Sets the default properties for sketches.



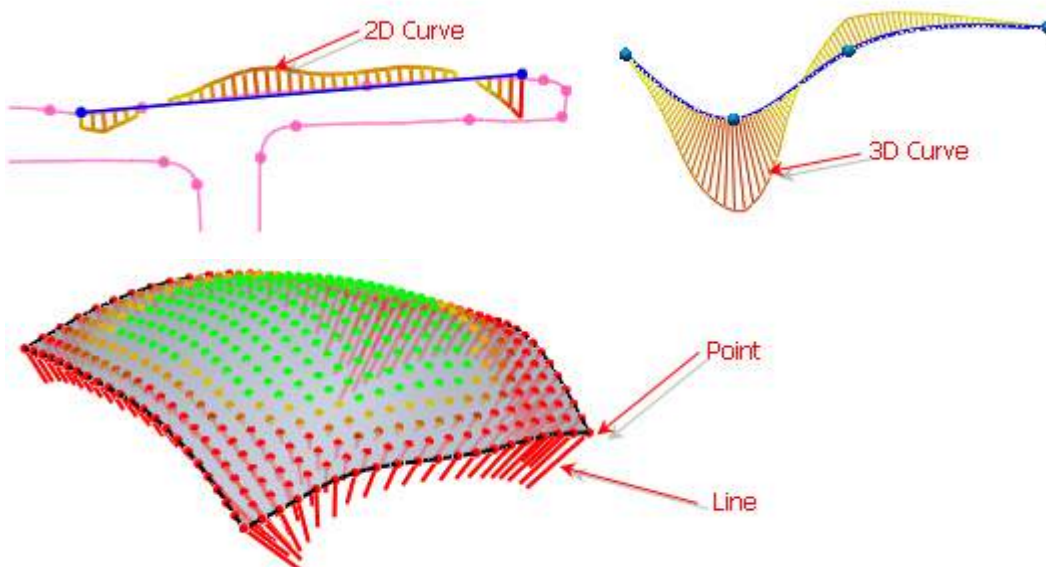
Manipulator – Sets the default properties for manipulators.



Widget – Sets the font if text in widgets.



Accuracy Analyzer™ – Sets the default properties for the Accuracy Analyzer™.

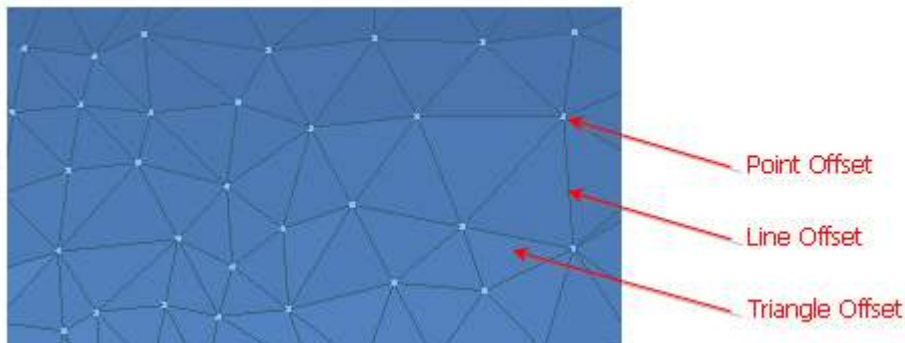


OpenGL – Sets the default properties for OpenGL.

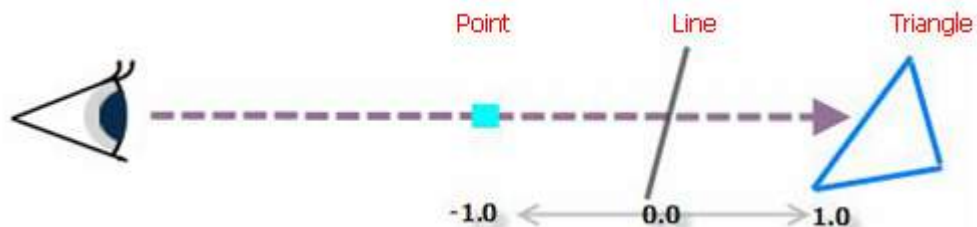
Triangle Offset – Displays the offset value of triangles when overlapping with other entities.

Line Offset – Displays offset values of lines when overlapping with other entities.

Point Offset – Displays offset values of points when overlapping with other entities.



The value range can be set from **-1.0** to **1.0**. If the value is set as **-1.0**, the entities closer to the viewer will be displayed. If the value is set as **1.0**, entities farther from the viewer will be displayed. Using this option can define the overlapping order of entities.



Picking Window Size (pixel) – Sets how many pixels from the mouse cursor the selection range will consider. This is used to fine-tune the selection of a position when picking target entities.

Enable Animation – Enables animation effects. When this option is set to True, animation effects will be applied when changing viewpoints or zooming a model.

Tolerance Tab

Mesh – Sets the Outlier For Allowable Deviation % value.

Outlier For Allowable Deviation (%) – Ignores the poly-vertices that cause the largest deviations within a given value while creating fitting surfaces from mesh data. After creating fitting surfaces, the area that will be less than this value.

Geometry Type Checking Control – Sets the criteria of geometry type classification.

Outlier For Geometry Type Checking (%) - Ignores poly-vertices that cause the largest deviations within a given value when fitting mesh regions to analytic geometry. After creating analytic geometry, the area will be less than this value.

Geometry Type Checking Tolerance (mm) – Tries to fit selected regions to specific analytic geometry after removing outliers by the value of the Outlier For Geometry Type Checking. If the maximum deviation is within this value, the application will recognize that analytic geometry.

Maximum Radius Of Analytic Shape (mm) – Accepts estimated analytic regions if their

radius values are less than a given value when the application estimates selected regions to be analytic geometry. This value prevents the application from estimating a selected region as a meaningless analytic geometry.

Geometry Fitting Control – Sets the geometry fitting control properties.

Set Maximum Point Number For Geometry Fitting And Type Finding – Limits the number of points used for fitting when this option is True. When it is False, all points inside of the searching zone will be used for fitting.

Maximum Point Number – Defines the maximum number of points to be used for fitting.

Hardware Tab

Processor Feature Support – Sets processor feature support properties. SSE2 (Streaming SIMD Extensions), SSE3 or SSSE3 can be enabled which are provided by a CPU for higher processes. These optimized settings are automatically set by the application after installing or running the Performance Profiler (Help > Launch Performance Profiler).

OpenGL Renderer Setup – Sets OpenGL rendering properties.

Use Trackball Depth Average – Determines whether OpenGL depth buffer is used for calculating rotation center.

Trackball Depth Average Resolution – Sets the resolution of queries of OpenGL depth buffer for calculating rotation center.

Frame Buffer Save/Restore – Chooses which routine is used for saving and restoring the frame buffer. The rendering and speed settings are automatically set by the application after installing or running the Performance Profiler (Help > Launch Performance Profiler).

Off-Screen Rendering Mode – Determines which graphic acceleration is used while capturing view. If a graphic device supports hardware acceleration, it is recommended setting this option as Hardware.

Hardware Multi-thread Rendering Support – Sets multi-threaded rendering support when Hardware is selected in the Off-Screen Rendering option by selecting true or false. Set to false when hardware cannot support this option.

Use Video RAM For Rendering Acceleration – Loads data into video memory and speeds up rendering performance when this option is set to True. The default setting is False, however, it is recommended to turn on this option for most graphic cards when handling millions of meshes to prevent any abnormal display problems.

Use Shader Programs For Rendering Acceleration - Determines whether shader programs are used for enhancing rendering speed.

Scanner Direct Control – Sets frequently used scanners so when the **LiveScan™** command is used only those scanners will be displayed in the Devices pull down menu.

File I/O

Default Import File Formats – Sets True/False values for importing commonly used file formats. Only commonly used formats will be displayed in the **Import** dialog box.

Locale

Language – Sets the display language of the application. Installed language packs will be listed.

Locale Format – Sets the numbers and time display format.

See Also

[Docking Bars > Display](#)

7.1.15. File > Summary Info

What

The **Summary Info** command allows for the input of summary information for a model.

Note:

While opening a saved file, the summary information shows if the **Show Preview & Info Box** option is checked in the **Open** dialog box.

Why

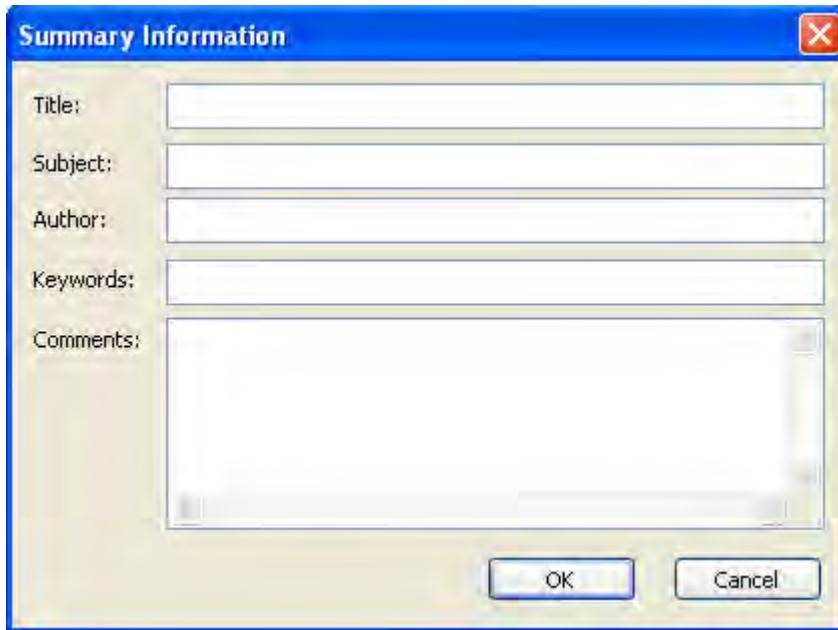
The **Summary Info** command is useful for:

- Changing basic file summary information

How to Use

How to change summary information

1. Go to **File > Summary Info**.
2. The **Summary Information** dialog box is displayed.



3. Change summary information as desire and click **OK** to accept or **Cancel** to exit the **Summary Information** command.

7.1.16. File > Recent Files

What

The **Recent Files** command lists the recently used files in the application.

Why

The **Recent File** command is useful for:

- Conveniently viewing recently used files

How to Use

How to view and open recent files

1. Go to **File > Recent Files**. Mouse-over the command and a black arrow pointing to the right will be displayed showing recently opened files.
2. Select a file to open it.

See Also

7.1.17. File > Exit

What

The **Exit** command closes the application.

Why

The **Exit** command is useful for:

- Closing the application after a project is finished
- Closing the application when the system is not in use to free computing resources

How to Use

How to exit the application

1. Go to **File > Exit**.
2. If there are unsaved changes a dialog box will be displayed.
3. Save files as desire and click **OK** to accept or **Cancel** to exit the **Exit** command.

See Also

[File > Save](#)

[File > Save As](#)















7.2. Select

Selection


When most of commands are executed, the selection mode is automatically activated. The Select menu features advanced selection tools, which allows the selection of various entities in the Model View.

Select Filter

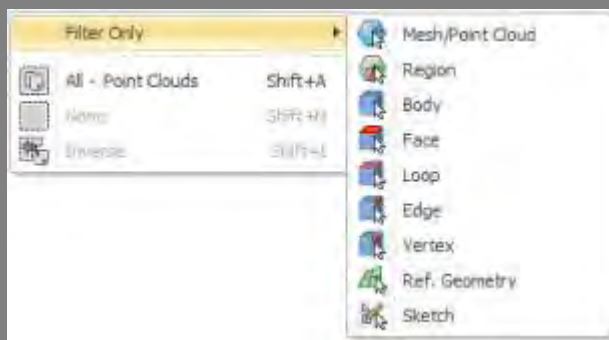
Select various kinds of entities within in the application.

-  **(Filter Meshes)** - Allows the selection of meshes only.
-  **(Filter Regions)** - Allows the selection of regions only.
-  **(Filter Poly-faces)** - Allows the selection of poly-faces only.
-  **(Filter Poly-vertices)** - Allows the selection of vertices and points only.
-  **(Filter Mesh Boundaries)** - Allows the selection of mesh boundaries only.
-  **(Filter Bodies)** - Allows the selection of bodies only.
-  **(Filter Faces)** - Allows the selection of faces only.
-  **(Filter Loops)** - Allows the selection of loops only.
-  **(Filter Edges)** - Allows the selection of edges and curves only.
-  **(Filter Vertices)** - Allows the selection of vertices and points only.
-  **(Filter Ref. Geometries)** - Allows the selection of ref. geometries only.
-  **(Filter Sketches)** - Allows the selection of sketches only.
-  **(Filter Constraints & Dimensions)** - Allows the selection of constraints and dimensions only.
-  **(Clear All Filters)** - Clears all selection filters (Shift + F).

Note:

When a selection filter is on but the cursor is over an empty area, an icon () beside the cursor will appear.

- When the **Mode** is changed, the **Selection Filters** will be reset.
- **Selection Filters** can be quickly accessed by the pop-up menu.



7.2.1. Select > Mode

What

The **Select Mode** command configures the shape of the selection tool.

Why

The **Select Mode** command is useful for:

- Changing the shape of the selection tool to provide flexibility in selecting entities

How to Use

How to use the selection tool

1. Go to **Select > Mode** or click the icon in the toolbar.
2. Choose the selection mode type.
3. Drag the selection tool over entities to select. To add a current selection to selections already made, use the **Shift** button. To remove a current selection from selections already made, use the **Ctrl** button.

Options in Detail

Note:

All selection modes are able to select poly-vertices and poly-faces. The Rectangle selection is the default selection mode in the application and can select any entities.

Line – Selects a straight line by clicking at one end of a supposed line and dragging to another.



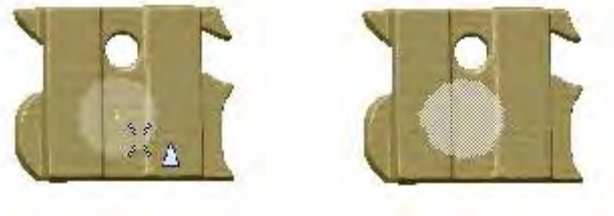
Note:

The width of a line can be edited by using Alt + left mouse buttons together.

Rectangle – Selects a region by dragging from one corner of an imaginary box to the opposite diagonal corner.



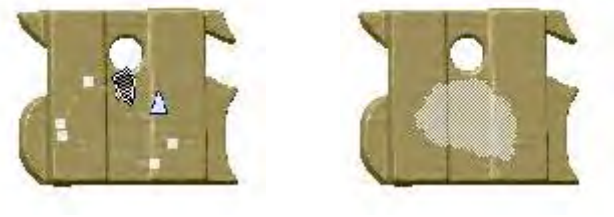
Circle – Selects a circular region by clicking in the center of a supposed circle and dragging outward.



Note:

The radius of a circle can be edited by using Alt + left mouse buttons together

Polyline – Selects the inside of a polygon by picking points in the viewing window and clicking the RMB or double-clicking the LMB to finalize.



Freehand – Selects an irregularly shaped region by tracing a perimeter with the mouse while dragging.



Paint Brush – Selects regions that the mouse cursor covers during the dragging.



Note:

The range of the paint brush can be edited by using Alt + left mouse buttons together

Flood Fill – Selects poly-faces within same boundary.



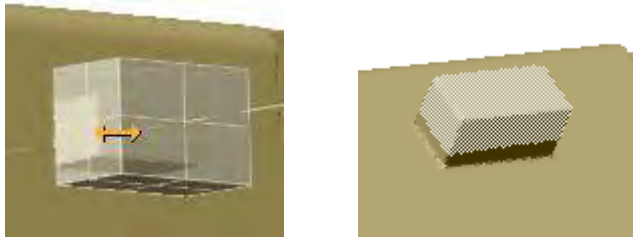
Extend To Similar – Select areas that consist of similar features.



Extend To Sudden Change – Selects areas that have similar curvatures.



Box – Configures the shape of the selection tool as a box.



Cylinder – Configures the shape of the selection tool as a cylinder.

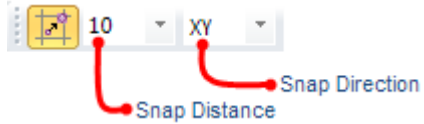


Sphere – Configures the shape of the selection tool as a sphere.



Visible Only – Decides whether selecting can take place on a visible area only or in space.

Position Snap – Snaps to meshes or bodies when an object is picked. The snap distance and snap plane can be defined in the bottom toolbar.



See Also

[Mode](#)

7.2.2. Select > All

What

The **Select All** command selects all entities.

Why

The **Select All** command is useful for:

- Selecting all entities in the model view

How to Use

How to select all entities

1. Go to **Select > All – Poly-Faces** in the **Mesh mode**, click the icon in the toolbar, or use the **Shift + A** keyboard shortcut. All poly-faces can also be selected by using the **RMB** (Right Mouse Button) and selecting all in the pop-up menu.



2. All poly-faces will be selected.



See Also

[Select > None](#)
[Select > Inverse](#)

7.2.3. Select > None

What

The **Select None** command de-selects all entities.

Why

The **Select None** command is useful for:

- Selecting all entities in the model view

How to Use

How to de-select all entities

1. Select poly-faces in the model.



2. Go to **Select > None**, click the icon in the toolbar, or use the **Shift + N** keyboard shortcut. All poly-faces can also be de-selected by using the **RMB** (Right Mouse Button) and selecting none in the pop-up menu.

3. No poly-faces will be selected.



See Also

[Select > All](#)
[Select > Inverse](#)

7.2.4. Select > Inverse

What

The **Select Inverse** command selects the inverse of selected entities.

Why

The **Select Inverse** command is useful for:

- Selecting the inverse of already selected entities

How to Use

How to select the inverse of entities

1. Select poly-faces in the model.



2. Go to **Select > Inverse**, click the icon in the toolbar, or use the **Shift + I** keyboard shortcut. The inverse can also be selected by using the **RMB** (Right Mouse Button) and selecting the inverse in the pop-up menu.
3. The inverse of the originally selected poly-faces will be selected.



See Also

[Select > All](#)

[Select > None](#)

7.2.5. Select > Custom Region

What

The **Select Custom Region** command selects an area made by user selected vertices.

Why

The **Select Custom Region** command is useful for:

- Selecting or de-selecting complex regions

How to Use

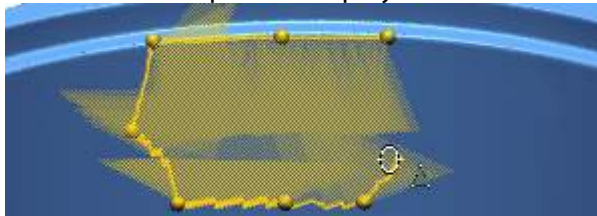
How to select a custom region

1. Go to **Select > Custom Region** or click the icon in the toolbar.

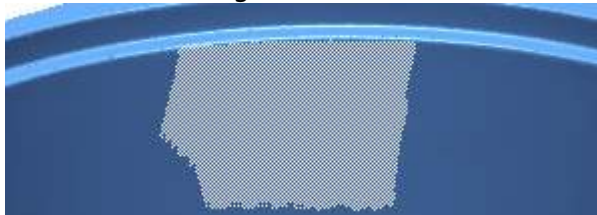
Note:

The **Custom Region** command can only be used in the **Mesh mode**.

2. Select the **Add** option. Pick poly-vertices to form a region.



3. Click on the starting node or use the **Close Path** button to close the path.



4. Click **OK** to finish the command.

Options in Detail

Custom Path – Creates a custom path.

Note:

A custom path drawn in a counterclockwise direction selects an inner region, but creating a custom path in a clockwise direction selects an outer region. The **Undo Last Path** button is used to cancel the last selection made, the **Close Path** button to close and finalize the custom path, and the **Reset** button to discard all selections made

Add – Adds a selected region to existing selections.

Remove – Removes a selected region from the existing selections.

Inverse – Selects the inverse regions of existing selections.

See Also

[Select > All](#)

[Select > None](#)

[Select > Inverse](#)

7.2.6. Select > Boundary Entities

What

The **Select Boundary Entities** command selects poly-vertices or poly-faces around selected boundaries.

Why

The **Select Boundary Entities** command is useful for:

- Selecting complex regions around a boundary

How to Use

How to select boundary entities

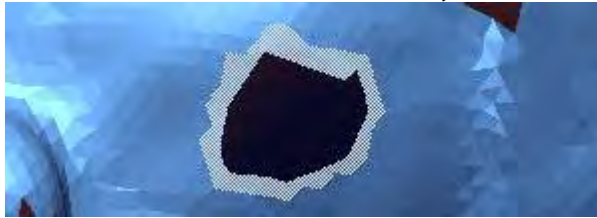
1. Go to **Select > Boundary Entities** or click the icon in the toolbar.



2. Change the **Type To Select** to **Poly-Faces** and select the boundary.



3. Click **OK** to finish the command. Poly-faces around the boundary will be selected.



Options in Detail

Type To Select – Chooses whether to select poly-faces or poly-vertices around a boundary.

Seed Boundaries – Selects boundaries on a mesh in the model view.

See Also

[Select > All](#)

[Select > None](#)

[Select > Inverse](#)

7.2.7. Select > Entities Around Edges/Faces

What

The **Select Entities Around Edges/Faces** command selects poly-vertices or poly-faces around selected entities.

Why

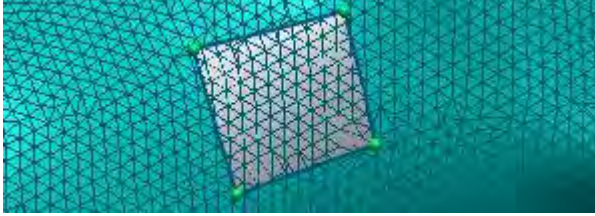
The **Select Entities Around Edges/Faces** command is useful for:

- Selecting entities around created edges and faces

How to Use

How to select entities around poly-faces

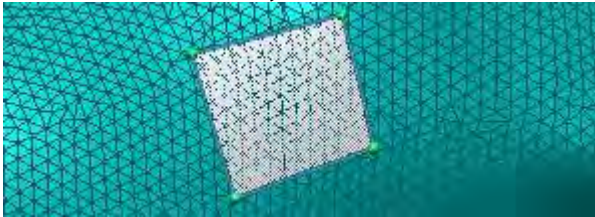
1. Go to **Select > Entities Around Edges/Faces** or click the icon in the toolbar.



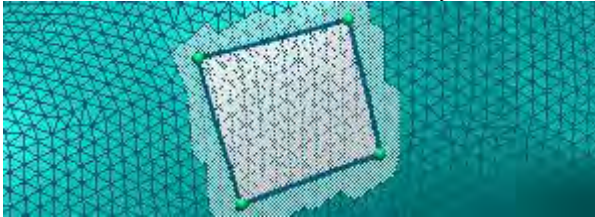
Note:

The Entities Around Edges/Faces command can only be used in Mesh, Region Group, and Point Cloud modes.

2. Select a face on a body for the **Seed Entities**. Change the option to **Poly-Faces**.



3. Click **OK** to finish the command. Poly-faces around the face will be selected.



Options in Detail

Seed Entities – Selects entities (curves, edges, faces) that will be used to find poly-vertices or poly-faces in the model view.

Options – Chooses Poly-Faces or Poly-Vertices are the final selected entity type.

See Also

[Select > All](#)
[Select > None](#)
[Select > Inverse](#)

7.2.8. Select > Enlarge

What

The **Enlarge** command enlarges a selection by a specified depth or angle. Depending on the type of selected entity, the options in the dialog tree will be shown differently. The command will not be available until entities have been selected.

Why

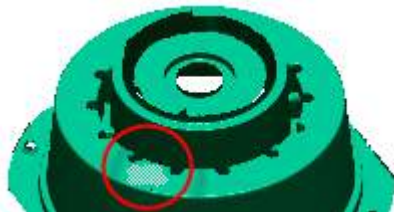
The **Enlarge** command is useful for:

- Selecting areas that would be difficult to select manually

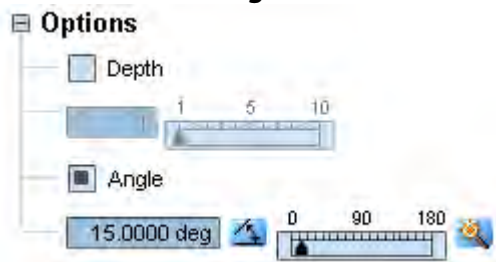
How to Use

How to select entities using the Enlarge command

1. Select faces. Poly-faces, Poly-vertices, Faces, Edges and Regions can be selected for the **Enlarge** command.



2. Go to **Select > Enlarge** or click the icon in the toolbar. Change the **Angle** to **15** degrees.



3. Click **OK** to finish the command. Poly-faces around the face will be selected.

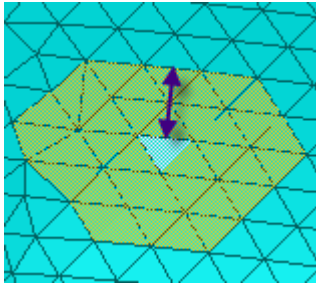


Options in Detail

Type To Select – Chooses entity type to select.

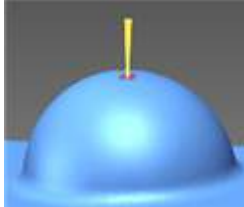
Poly-Faces/Poly-Vertices/Faces/Edges/Region – Shows currently selected entities.

Depth – Expands the selection of entities by a specified depth.

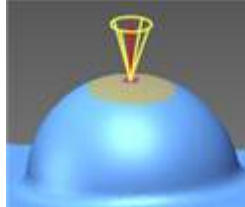


Depth 2

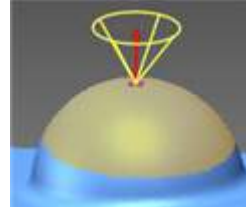
Angle – Enlarges regions by a specified angle for poly-faces and poly-vertices.



Initial Selection



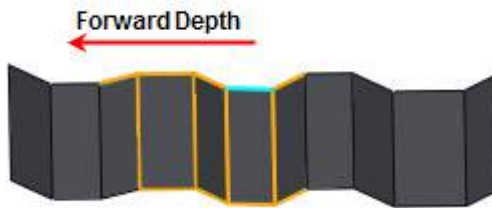
Angle: 25



Angle: 60

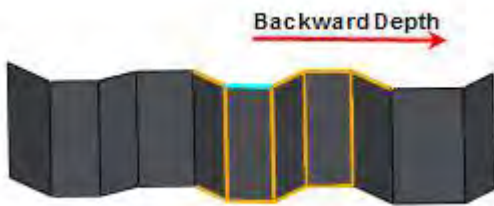
Angle Tolerance – Limits the angle tolerance when comparing other neighboring entities when an edge or face has been selected.

Forward Depth – Selects more edges when an edge or face is selected.



Forward Depth from the Initial Selection

Backward Depth – Selects more edges in the opposite direction from **Forward Depth** when an edge or face is selected.



Forward Depth from the Initial Selection

See Also

- [Select > All](#)
- [Select > None](#)
- [Select > Inverse](#)
- [Select > Shrink](#)

7.2.9. Select > Shrink

What

The **Shrink** command shrinks selected poly-faces by a specified depth. The command will not be available until entities have been selected.

Why

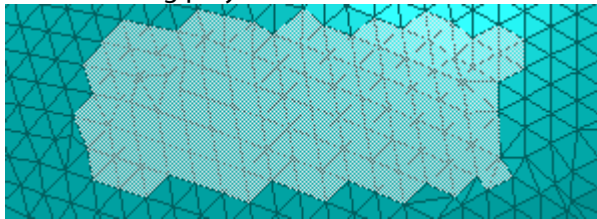
The **Shrink** command is useful for:

- Shrinking a selected area that might otherwise be difficult to do manually

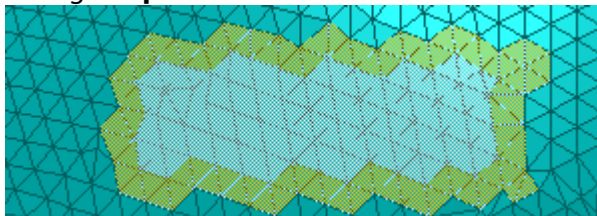
How to Use

How to select shrink entities

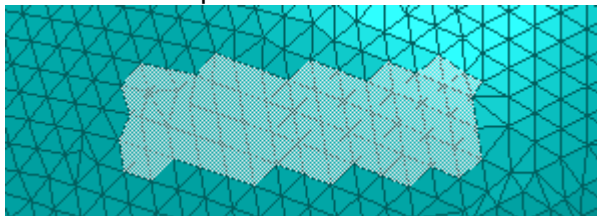
1. Go to **Select > Shrink** or click the icon in the toolbar. The pop-up menu can also be used after selecting poly-faces.



2. Change **Depth** to 1.



3. Click **OK** to complete the command.



Options in Detail

Type To Select – Chooses entity type to select.

Poly-Faces/Poly-Vertices – Shows currently selected entities.

Depth – Shrinks the selection of entities by a specified depth.

See Also

[Select > All](#)

[Select > None](#)

[Select > Inverse](#)

[Select > Enlarge](#)

7.2.10. Select > Convert Current Selection

What

The **Convert Current Selection** command converts entities to another entity type. The command will not be available until entities have been selected.

Why

The **Convert Current Selection** command is useful for:

- Converting one type of entity to another quickly and effectively

How to Use

How to convert entities

1. Select an edge to convert to a vertex.



2. Go to **Select > Convert Current Selection** or click the icon in the toolbar. The pop-up menu can also be used after selecting poly-faces or poly-vertices.

3. Click **OK** to complete the command. The edge will be changed to a vertex.



Options in Detail

From – Shows selected entity type.

To – Shows entity type for conversion.

Conversion List

From	To
Poly-Vertex	Poly-Face
Region	Poly-Face
Mesh Loop(Boundary Loop)	Poly-Face
Modeling Feature	Body Face
Body Face	Body Edge
Body Edge	Body Vertex
2D/3D Curve	2D/3D Node Point

See Also

[Select > All](#)

[Select > None](#)

[Select > Inverse](#)

7.2.11. Select > By Criteria

What

The **Select By Criteria** command selects components of a model, 2D and 3D sketch entities based on criteria such as edge, length, angle, area, or other components.

Why

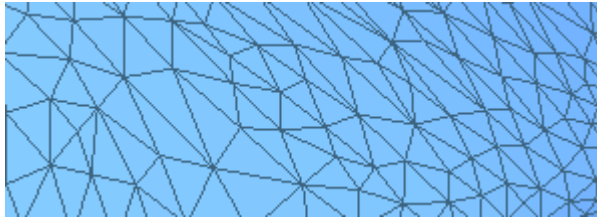
The **Select By Criteria** command is useful for:

- Selecting large amounts of components based on type quickly

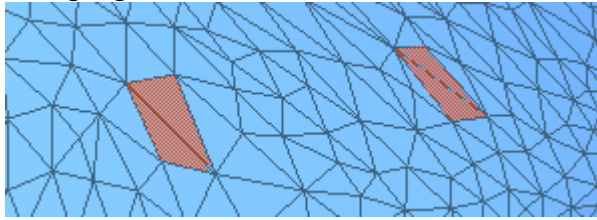
How to Use

A. How to select components by criteria in Mesh or Region Group mode

1. Go to **Select > By Criteria** or click the icon in the toolbar.

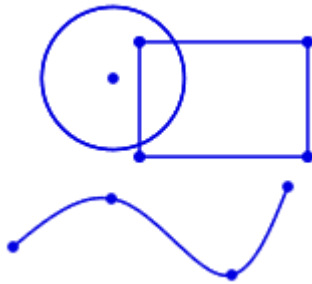



2. Check the **Larger Than** option under the **Max. Edge Length** option to select all faces larger than a specified value. Click the **Estimate** button and the application will recommend a reasonable value.
3. Click **OK** to complete the command. Mesh that is larger than the recommend value will be highlighted.



B. How to select components by criteria in Sketch or Mesh Sketch mode

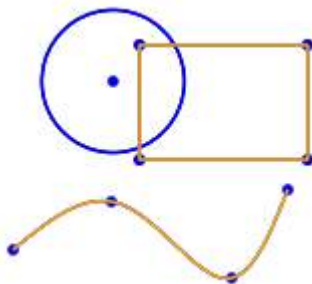
1. Go to **Select > By Criteria** or click the icon in the toolbar.



2. Select the **Entity Type(s)** for selection and **Larger Than** for the **Length** and click the  **Estimate** button. The application will recommend a reasonable value.



3. Click **OK** to complete the command. Sketch entities that are larger than the recommend distance will be highlighted.



Options in Detail

Max. Edge Length – Sets criteria as length.

Smaller Than – Sets length criteria for a maximum length.

Larger Than – Sets length criteria for a minimum length.

Area – Sets criteria as area.

Smaller Than – Sets area criteria for a maximum area.

Larger Than – Sets area criteria for a minimum area.

Max./Min. Edge Length Ratio Is Larger Than – Selects all poly-faces that have a ratio between the length of the longest edge and length of the shortest edge that is larger than a specified value.

Max. Edge Length/Max. Height Ratio Is Larger Than – Selects all poly-faces that have a ratio between the length of the longest edge and height of the longest edge that is larger than a specified value.

Angle Deviation From Plane Is Larger Than – Selects faces that have an angle deviation from a plane that is larger than a specified value and is useful for detecting sharp faces on a smooth surface.

Crease Angle Is Larger Than – Selects faces that have a crease angle larger than a specified angle.

Entity Type – Sets entities as criteria from **Lines, Circles, Arcs, Ellipses, Partial Ellipses, and Parabolas.**

Major Radius – Sets criteria as major radius.

Smaller Than – Sets major radius criteria for a maximum length.

Larger Than – Sets major radius criteria for a minimum length.

Minor Radius – Sets criteria as minor radius.

Smaller Than – Sets minor radius criteria for a maximum length.

Larger Than – Sets minor radius criteria for a minimum length.

See Also

[Select > All](#)

[Select > None](#)

[Select > Inverse](#)

7.2.12. Select > By Color

What

The **Select By Color** command selects poly-faces or poly-vertices by color. The command is enabled when poly-faces or poly-vertices include color or texture information and can be used in **Mesh**, **Region Group**, or **Point Cloud** mode.

Why

The **Select By Color** command is useful for:


- Selecting poly-faces or poly-vertices by color

How to Use


How to select entities by color

1. Enter **Mesh mode**.



2. Go to **Select > By Color** or click the icon in the toolbar.
3. Change the **Type To Select** option to **Poly-Faces**. Click the **Pick Color** button  and extract a color from the mesh. Change the **Fuse** option to **200**.



4. Click the **Add Color**  button to extract similar colors on the mesh.



5. Click **OK** to complete the command.

Options in Detail

Type To Select – Selects a type of entity to be selected.

Color – Selects colors allows the use of the **Pick Color**, **Add Color**, and **Remove Color** buttons to select colors.

Fuse – Selects the level of diffusion when selecting similar colors.

Add – Adds a selection when Poly-Faces or Poly-Vertices are selected before using the **Select By Color** command.

Remove – Removes a selection when Poly-Faces or Poly-Vertices are selected before using the **Select By Color** command.

Replace – Replaces a current selection when Poly-Faces or Poly-Vertices are selected before using the **Select By Color** command.

See Also

[Select > All](#)

[Select > None](#)

[Select > Inverse](#)

7.2.13. Select > Exterior Poly-Face

What

The **Select Exterior Poly-Face** command selects exterior regions of poly-faces on mesh data. It can be used on volume reconstructed mesh or mesh with multiple layers. It can be used in **Mesh** or **Region Group** modes with mesh data.

Why

The **Select Exterior Poly-Face** command is useful for:

- Separating exterior poly-faces from interior poly-faces

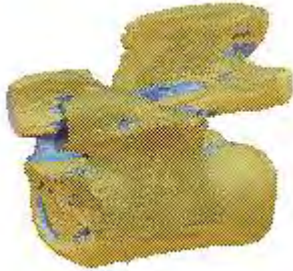
How to Use

How to select exterior poly-faces

1. Go to **Select > Exterior Poly-Face** or click the icon in the toolbar. Mesh that has poly-faces inside and outside such as a model captured by a CT scanner can be used.



2. Set **Initial Radius** to **10**, **Expand Radius** to **5**, and **Tolerance** to **0.3**.



3. Click **OK** to complete the command.
4. Copy the selected outside surface by going to **Edit > Copy**.
5. Exit **Mesh mode** and go to **Edit > Paste** to paste the exterior poly-face mesh.



Options in Detail

Initial Radius – Changes the initial radius value used to detect exterior poly-faces.

Expand Radius – Changes the expanded radius value from a seed region, which is an initially selected point or face.

Tolerance – Changes the interruption areas caused by noise on mesh.

See Also

[Select > All](#)

[Select > None](#)

[Select > Inverse](#)

7.3. Edit

The **Edit** menu features commands to handle entities and operation.

Repeating Operation

- **Undo & Redo**

Whenever you want to cancel the applied operation, use the Undo command. If you want to restore the operation from Undo, use the Redo command.

- **Repeat Last Command**

Quickly apply the operation that you used previously.

Editing Entities

- **Cut, Copy, Paste and Delete**

The Cut, Copy, Paste and Delete operations are available to the selected entities.

Navigating Modeling History

- **Rebuild**

Rebuilds a model with the current version of the application based on the model history.

- **Rollback, Roll Forward, Roll To End**

Navigates modeling history to return to a certain point of modeling feature.

Showing or Hiding Features in History

- **Suppress and Unsuppress**

Temporarily removing features from the model.

Editing Feature

- **Feature**

Edits a selected feature.

7.3.1. Edit > Undo

What

The **Undo** command undoes previous operations.

Why

The **Undo** command is useful for:

- Reversing mistakes made
- Going back to a previous point

How to Use

How to undo an action

1. Go to **Edit > Undo** or use the **Ctrl + Z** keyboard shortcut.

Note:

The depth of the **Undo** command can be changed by going to **File > Preferences**. The default depth is 50.

See Also

[Edit > Redo](#)

7.3.2. Edit > Redo

What

The **Redo** command reverses the **Undo** command or advances operations to a more current state step-by-step. This command is opposite of undo command.

Why

The **Redo** command is useful for:

- Advancing operations to a more current state
- Reversing the **Undo** command

How to Use

How to undo an action

1. Go to **Edit > Redo** or use the **Ctrl + Y** keyboard shortcut.

See Also

[Edit > Undo](#)

7.3.3. Edit > Repeat Last Command

What

The **Repeat Last Command** repeats the last used command.

Why

The **Repeat Last Command** command is useful for:

- Repeating the last command made if it is going to be used often in a row

How to Use

How to repeat the last command

1. Go to **Edit > Repeat Last Command** or use the **Ctrl + Space** keyboard shortcut.

See Also

[Edit > Undo](#)

[Edit > Redo](#)

7.3.4. Edit > Cut

What

The **Cut** command deletes an entity and copies it, saving it to the clipboard to be pasted.

Why

The **Cut** command is useful for:

- Removing an entity to use for another time or place

How to Use

How to cut an entity

1. Select an entity to cut.
2. Go to **Edit > Cut** or use the **Ctrl + X** keyboard shortcut.

See Also

[Edit > Paste](#)

[Edit > Copy](#)

7.3.5. Edit > Copy

What

The **Copy** command copies a selected entity, saving it to the clipboard to be pasted.

Why

The **Copy** command is useful for:

- Creating a copy of an existing entity to be used at a later time or place

How to Use

How to copy an entity

1. Select an entity to copy.

Note:

Mesh, Point Cloud, 2D sketches, 3D sketches and Ref. Geometries can be copied in the default mode.

2. Go to **Edit > Copy** or use the **Ctrl + C** keyboard shortcut.

See Also

[Edit > Cut](#)

[Edit > Paste](#)

7.3.6. Edit > Paste

What

The **Paste** command copies a selected entity, saving it to the clipboard to be pasted.

Why

The **Paste** command is useful for:

- Pasting a cut or copied entity

How to Use

How to paste an entity

1. Select an entity and use the **Cut** or **Copy** command.
2. Go to **Edit > Paste** or use the **Ctrl + V** keyboard shortcut.

See Also

[Edit > Cut](#)

[Edit > Copy](#)

7.3.7. Edit > Delete

What

The **Delete** command deletes selected entities.

Why

The **Delete** command is useful for:

- Removing unnecessary entity

How to Use

How to delete an entity

1. Select an entity.
2. Go to **Edit > Delete** or use the **Delete** key on the keyboard.

See Also

[Edit > Undo](#)

[Edit > Redo](#)

7.3.8. Edit > Rebuild

What

The **Rebuild** command rebuilds a model with the current version of the application based on the model history.

Why

The **Rebuild** command is useful for:

- Updating and rebuilding a model using any changed algorithms or commands in the application

How to Use

How to rebuild a model

1. Go to **Edit > Rebuild** or use the icon in the toolbar.

7.3.9. Edit > Rollback

What

The **Rollback** command temporarily reverts a model to an earlier state. New features can be added or existing features can be edited.

Why

The **Rollback** command is useful for:

- Going back to previous step in the modeling process

How to Use

How to rollback a model

1. Select an entity in the feature tree.
2. Go to **Edit > Rollback** or use the icon in the toolbar. All entities after the rollback entity will be dimmed in the model tree.

See Also

[Edit > Roll Forward](#)

[Edit > Roll To End](#)

7.3.10. Edit > Roll Forward

What

The **Roll Forward** command rolls a model forward step-by-step after executing the **Rollback** command. New features can be added or existing features can be edited.

Why

The **Roll Forward** command is useful for:

- Going forward step-by-step after using the **Rollback** command to see changes

How to Use

How to Roll Forward a model

1. After executing the **Rollback** command, select an entity in the feature tree.
2. Go to **Edit > Roll Forward** or use the icon in the toolbar.

See Also

[Edit > Rollback](#)

[Edit > Roll To End](#)

7.3.11. Edit > Roll To End

What

The **Roll To End** command rolls a model to the end of the modeling history after executing the **Rollback** command.

Why

The **Roll To End** command is useful for:

- Going to the end of the modeling history after using the **Rollback** command

How to Use

How to roll a model to the end

1. After executing the **Rollback** command, go to **Edit > Roll To End** or use the icon in the toolbar.

See Also

[Edit > Rollback](#)

[Edit > Roll Forward](#)

7.3.12. Edit > Suppress

What

The **Suppress** command suppresses a feature, temporarily removing it from the model.

Why

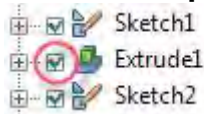
The **Suppress** command is useful for:

- Seeing and working with a model without a specific feature

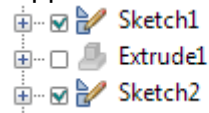
How to Use

How to suppress a feature

1. Select an entity in the feature tree.
2. Go to **Edit > Suppress**, or click the icon in the toolbar.



3. Suppressed entities will be dimmed in the **Model Tree**.



See Also

[Edit > Unsuppress](#)

7.3.13. Edit > Unsuppress

What

The **Unsuppress** command unsuppresses a feature after using the **Suppress** command.

Why

The **Unsuppress** command is useful for:

- Unsuppressing a feature

How to Use

How to unsuppress a feature

1. Select a suppressed entity in the feature tree.
2. Go to **Edit > Unsuppress**, or click the icon in the toolbar.

See Also

[Edit > Suppress](#)

7.3.14. Edit > Feature

What

The **Feature** command allows the editing of a selected feature.

Why

The **Feature** command is useful for:

- Selecting and editing a feature

How to Use

How select a feature

1. Select an entity in the feature tree.
2. Go to **Edit > Feature**, or click the icon in the toolbar. The entity will now be editable.

7.4. Insert

The **Insert** menu features various commands used to generate or import entities and register them

as features with a generation history in the **Feature Tree**.

The **Insert** menu provides tools used to:

Import

- **Import Scan Data or CAD Data**

Imports entities that are also treated as features in the application with an importing and editing order that reflects the compositions in the modeling history

Mesh

- **Edit meshes**

Fixes errors, optimizes surface quality, and re-constructs the structure of mesh which is normally polygonal data from scanners or other software

Region Group

- **Classify Regions**

Generates, edits and optimizes region groups that are groups or regions made up of poly-faces that assist in the alignment of mesh data to the global coordinate system and in the selection of a region of interest used for extracting design intent and converting to a surface

Point Cloud

- **Edit Point Clouds**

Enables editing and optimizing of point clouds which are a set of vertices in the three-dimensional coordinate system. These vertices are usually defined by the X, Y, and Z coordinates

Solid

- **Generate Solid Bodies**

Generates and edits solid bodies which are A B-rep entities that have both geometry and topology information based on a mesh or point cloud

Surface

- **Generate Surface Bodies**

Generates and edit solid bodies which are zero-thickness 3D entities with edge boundaries based on a mesh or point cloud

Mesh Sketch

- **Generate Sketches Based on Scan Data**

Generates and edits sectional profiles that consist of a set of 2D objects (lines, rectangles, circles,

arcs, ellipses, splines, etc) on a plane, a face, or a planar cross-section, based on a mesh or point cloud

Sketch

- **Generate Sketches**

Generates and edits sketch entities that consist of a set of 2D objects (lines, rectangles, circles, arcs, ellipses, splines, etc) on a planar entity such as those created in other CAD software

3D Mesh Sketch

- **Generate Splines Based on Scan Data**

Generates a curve network which is a set of 3D curves and lines on mesh faces used for boundary fitting surfaces or extracting character curves on a mesh

3D Sketch

- **Generate Splines**

Freely draws splines which are a set of 3D curves and lines in space regardless of a mesh

7.4.1. Insert > Import

What

The **Insert > Import** command imports scan data or CAD data. Imported data is added and managed under the **Feature Tree**.

Supported file formats are shown in the following table:

Extension	Application	Import	Export	Color	Texture	Property
xdl, xrl, rwl	XO Model	Yes	Yes	Yes		Point, Polygon, Curve, Surface, Solid
xpc	Rapidform Extreme Point Cloud	Yes	Yes	Yes		Point
mdl	Rapidform2006	Yes	Yes	Yes	Yes	Point, Polygon, Curve, Surface, Solid
asc, xyz, txt	Ascii Points	Yes	Yes	Yes		Point (ASCII), xyz or xyzrgb
pts	Geomagic Point	Yes		Yes		Point file (ASCII)
fcs	Geomagic Polygon	Yes	Yes	Yes		Polygon file (ASCII)
stl	Stereolithography	Yes	Yes			Polygon (ASCII or Binary)

obj	OBJ	Yes	Yes	Yes	Yes	Point (Import only), Polygon
ply	CyberWare	Yes	Yes	Yes		Polygon (ASCII or Binary)
3ds	3D Studio	Yes	Yes		Yes	Point or Polygon
wrl	VRML 1.0/97	Yes	Yes	Yes	Yes	Point or Polygon
icf	INUS Compression File Format	Yes	Yes	Yes	Yes	Polygon file
igs, iges	IGES	Yes		Yes		Point, Curve, Surface
stp, step	STEP	Yes		Yes		Point, Curve, Surface
x_t, x_b	Parasolid Text/Binary	Yes		Yes		Solid or Surface
sat, sab, asat, asat	ACIS Text/Binary Part/Assembly	Yes				Solid, Surface, Curve, Point
model	CATIA V4	Yes	Yes			Solid, Surface, Curve, Point
catpart, catproduct	CATIA V5 Part/Assembly	Yes	Yes			Solid, Surface, Curve, Point
prt, prt.*, asm, asm.*	Creo(Pro/Engineer) Part/Assembly	Yes				Solid, Surface, Curve, Point
prt	Siemens NX	Yes				Solid, Surface, Curve, Point
sldprt, sldasm	SolidWorks	Yes				Solid, Surface, Curve, Point
dxf	AutoCAD DXF	Yes	Yes			Point, Polygon, Curve
scn	NextEngine	Yes			Yes	Polygon, Curve, Surface
ac	Steinbichler	Yes		Yes		Point or Polygon
Bre, ctr	Breuckmann	Yes		Yes		Point
vvd, cdm, cdk	VIVID	Yes			Yes	Polygon
rgv, rvm	Range7	Yes				Point, Polygon
pix	PICZA (Roland)	Yes				Point, Polygon
g3d	GOM	Yes		Yes		Point or Polygon
cloud		Yes		Yes		Point
surf		Yes		Yes		Point
pmj, pmjx	3D Digital Corp.	Yes				Point
pts, ptg, ptx	Leica	Yes		Yes		Point
fls	FARO	Yes		Yes		Point
fws		Yes		Yes		The information of corresponding fls files
iqscan	iQvolution	Yes		Yes		Point
3dd, rxp	Riegl	Yes		Yes		Point or Polygon
xyz, crs, lin, smh, bin	Opton	Yes				Point
soi	MENSI	Yes		Yes		Point
btx	Surphaser	Yes		Yes		Point, Polygon
stb	Scantech	Yes				Point
3pi	ShapeGrabber	Yes				Point
sab, sab2	Nikon Metrology	Yes		Yes		Point or Polygon

swl, bin	swb,	Perceptron	Yes				Point
rtpi, xyzi, xyzrgb		DeltaSphere	Yes		Yes		Point
psl, pcn	scn,	LDI	Yes				Point
cbk, cwk	grk,	Kreon	Yes				Point
icv		Solutionix	Yes		Yes		Polygon
snx			Yes				Polygon
tfm		Wicks and Wilsons	Yes		Yes		Point
m3d		Steintek	Yes				Point, Polygon
hym		Hymarc	Yes				Point
xyz, xyzn		Cognitense	Yes				Point, Polygon
opd		Optimet	Yes		Yes		Point, Polygon
mpc		Mantis Vision	Yes		Yes		Point, Polygon
dat, nas		NASTRAN	Yes				Polygon
ans		ANSYS	Yes				Polygon
iv		Inventor	Yes		Yes	Yes	Point, Polygon
vda		VDA-FS	Yes				Curve, Surface
3dm		OpenNURBS(Rhino)	Yes				Curve, Surface
cls		Topcon	Yes		Yes		Point
ptc		Kubit	Yes		Yes		Point
lsf		LiveScan	Yes				Point
i3d		Imetric	Yes			Yes	Point, Polygon
rge		Shining 3D	Yes				Point, Polygon
csd		Fujifilm	Yes			Yes	Point, Polygon

Color means colors on Poly-Vertex/Face and Texture means an image file such as .jpg.

Note:

The application supports standard file formats (IGES, STEP, VDA-FS, ACIS and Parasolid) without any licenses, but for the file formats listed in the table below, the Geomagic Exchange™ License is needed to use them.

CAD	File Extension	Version Supported	Comments
CATIA V4	.model	CATIA 4.1.9 to CATIA 4.2.4	- Geometry Only
CATIA V5	.CATPart, .CATProduct	CATIA V5 R6 through R22 (V5-6 R2012)	- Geometry Only - Unsupported non-ASCII character in the file name
Creo(PRO/E)	.prt, .prt.*, .asm, .asm.*	Pro/E 16 to Wildfire 5, Creo 1.0 to 2.0	- Geometry Only
Siemens NX	.prt	Unigraphics 11 to 18 and NX1 to NX8	- Geometry Only
SolidWorks	.sldprt, .sldasm	SolidWorks 98 to 2013	- Geometry Only

Note:

The .icv and .snx Solutionix file formats are supported in the 32-bit version of Geomagic Design X only.

Note:

The following point cloud formats can be converted into polygons with its grid information or 2D triangulation processes during the importing process: Ac, Bre, cloud, surf, opton, 3pi, stb, pmj/pmjx, grk, tfm, m3d.

Note:

It is possible to import multiple files at once. To do so, hold down **Ctrl** or **Shift** key while selecting files in the importing dialog box.

Note:

The following formats can be imported without loss of color information:
WRL, OBJ, PLY, Geomagic (fcs, icf, mdl, xdl).

Note:

If the Import dialog box is opened to import files, all of the supported files may not be listed under "File Format" because of the limitations of the OS. Favorite file formats can be shown by choosing **True** next to the file format in the **File I/O** tab in **File > Preferences** and only these formats will be shown in the Import dialog box.

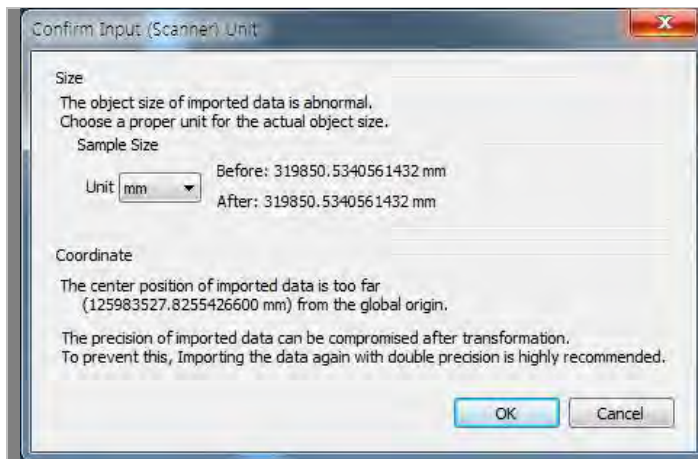
How to Use

A. How to import scan data

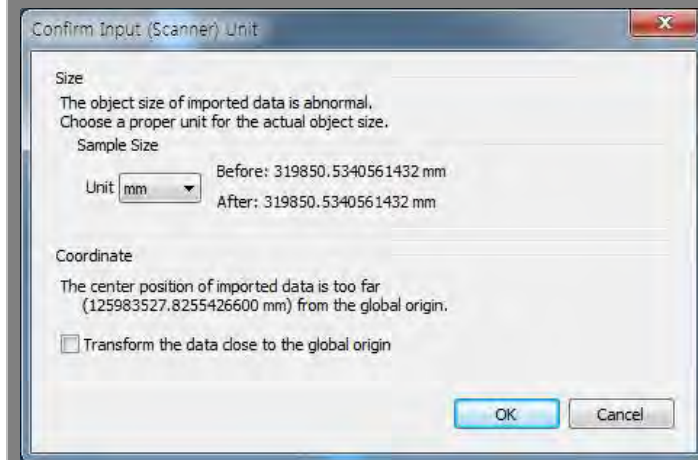
1. Go to **Insert > Import**.
2. Select the files to import by click-dragging the mouse cursor over them, or by using the **Shift** or **Ctrl** key and selecting them one by one.
3. Click **Import Only** to import the files without executing any operations or use the **Run Mesh Buildup Wizard** to generate a single mesh from multiple scan shots. The **Mesh Buildup Wizard** can be executed any time after importing scan data (**Tools > Scan Tools > Mesh Buildup Wizard**).

Note:

If an imported scan data part is very small or large, it can be problematic to generate features based on the mesh. If this type of data is imported, the application will show the 'Confirm Input (Scanner) Unit' dialog box to verify the input unit of the scan data. Normally long range scan data files are very large and entities in the file can span far away from the global origin. In this case it is recommended to import the scan data with the **Double Precision** option to preserve the accuracy of the scan data.

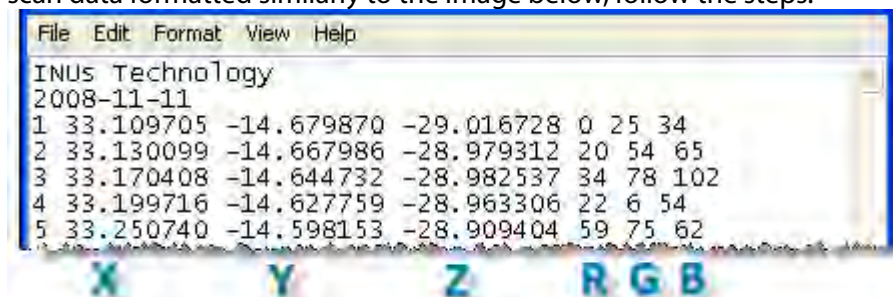


If importing a large scan data with Double Precision, the **Transform the data close to the global origin** option will appear. This option aligns the scan data to the global origin while importing.

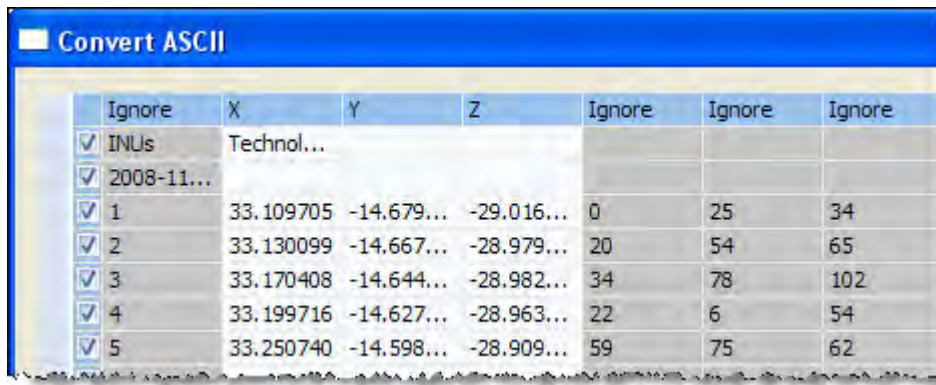


B. How to use the ASCII Converter to import an unknown scan format (text file)

1. Go to **Insert > Import** or click the icon in the toolbar and select a text file. To import scan data formatted similarly to the image below, follow the steps.



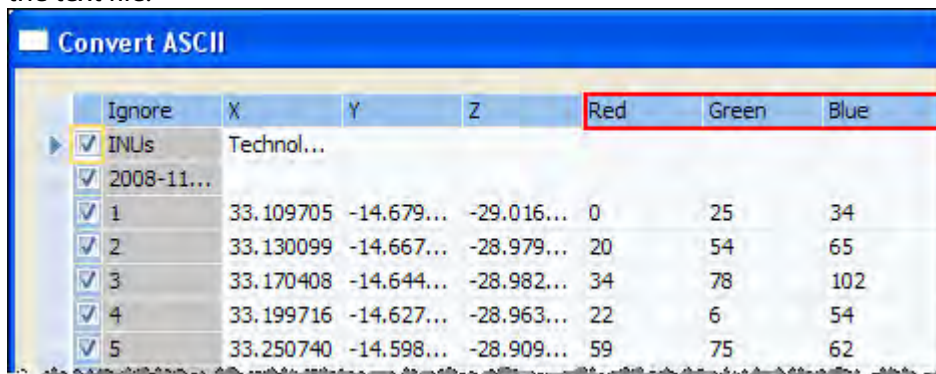
2. Check the **Use ASCII Convert** option.
3. Click **Import Only**.
4. The **Convert ASCII** dialog box will appear and columns and rows can be selected to be imported.



- Define the separator in the **Separator** section. Since different files may use different separators to separate data, a suitable separator in the **Separator** section may need to be defined. If a space or a tab is used as the separator, the **Whitespace (space, tab)** box will need to be checked. Otherwise, check the **Other Character** box and input a suitable character. If **,"** is used for a decimal point, check the **Comma (,) For Decimal Point** option to distinguish the separator and the decimal point.

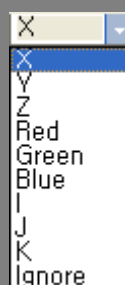


- Change the **Ignore** columns to **Red, Green and Blue**, if color information is included in the text file.



Note:

When the title of the column is clicked, a pull-down menu appears. **X, Y, Z** characters define coordinate information, **R, G, B** characters define color information and **I, J, K** characters define normal information on each point. **Ignore** ignores the content in a selected column.

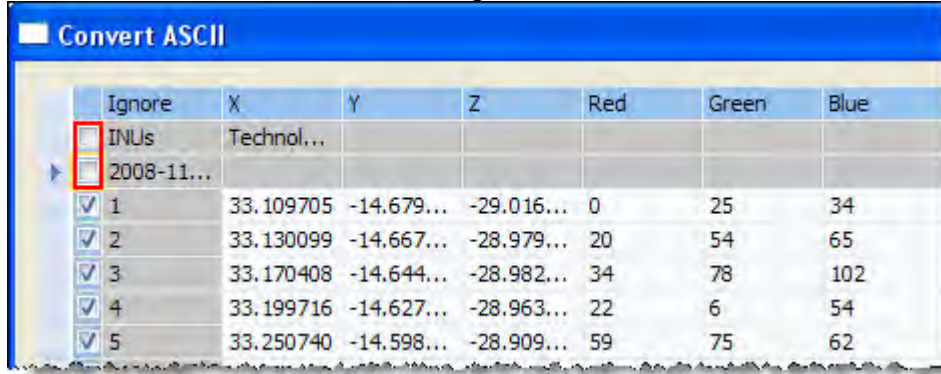


Normally a file contains X, Y, Z or X, Y, Z, R, G, B information. If only one of the R, G, B color attributes has a value, it will not be the color information, but the intensity value. However,

if more than 2 of the R, G, B color attributes have values, they will be the color information.

Since different files may use different values to represent color information, a suitable Multiplier value may be needed. The range of colors is 0-255. For example, if the R, G, B values in a file are between 0 and 255, the Multiplier value will simply be ignored. But if the R, G, B values on a file are between 0 and 1, 255 will be needed for the Multiplier value.

7. Uncheck the first and second rows to ignore them.



8. Click **OK**. The scan data will be imported into the application.

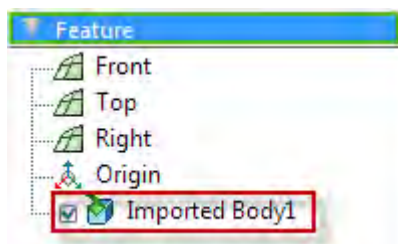


Note:

To define columns and save them, use the **Edit Template** button. A pre-defined template can be loaded again without editing columns.

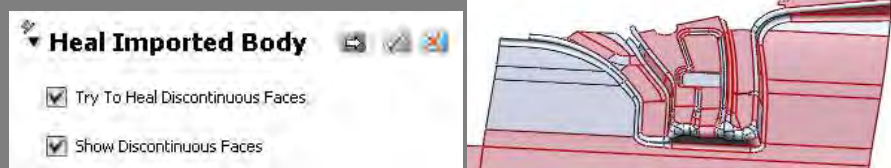
C. How to import CAD data

1. Go to **Insert > Import** or click the icon in the toolbar.
2. Select a CAD file.
3. Click **Import Only**. After the importing process, the CAD data and the feature named **Imported Body1** in the **Feature** tree can be viewed.



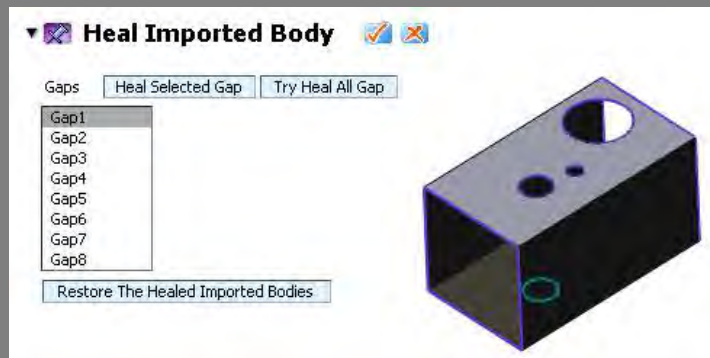
Note:

If an **IGES, VDA, STEP** or **3DM(OpenNurbs)** file containing holes and erroneous faces is imported, the application will ask “Do you want to run healing imported body process?” click “Yes” and the healing process will be performed with the **Heal Imported Body** dialog tree.



Try To Heal Discontinuous Faces – Attempts to heal faces that are internally (C1) discontinuous. If these faces are not healed, they cannot be extended, lofted, or swept.

Show Discontinuous Faces – Highlights discontinuous faces in red



Heal Selected Gap – Fills selected gaps in the list box

Try Heal All Gap – Attempt to heal all gaps.

Restore The Healed Imported Bodies – Resets all changes

Options in Detail



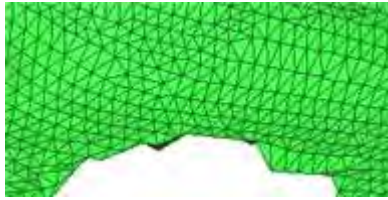
Fix Erroneous Data – Checks non-manifold faces, coincident vertices and degenerated faces and fixes them if needed.

Use ASCII Convert – Used for unknown text formats. The **Convert ASCII** dialog box will appear and columns, rows and separators can be defined.

Import Hidden Bodies – Imports bodies which state is hidden from other CAD software.

Use Custom Setting – Specifies sub-options that will be preserved when importing scan data again. If this option is not checked, the settings of sub-options will be changed to a selected file type.

Points Only – Imports point clouds without poly-faces.

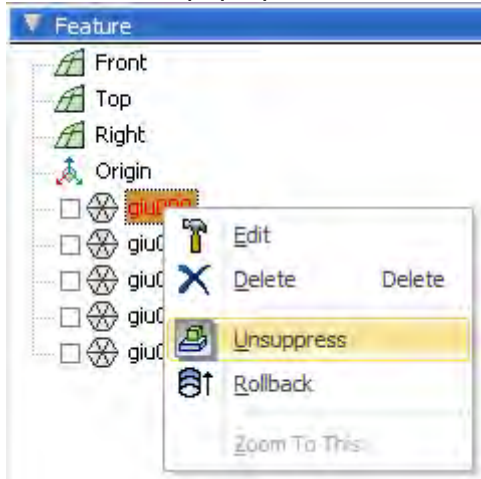


Mesh Data



Points Only

Suppress Mesh – Suppresses scan data during importing which loads meshes onto the hard disk helping to deal with large files efficiently. The Suppress/Unsuppress mechanism manages data and memory by swapping the data dynamically between the physical memory and the hard disk. The swapping time for Load / Unload is very short, so the calculation overhead is negligible. After importing the meshes the **Unsuppress** (load on the memory) command can be used in the pop-up menu in the **Feature** tree.



Unit – Defines a scan data file measurement unit.

Include Vertex Normal – Preserves normal information.

Combine Point Clouds – Combines multiple point clouds into a single point cloud.

Precision – Indicates how many decimal places a position in space will carry with it throughout calculations. The float precision is the default setting and it is appropriate for most cases, but double precision can be specified for large scale data or for higher accuracy.

Include Texture – Ignores texture information when **None** is selected. When **As It Is** is selected, texture information will be preserved in its original status (an image or color points). When **As Color Point** is selected, texture information from an image will be converted to color points.



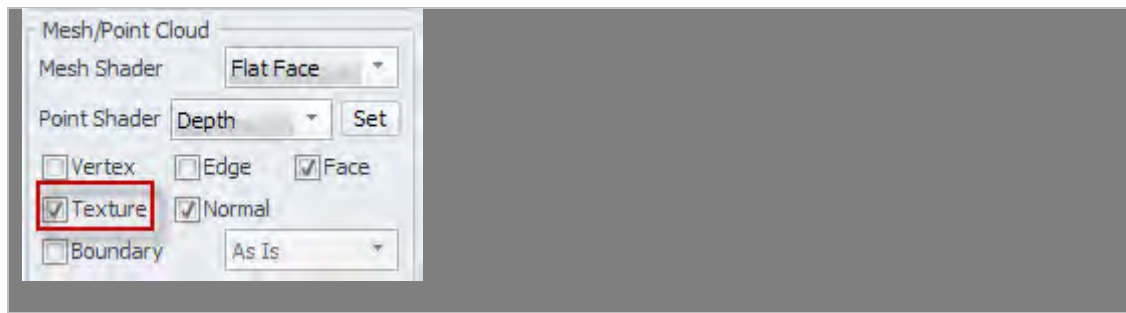
Include Texture



Exclude Texture

Note:

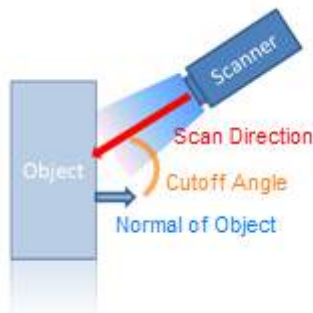
After importing a texture, it can be shown or hidden in the **Display** panel.



Sampling Ratio – Samples data points by a specified value. When the ratio is 100%, all selected data will be used. When the ratio is 50%, half of the selected data will be used.

Triangulation Cutoff Angle – Filters noisy vertices that have a normal angle compared to neighboring vertices whose angles are larger than a specified value during triangulation. Normally side walls cannot clearly be scanned from a scanning direction. This data can be defined as noise.

This option is available when selecting a scan file format that includes scan direction information.



See Also

[File > Preferences](#)

[Tools > Scan Tools > Mesh Buildup Wizard](#)

[Docking Bars > Display](#)

7.4.2. Insert > Mesh

Going to **Insert > Mesh** from the menu and clicking the **Mesh** mode icon in the tool palette produce different results. Going to **Insert > Mesh** enables only the addition of mesh features in the **Feature tree**, however, entering the mesh mode with the icon enables full editing of a mesh entity.

Insert Mesh

Using **Insert > Mesh** enables the addition of mesh feature in the **Feature tree**.

A. How to add a mesh by converting CAD data

1. Go to **Insert > Mesh**.
2. Go to **Tools > Mesh Tools > Convert Body**.
3. Select a solid or surface body.
4. Set the slider of Quality to define resolution.
5. Click **OK** to complete the command.

B. How to add a mesh by copying a part of a mesh

1. Go to **Insert > Mesh**.
2. Show other meshes.
3. Select poly-faces from other meshes.
4. Go to **Edit > Copy** or use the **Ctrl + C** shortcut key.
5. A message box will appear to confirm the copied entities. Click **OK** to complete the command.



6. Go to **Edit > Paste** or use the **Ctrl + V** shortcut key.

Note:

Copying a whole mesh using **Insert > Mesh** is not necessary. In the default mode, select a mesh and copy and paste it.

Mesh Mode



The **Mesh** mode features functions to edit, heal, enhance and optimize mesh. To enter the **Mesh mode**, select a mesh and then click the **Mesh** icon or double-click the mesh in the Feature or Model tree or in the Model View.

Mesh functions for a whole mesh or partial mesh can be performed by selecting partial areas on the mesh.

Navigation Palette

The navigation palette in the **Mesh** mode finds and navigates through holes, regions and erroneous poly-faces. If the **Find Defect** command is executed and the application finds errors, the errors are listed in the pull-down menu. Errors on the mesh can be cycled through by clicking the << or >> buttons.

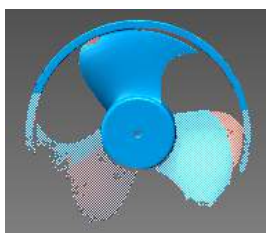


The middle button changes depending on the function. For example with the **Fill Hole** command, the middle button changes to **OK**, but with the **Find Defect** command, the middle button changes to **DEL**.

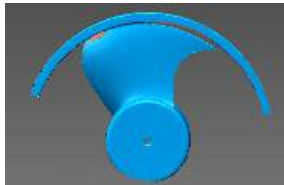
How to Use

How to delete poly-faces

1. Select a mesh and click the **Mesh Mode** button on the toolbar, or double click the mesh.
2. The toolbar will show functions related to mesh tools.
3. Select poly-faces on the mesh.



4. Go to **Edit > Delete** or press the **Delete** key on the keyboard to remove them.



5. Click the **Apply** button at bottom right corner of the **Model View** or click the mesh mode icon **again** to apply all changes and exit the **Mesh** mode.

Note:



Clicking the cancel button will exit the mesh mode and cancel all changes made in the mode.

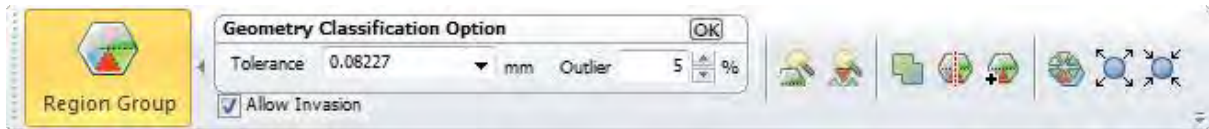
See Also

[Tools > Mesh Tools > Convert Body](#)

[Edit > Copy](#)

[Edit > Paste](#)

7.4.3. Insert > Region Group

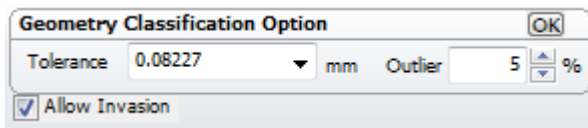


Region Group classifies areas of mesh based on geometry features. The **Region Group** mode automatically or manually classify and edit regions on a mesh with the following characteristics:

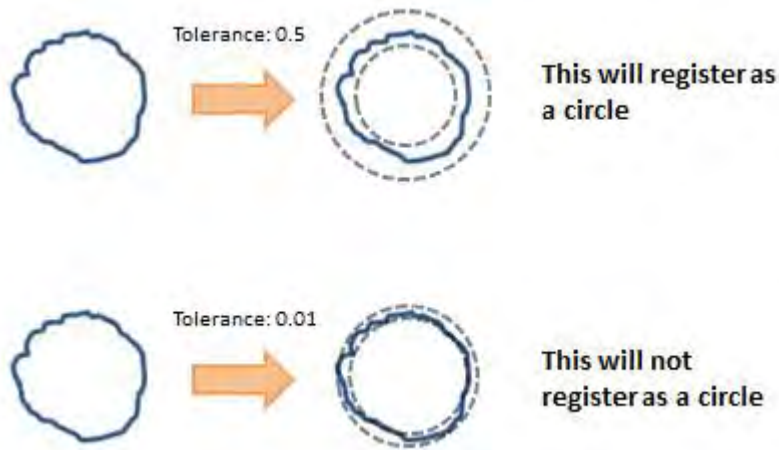
- Generating and editing region history will not register in the feature tree.
- The **Region Group** mode icon will be dimmed if more than one mesh exists until one of them is selected.
- The Region Group mode can be exited by clicking the Region Group icon or the Apply button at bottom right corner.
- Clicking the Cancel button at the bottom right corner removes all changes in the region group mode.

Geometry Classification Option

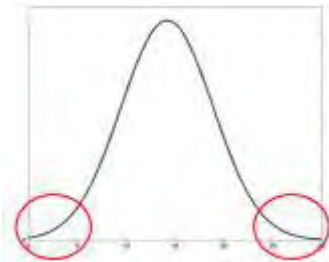
This option classifies geometry types such as plane, cylinder, and cones as a specified input condition. For example, to classify regions of very small object it is necessary to judge geometry types due to the small geometry of the object.



Tolerance – Judges an analytic shape. When a large value is set, a shape that is not a circle can be registered as one.



Outlier – Data that is far from the standard deviation which may include unreliable information that can deform results from ideal geometry.



Outlier in Statistic



Outlier in Cylinder Geometry

Allow Invasion – Allows expanding regions to invade other regions. If this option is unchecked, regions cannot be overflowed into other regions.

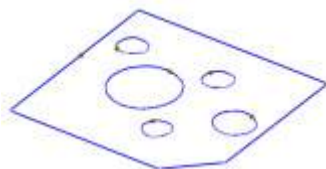
7.4.4. Insert > Point Cloud

Insert Point Cloud

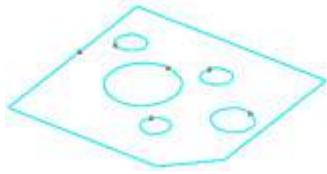
The **Insert > Point Cloud** command inserts point clouds by converting other entities into point clouds or copying poly-vertices from existing mesh.

A. How to add a point cloud by converting a curve

1. Prepare a curve and go to **Insert > Point Cloud** or click the icon in the toolbar.



2. Go to **Tools > Point Cloud Tools > Convert Entities**.
3. Select a curve.



4. Set the slider of **Quality** to define the resolution.
5. Click **OK** to complete the command.



B. How to add a point cloud from other meshes

1. Go to **Insert > Point Cloud** or click the icon in the toolbar.
2. Show other meshes.
3. Select poly-vertices from other meshes.
4. Go to **Edit > Copy** or use the **Ctrl + C** shortcut key.
5. A message box will appear to confirm the copied entities. Click **OK**.



6. Go to **Edit > Paste** or use the **Ctrl + V** shortcut key.

Point Cloud Mode



Point Cloud mode features functions to add and optimize point clouds and create meshes. To enter the Point Cloud mode, select a point cloud and then click the Point Cloud icon or double-click the

point cloud in the Model tree or in the Model View.

Point cloud functions can be performed for a whole point cloud or only for a part of the point cloud by selecting partial poly-vertices. Point Cloud mode has the following characteristics:

- All functions are available on the toolbar and in the **Tools > Point Cloud Tools** menu.
- Generating and editing point cloud will not create any history or features in the feature tree.
- Allows functions to be applied to any existing point clouds.
- Clicking the Point Cloud icon or Apply button at bottom right corner will exit the mode.
- Clicking the Cancel button at the bottom right corner removes all changes in the **Point Cloud** mode
- Commands under **Tools > Scan Tools** can directly apply to point clouds without entering the **Point Cloud** mode.

See Also

[Tools > Point Cloud Tools > Convert Entities](#)

[Edit > Copy](#)

[Edit > Paste](#)

7.4.5. Insert > Ref. Geometry

The **Ref. Geometry** menu features various commands used to generate primitive geometry and edit other entities. The description of reference geometries are for normal situations. In some situations the geometries can be used for different purposes.

Ref. Geometries

- **Plane**
Used for a sketch plane and a cutting plane
- **Vector**
Used for an axis of a feature
- **Coordinate**
Used for local coordinate of an object
- **Polyline**
Used for a reference to extract design indent from a mesh
- **Point**
Marks a position

Note:

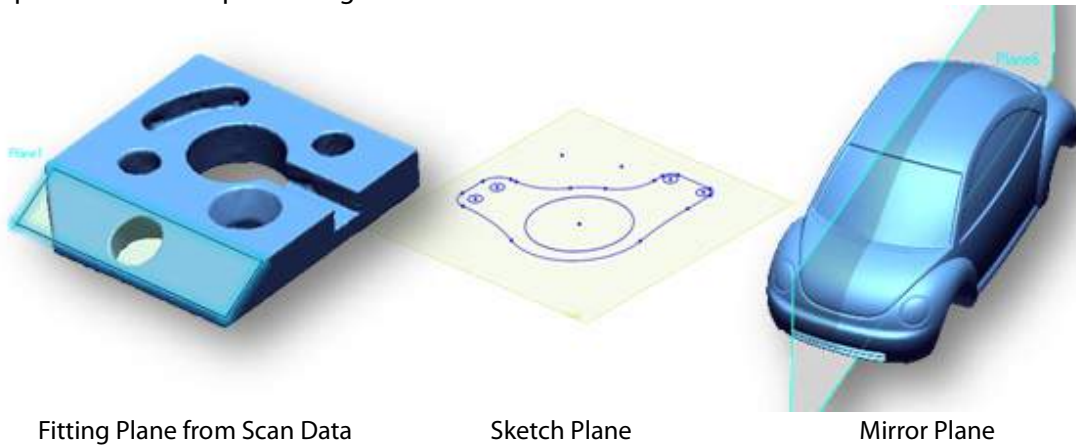
After generating ref. geometries, they can be exported by using the **File > Export** command or the pop-up menu on selected ref. geometry in **Model Tree**.



Insert > Ref. Geometry > Plane

What

A reference plane is a virtual plane that has a normal direction but an infinite size. The ref. plane is not a surface body. They are used to generate other features. The following images show some examples of reference plane usage.



Fitting Plane from Scan Data

Sketch Plane

Mirror Plane

Why

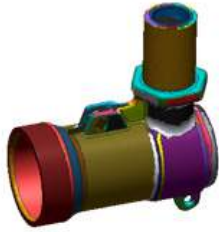
The **Ref. Plane** command is useful for:

- Setting a sketch plane
- Extracting a plane by fitting scan data's planar area
- Cutting object to trim meshes or bodies
- Generating mirrored features
- Finding the middle plane of a symmetry mesh

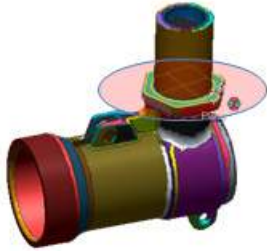
How to Use

A. How to extract a sketch plane from a mesh

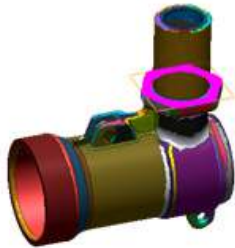
1. Prepare a mesh. A mesh with region groups is ideal.



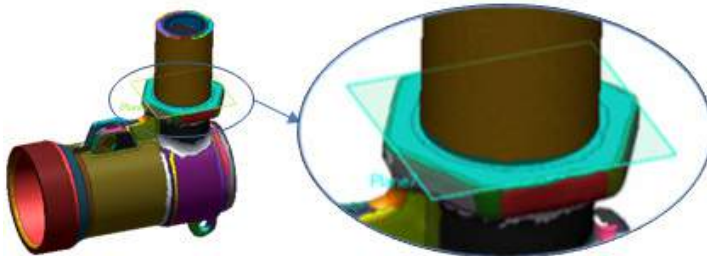
2. Go to **Insert > Ref. Geometry > Plane** or click the appropriate icon in the toolbar.
3. Change the method to **Extract**.
4. Select poly-vertices or a region. The top face of hexagonal object is selected in the following image.



5. When a region is selected the preview plane is shown with highlighted poly-vertices which are used for fitting.



6. Use other options or click **OK** to complete the command.



Note:

The Extract method is available not only for mesh but also for point clouds, bodies, faces, edges, sketch entities, curves, boundaries, etc.

B. How to offset a plane

1. Select an existing plane.
2. Press the **Ctrl** key and drag a plane.



3. Release the mouse button.
4. To edit the offset distance, double click the offset plane and change the distance. This is the same method as **Offset** in Ref. Plane command.

Options in Detail

Target Entities

Entities – Selects target entities. The required target entities are different depending on the method.

Methods

The following methods are available in this command:

- Definition
 - Extract
 - Projection
 - Pick Multiple Points
 - Pick Point & Normal
 - Pick Point & Coplanar Axis
 - Convert
 - N Division
 - Offset
 - Rotation
 - Average
 - View Direction
 - Tangent
 - Orthogonal
 - Draw Line
 - Mirror
 - Extreme Position
-

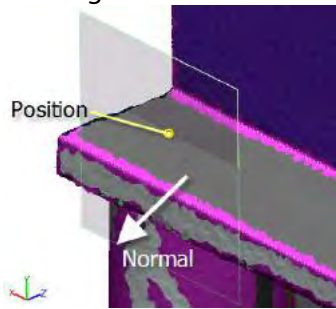
Definition – Creates a ref. plane by using the mathematical definition of the plane. The value can be input numerically or interactively picked by a point in the Model View.

Position & Normal – Creates a ref. plane using **Position** and **Normal** values.

Position – Specifies the values of X, Y and Z, or manually selects a point where the

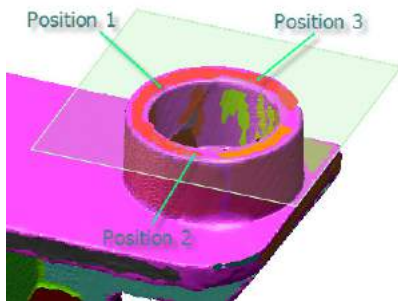
normal value will be automatically acquired.

Normal – Selects a point on the model to acquire a normal value of a picked object or used to specify a numerical value. By clicking the X, Y, or Z buttons, a normal can be aligned to a selected direction.



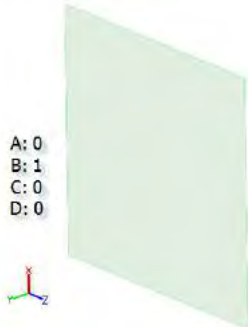
3 Positions – Creates a plane which intersects with three positions.

Position 1 and Position 2 and Position 3 – Chooses a position on the model or directly inputs a X,Y and Z value.

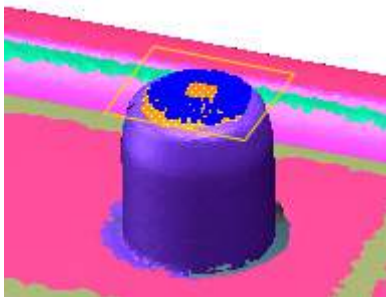


Coefficient - Creates a plane using specified A, B, C and D values which are coefficients of the plane equation ($Ax + By + Cz + D = 0$).

A, B, C, D – Inputs the numerical value of coefficient of a plane.

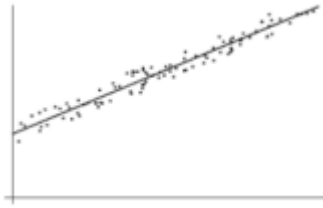


Extract – Extracts a plane from selected entities by using a fitting algorithm.



Fitting Type – Creates a best-fitted value by the least square fitting method when the **Best**

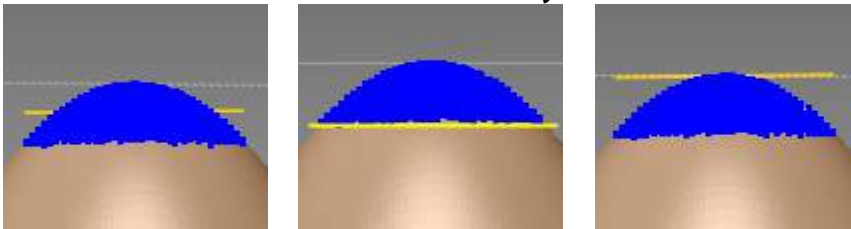
Fit option is selected. The least squares method is the approximate solution from distributed data.



Least Square Method

The lowest value which is parallel to Best Fit will be created from the minimum bounded box of selected entities when **Min Boundary** is selected.

The highest value which is parallel to Best Fit will be created from the maximum bounded box of selected entities when **Max Boundary** is selected.



Best Fit

Min Boundary

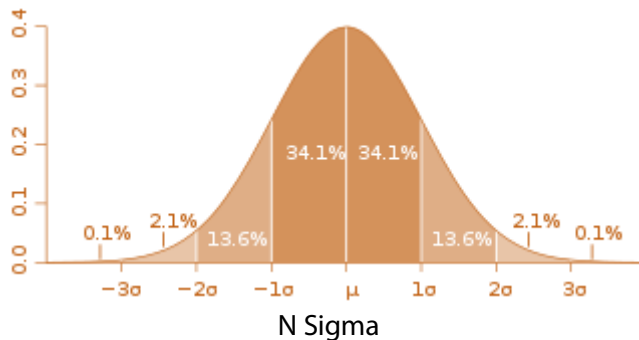
Max Boundary

Remove Outlier – Removes outlier data to achieve more accurate fitting results.

The **N*Sigma** method requires an N value and excludes data that is larger than N times the standard deviation, also known as the sigma.

The **Ratio method** requires a ratio value and only data that falls into the selected value will be used.

The **Absolute Distance** method requires a distance and the fitting will be done by excluding any data outside of that distance.



Use Normal – Excludes points that have a normal angle larger than the normal of fitted ref. geometry. After removing points, the application will internally generate fitted ref. geometry again from the refined points.

Sampling Ratio – Samples data points by a specified value. When the ratio is 100%, all selected data will be used. When the ratio is 50%, half of the selected data will be used.

Use Only Effective Data – Uses only effective data when fitting. Effective data is the actual data points used in fitting after sampling and outlier removal.

Axis Constraint – Constrain the direction of an axis.

Use Specified Direction As Initial Guess – Defines an initial axis that can be used

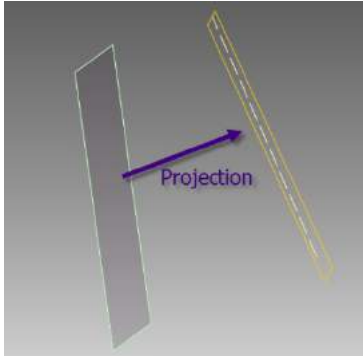
for finding the direction of a fitting shape.

Use Specified Direction As It Is – Manually define the direction of an axis. The direction of an axis can be defined by selecting an entity that has a vector property or manually typing the values.

Projection – Creates a plane by projecting a planar entity onto a linear entity.

To make a plane using the **Projection** method:

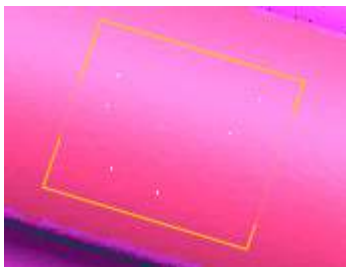
1. Select a planar entity such as a plane, face, or region.
2. Select a linear entity such as a line, vector, or edge.
3. Click **OK**.



Pick Multiple Points – Creates a plane by picking three or more points.

To make a plane using the **Pick Multiple Points** method:

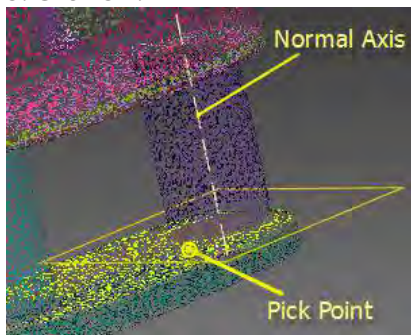
1. Pick three or more points.
2. Click **OK**.



Pick Point & Normal Axis – Creates a plane by picking a point (position) and selecting a normal axis.

To make a plane using the **Pick Point & Normal** method:

1. Select a linear entity for normal of the plane.
2. Pick a point for the position of the plane.
3. Click **OK**.

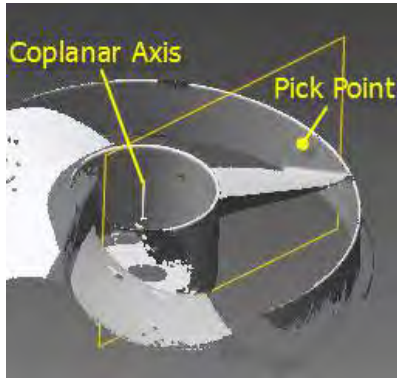


Pick Point & Coplanar Axis – Creates a plane on a coplanar axis.

To make a plane using the **Pick Point & Normal** method:

1. Select a linear entity for coplanar axis.
2. Pick a point for the position of the plane.

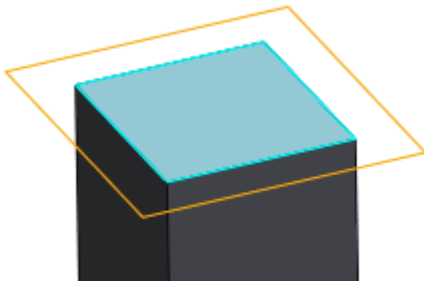
3. Click **OK**.



Convert – Creates plane on a selected planar entity.

To make a plane using the **Convert** method:

1. Select a planar entity (plane, face, region, etc).
2. Click **OK**.

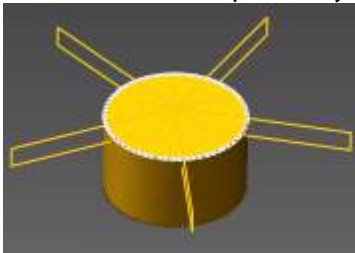


N Division – Creates planes by dividing selected entities. Multiple planes will be created evenly and perpendicularly to the selected entity.

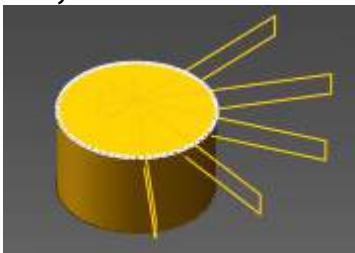
To make a plane using the **N Division** method:

1. Select a line, curve, edge or etc.
2. Define sub option.
3. Click **OK**.

Uniform – Creates planes by evenly dividing a selected entity by specified number.



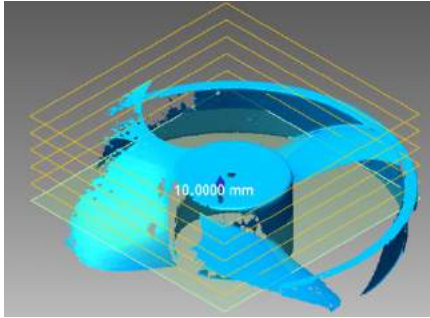
Offset – Creates planes by a specified offset distance and total number on a selected entity.



Offset – Creates a specified number of planes by offsetting them from a selected planar entity.

To make a plane using the **Offset** method:

1. Select a planar entity such as a plane, face, or region.
2. Set the total number and distance.
3. Click **OK**.



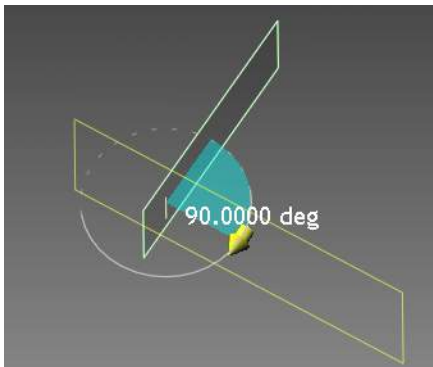
Note:

A ref. plane can be offset by using the **Ctrl** key and mouse drag an existing ref. plane.

Rotation – Creates multiple planes by rotating a planar entity.

To make a plane using the **Rotation** method:

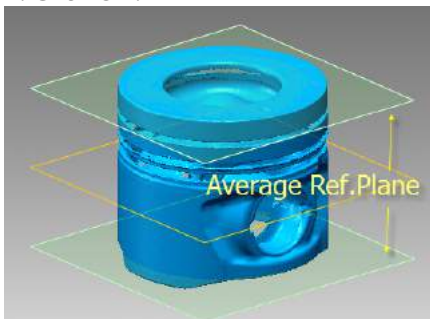
1. Select a planar entity such as a plane, face, or region.
2. Select a linear entity such as a line, vector, or edge which will be the rotation axis.
3. Set the total number of planes and the rotation angle.
4. Click **OK**.



Average – Creates a plane by averaging two selected planar entities. The planar entities do not need to be parallel.

To make a plane using the **Average** method:

1. Select two planar entities such as a plane, face, or region.
2. Click **OK**.



View Direction – Creates a plane in the current view direction.

To make a plane using the **View Direction** method:

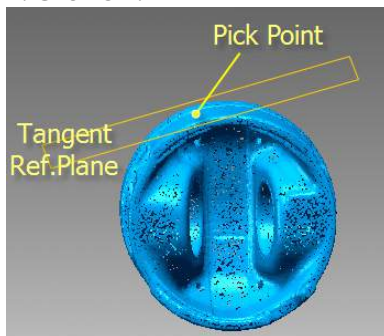
1. Pick a point, also known as a position, on the Model View.
2. The plane normal will be the viewing direction.
3. Click **OK**.



Tangent – Creates a plane that is tangent to a picked entity.

To make a plane using the **Tangent** method:

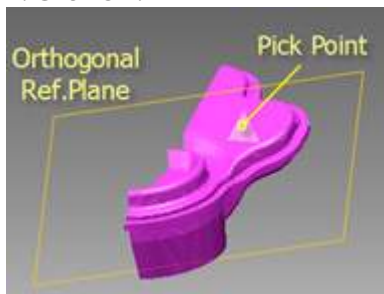
1. Pick a point also known as a position, on a curve point, poly-vertex, vertex, face or ref. point.
2. Click **OK**.



Orthogonal – Creates an orthogonal plane from a selected point (point-element entity) on a mesh. A point and two body faces can also be used.

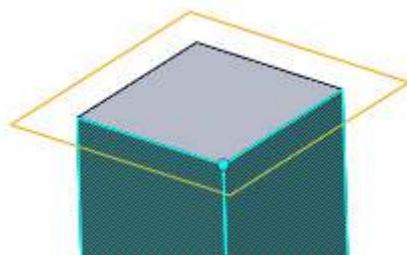
To make a plane using the **Orthogonal** method on a mesh:

1. Pick a point on a mesh.
2. Click **OK**.



To make a plane using the **Orthogonal** method on a body:

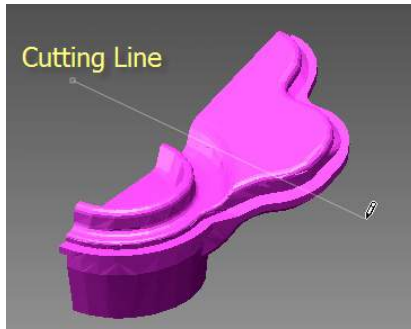
1. Pick a vertex on a body.
2. Select two body faces.
3. Click **OK**.



Draw Cutting – Creates a plane by drawing a line on the screen.

To make a plane using the **Draw Cutting** method on a body:

1. Draw a linear line on the screen.
2. Click **OK**.

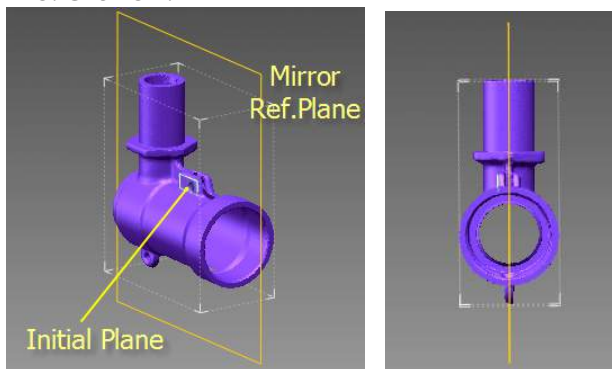


Mirror – Finds and creates a symmetrical plane of a mesh automatically. An initial plane which roughly indicates the normal direction of the plane is necessary.

Lock Normal Direction Of Mirror Plane As Selected Planes' uses the normal of the initial plane as it is.

To make a plane using the **Mirror** method:

1. Select an initial plane.
2. Select a mesh.
3. Click **OK**.

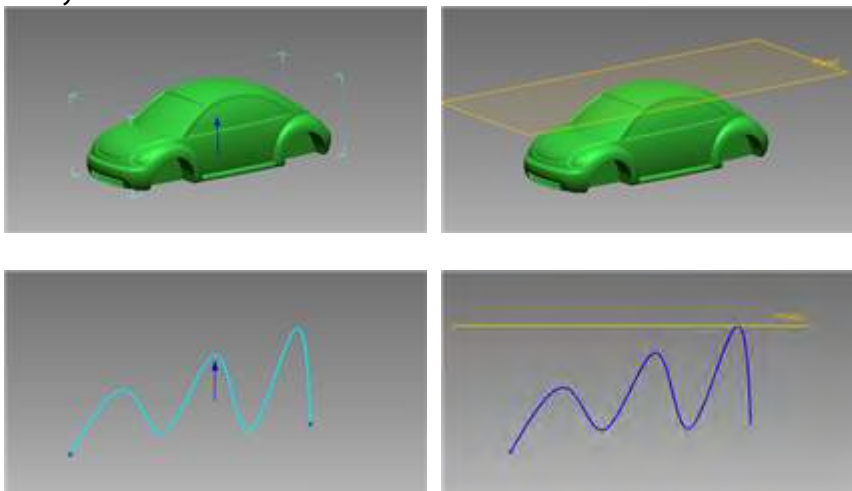


Extreme Position – Creates a plane at a maximum or minimum position of a selected entity in a specified direction.

To make a plane using the **Extreme Position** method:

1. Click the Direction button and select a linear entity such as a line, or vector.
2. Select a target entity.
3. Click **OK**.

The following images show generated planes at the extreme position to the Z direction on an entity.



See Also

[Insert > Ref.Geometry > Vector](#)

[Insert > Ref.Geometry > Coordinate](#)

[Insert > Ref.Geometry > Polyline](#)

[Insert > Ref.Geometry > Point](#)

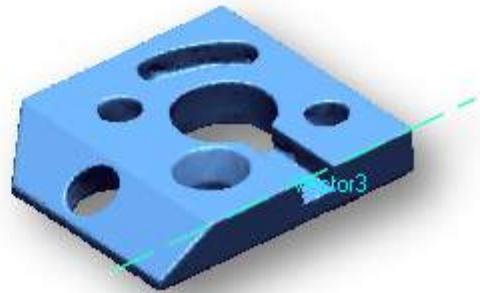
Insert > Ref. Geometry > Vector

What

A reference vector is a virtual axis with a direction and infinite size. A ref. vector is not a line entity and is used just to generate other features.



Cylinder Axis



Intersecting Two Planes

Why

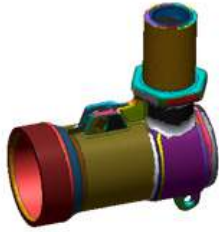
The **Ref. Vector** command is useful for:

- Setting an axis of a revolving body
- Creating a rotation axis for a circular pattern
- Finding the axis of a cylinder
- Finding the axis of a cone
- Finding revolving and extruding axes

How to Use

How to extract a cylinder axis from a mesh

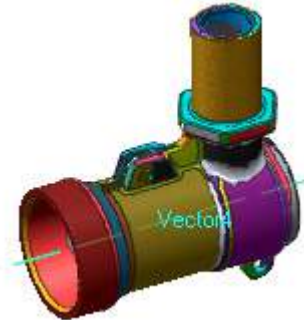
1. Prepare a mesh. A mesh with region groups is ideal.



2. Go to **Insert > Ref. Geometry > Vector** or click the icon in the toolbar.
3. Change the method to **Find Cylinder Axis**.
4. Select cylindrical regions.



5. Click **OK** to complete the command. The application will find an axis from the cylindrical regions.



Options in Detail

Target Entities

Entities – Selects target entities.

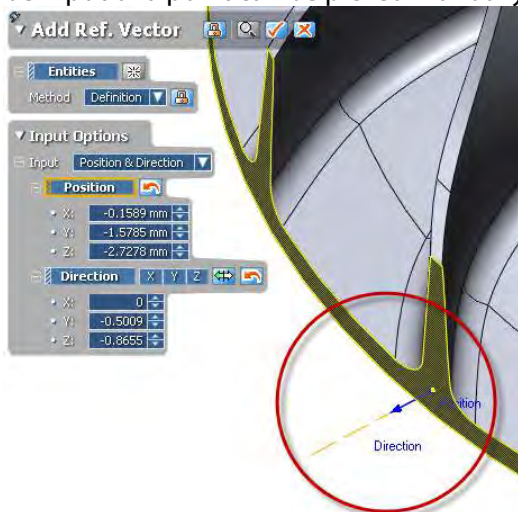
Methods

The following methods are available in this command:

- Definition
 - Extract
 - Find Slot Axis
 - Find Cylinder Axis
-

- Find Cone Axis
- Projection
- Pick Multiple Points
- Pick Point & Line
- Convert
- Intersect 2 Planes
- Average
- Tangent
- Cross 2 Line
- Revolving Axis
- Extrusion Axis
- Pattern Revolving Axis
- Pattern Translation Axis

Definition – Creates a vector using the mathematical definition of a vector. A numeric value can be input or a point can be picked manually in the Model View.



Position & Direction – Creates a ref. vector with **Position** and **Direction** values.

Position – Selects a point in the Model View or allows the manual input of specific values for X, Y and Z.

Direction – Selects a linear entity to acquire a directional value or allows the manual input of a specific numerical value. The X, Y or Z buttons align the direction to a selected direction.



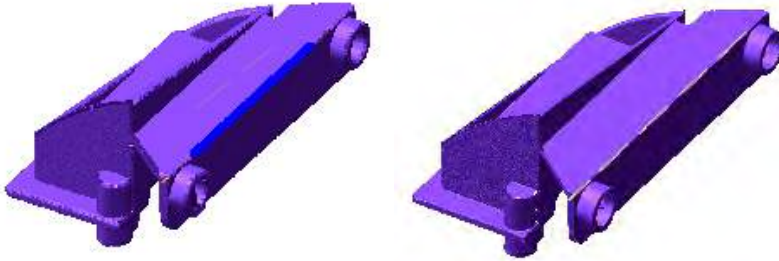
Start & End – Creates a vector by defining the **Start Position** and the **End Position**.

Start Position and End Position – Creates a vector by connecting two positions.

The positions can be selected in the Model View or manually input into the **X, Y, Z** boxes.



Extract – Extracts a vector from selected entities by using a fitting algorithm.

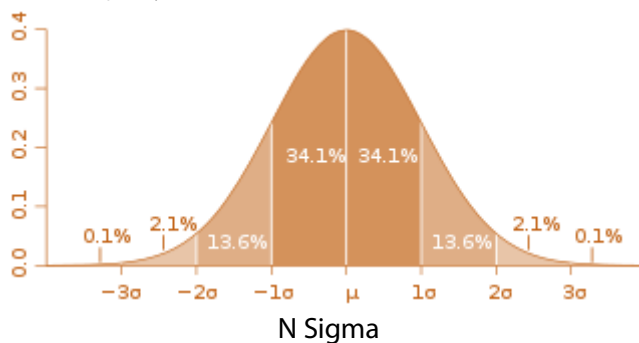


Remove Outlier – Removes outlier data to achieve more accurate fitting results.

The **N*Sigma** method requires an N value and excludes data that is larger than N times the standard deviation, also known as the sigma.

The **Ratio method** requires a ratio value and only data that falls into the selected value will be used.

The **Absolute Distance** method requires a distance and the fitting will be done by excluding any data outside of that distance.



Sampling Ratio – Samples data points by a specified value. When the ratio is 100%, all selected data will be used. When the ratio is 50%, half of the selected data will be used.

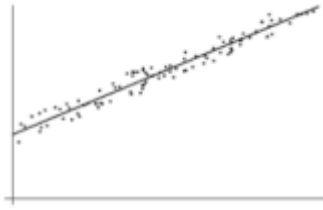
Use Only Effective Data – Uses only effective data when fitting. Effective data is the actual data points used in fitting after sampling and outlier removal.

Project On Plane - Projects a created vector onto a selected planar entity such as a plane, face, region, or sketch.

Find Slot Axis – Creates a vector of a slot on a selected entity using a fitting algorithm. The fitting option is the same as the **Extract** method.



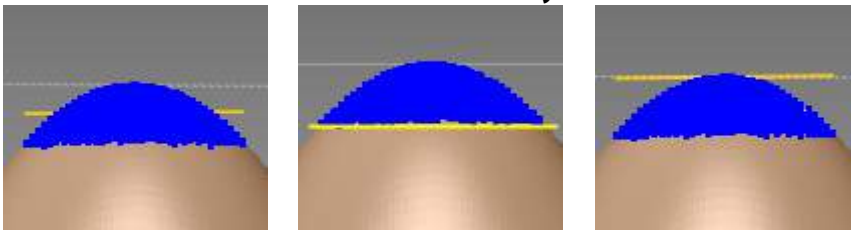
Fitting Type – Creates a best-fitted value by the least square fitting method when the **Best Fit** option is selected. The least squares method is the approximate solution from distributed data.



Least Square Method

The lowest value which is parallel to Best Fit will be created from the minimum bounded box of selected entities when **Min Boundary** is selected.

The highest value which is parallel to Best Fit will be created from the maximum bounded box of selected entities when **Max Boundary** is selected.

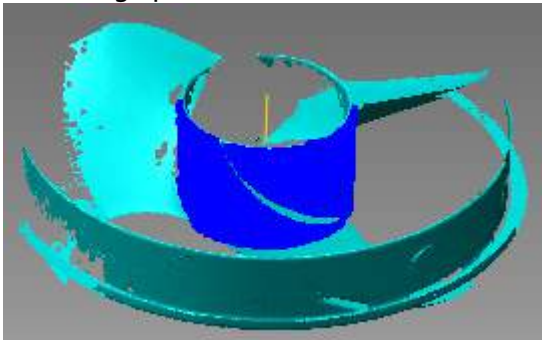


Best Fit

Min Boundary

Max Boundary

Find Cylinder Axis – Creates a cylinder axis from a selected entity by using the fitting algorithm. The fitting option is same as the **Extract** method.



Use Normal – Excludes points that have a normal angle larger than the normal of fitted ref. geometry. After removing points, the application will internally generate fitted ref. geometry again from the refined points.

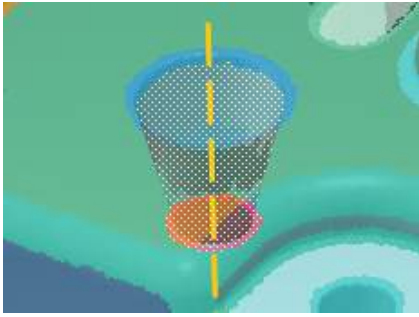
Fix Radius – Specifies the value of the radius of a cylinder if it is already known.

Axis Constraint – Constrain the direction of an axis.

Use Specified Axis As Initial Guess – Defines an initial axis that can be used for finding the direction of a fitting shape.

Use Specified Axis As It Is – Manually define the direction of an axis. The direction of an axis can be defined by selecting an entity that has a vector property or manually typing the values.

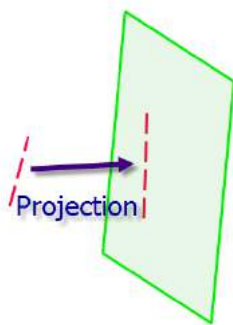
Find Cone Axis – Creates a cone axis from selected entities by using a fitting algorithm. The fitting option is same as the **Find Cylinder Axis** method.



Projection – Creates a vector by projecting a planar entity onto a linear entity.

To make a vector using the **Projection** method:

1. Select a planar entity such as a plane, face, or region.
2. Select a linear entity such as a line, vector, or edge.
3. Click **OK**.



Pick Multiple Points - Creates a vector by picking two or more points.

To make a vector using the **Pick Multiple Points** method:

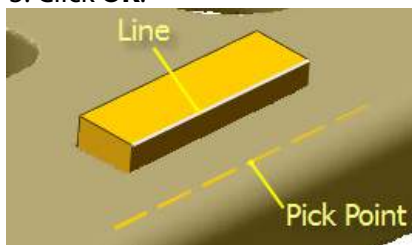
1. Pick two or more points.
2. Click **OK**.



Pick Point & Line – Creates a vector by picking a point as the position and a linear entity as the direction.

To make a vector using the **Pick Point & Line** method:

1. Select a linear entity for the direction of a vector.
2. Pick a point for the position of the vector.
3. Click **OK**.



Convert – Creates a vector as an axis of a selected entity or on a selected entity.

To make a vector using the **Convert** method:

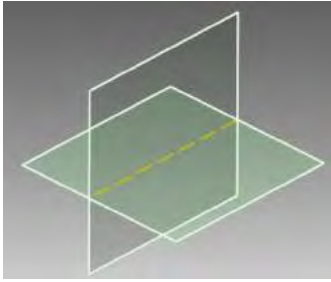
1. Select a cylindrical entity such as a face or a region, or a linear entity such as a line or an edge.

2. Click **OK**.

Intersect 2 Planes – Creates a vector from intersecting two planes.

To make a vector using the **Intersect 2 Planes** method:

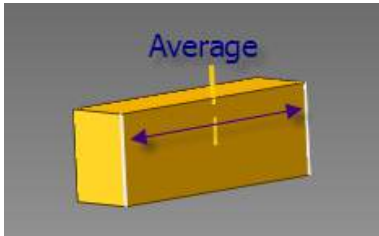
1. Select two planar entities such as planes, faces, regions.
2. Click **OK**.



Average – Creates a vector by averaging two linear entities.

To make a vector using the **Average** method:

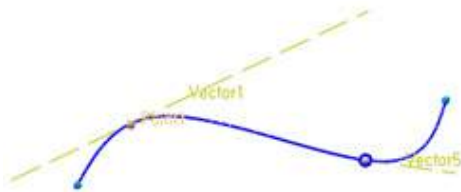
1. Select two linear entities such as lines, edges, vectors.
2. Click **OK**.



Tangent – Creates a vector that is tangent to a selected entity.

To make a vector by using the **Tangent** method:

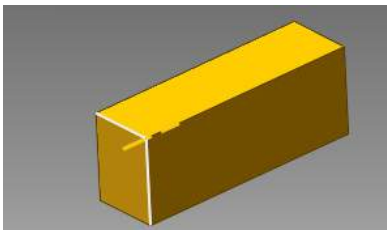
1. Pick a point on an entity such as a curve, edge, or poly-vertex.
2. Click **OK**.



Cross 2 Lines – Creates a vector from crossing two linear entities.

To make a vector using the **Tangent** method:

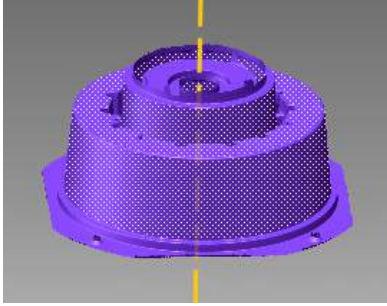
1. Select two linear entities such as lines, edges, or vectors.
2. Click **OK**.



Revolving Axis – Creates a vector that is an axis from a revolved mesh feature. The fitting option is same as the **Extract** method.

To make a vector using the **Revolving Axis** method:

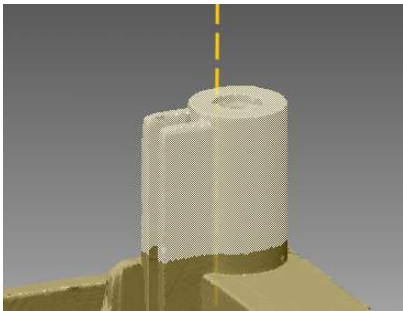
1. Select a revolved area on a mesh.
2. Click **OK**.



Extrusion Axis – Creates a vector in an extrusion direction from an extruded mesh feature. The fitting option is the same as the **Extract** method.

To make a vector using the **Extrusion Axis** method:

1. Select extruded area on a mesh.
2. Click **OK**.

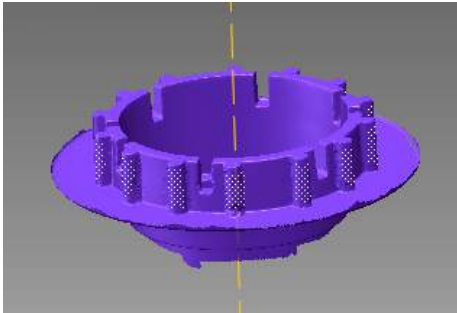


Use Specified Axis As Initial Guess – Defines an initial axis used to find the axis of the plane. A linear entity or a user defined direction can be chosen as a vector.

Pattern Revolving Axis – Creates a vector from the center axis of a revolved pattern feature.

To make a vector using the **Pattern Revolving Axis** method:

1. Select two or more pattern features on a mesh.
2. Click **OK**.

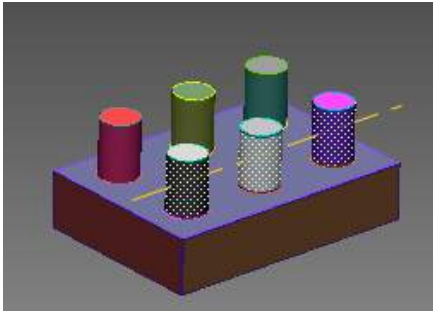


Sampling Ratio – Samples data points by a specified value. When the ratio is 100%, all selected data will be used. When the ratio is 50%, half of the selected data will be used.

Pattern Translation Axis – Creates a vector in a pattern direction from pattern features.

To make a vector using the **Pattern Translation Axis** method:

1. Select two or more pattern features on a mesh.
2. Click **OK**.



Sampling Ratio – See the **Pattern Revolving Axis** option.

See Also

[Insert > Ref.Geometry > Plane](#)

[Insert > Ref.Geometry > Coordinate](#)

[Insert > Ref.Geometry > Polyline](#)

[Insert > Ref.Geometry > Point](#)

Insert > Ref. Geometry > Coordinate

What

A reference coordinate is a local coordinate system that a user can use to define additional coordinates.



Why

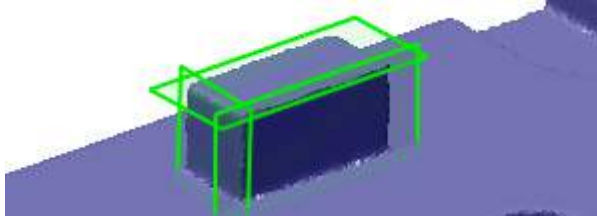
The **Ref. Coordinate** command is useful for:

- Transforming scan data
- Aligning scan data

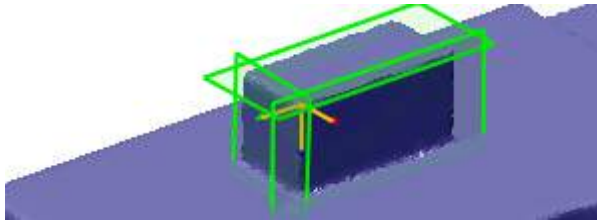
How to Use

How to add a coordinate using the intersection of 3 planes

1. Go to **Insert > Ref. Geometry > Coordinate**.
2. Change the method to **Intersect 3 Planes**.
3. Select three defined ref. planes.



4. Click **OK** to complete the command. A ref. coordinate at the corner of the 3 ref. planes will be created.



Options in Detail

Target Entities

Entities – Selects target entities.

Methods

The following methods are available in this command:

- Definition
 - Extract
 - Pick Multiple Points
 - Pick Point & 2 Lines
 - Pick Point & Coordinate
 - Insert 3 Planes
 - Tangent
-

Definition – Creates a ref. coordinate by the mathematical definition of the coordinate.

Origin & Axis – Creates a coordinate using the **Origin** and **Axis1** value.

Origin – Selects a point in the Model View or allows the manually input of the X, Y and Z values.

Axis1 / Axis2 – Pick a point on the model to acquire the direction of the picked object or specify numerical value. If you click X, Y or Z button, you can align normal to the selected direction.

Order – Defines the order of **Axis1** and **Axis2** as X, Y or Z. If the order is **XY**, **Axis1** will be set to **X** direction and **Axis2** will be set to **Y** direction.

Origin & Rotation – Creates a coordinate with the **Origin** and **Rotation** value.

Origin – Selects a point in the Model View or allows the manually input of the X, Y and Z values.

Rotation – Specifies the degree of an angle of an axis that a coordinate will rotate around.

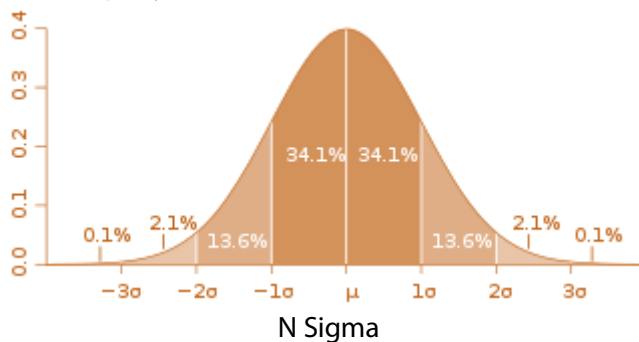
Extract – Extracts a coordinate from selected entities by using a fitting algorithm.

Remove Outlier – Removes outlier data to achieve more accurate fitting results.

The **N*Sigma** method requires an N value and excludes data that is larger than N times the standard deviation, also known as the sigma.

The **Ratio** method requires a ratio value and only data that falls into the selected value will be used.

The **Absolute Distance** method requires a distance and the fitting will be done by excluding any data outside of that distance.



Use Normal – Excludes points that have a normal angle larger than the normal of fitted ref. geometry. After removing points, the application will internally generate fitted ref. geometry again from the refined points.

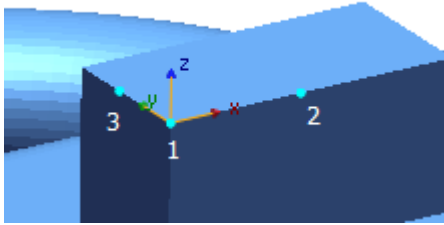
Sampling Ratio – Samples data points by a specified value. When the ratio is 100%, all selected data will be used. When the ratio is 50%, half of the selected data will be used.

Use Only Effective Data – Uses only effective data when fitting. Effective data is the actual data points used in fitting after sampling and outlier removal.

Pick Multiple Points – Creates a coordinate by picking points more than 3 points.

To make a coordinate using the **Pick Multiple Points** method:

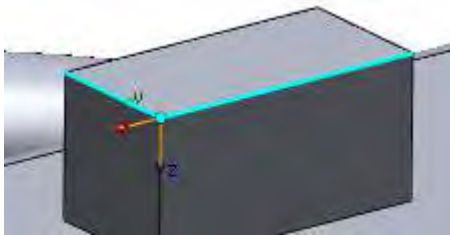
1. Pick 3 points.
2. Click **OK**.



Order – Refer to the **Definition** method.

Direction – Flips the directions of defined axes by clicking the Flip button 

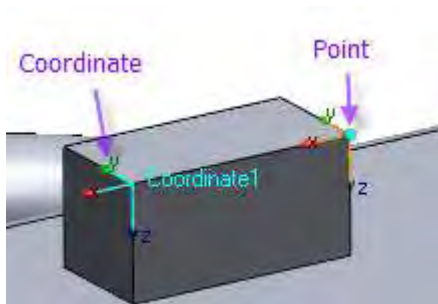
Pick Point & 2 Lines – Creates a coordinate from a point used for an origin and two lines used for directions. The direction of the coordinate on the first and second line will follow the settings in the **Order** option. For more about the **Order** option, see the **Definition** method.



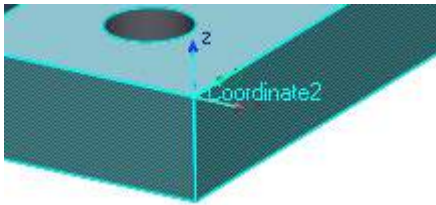
Order – Refer to the **Definition** method.

Direction – Refer to the **Pick Multiple Points** method.

Pick Point & Coordinate – Creates a coordinate from a point used for an origin and another coordinate.

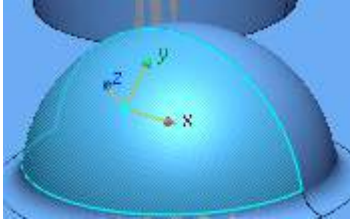


Intersect 3 Planes – Creates a coordinate from three planar entities.



Direction – Refer to the **Pick Multiple Points** method.

Tangent - Creates a coordinate tangent to a manually picked entity using a point on a mesh or a body face.



See Also

[Insert > Ref.Geometry > Plane](#)

[Insert > Ref.Geometry > Vector](#)

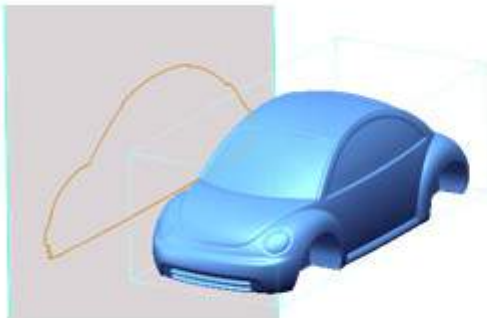
[Insert > Ref.Geometry > Polyline](#)

[Insert > Ref.Geometry > Point](#)

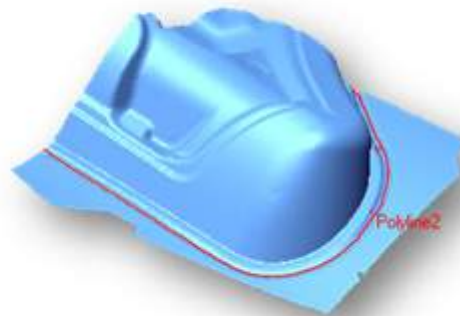
Insert > Ref. Geometry > Polyline

What

A reference polyline is a connected series of line segments. The polyline is used to extract the design intent or character lines from a mesh.



Silhouette



Beadline

Why

The **Ref. Polyline** command is useful for:

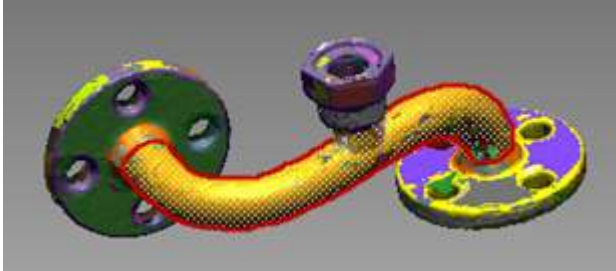
- Extracting silhouette lines
- Extracting parting lines
- Extracting fillet center lines
- Extracting pipe center lines
- Extracting bead center lines
- Extracting boundaries from selections
- Extracting revolving and extrusion profiles

- Extracting sharp edges on rounded shapes

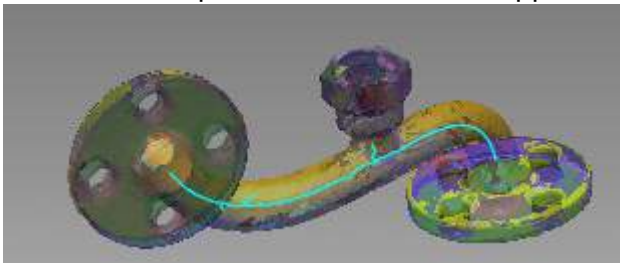
How to Use

How to find the curve in the center of pipe using mesh

1. Go to **Insert > Ref. Geometry > Polyline** or click the icon in the toolbar.
2. Change the method to **Pipe**.
3. Select the pipe area.



4. Click **OK** to complete the command. The application will find the pipe line.



Options in Detail

Target Entities

Entities – Selects target entities

Methods

The following methods are available in this command:

- Intersection
 - Silhouette
 - Parting
 - Fillet
 - Sweep
 - Pipe
 - Bead Line
 - Boundary
-

-
- Projection
 - Revolving Profile
 - Extrusion Profile
 - Extract Virtual Sharp Edge
-

Intersection – Creates polylines at an intersection between a mesh and a planar entity.

To make a polyline using the **Intersection** method:

1. Select a mesh.
2. Select a planar entity such as a plane, face, or region.
3. Click **OK**.

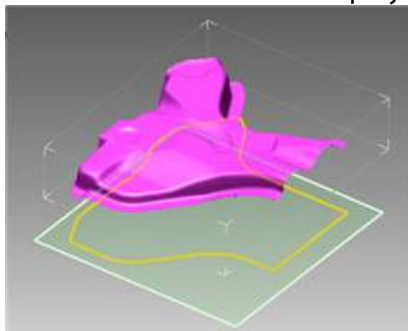


Silhouette – Creates silhouette polylines using a mesh and a plane or vector

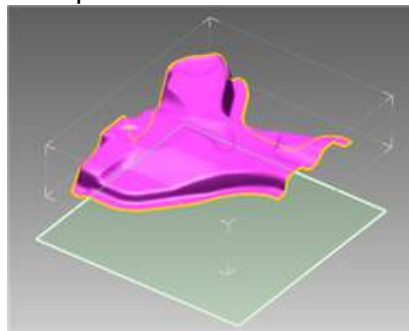
To make a polyline using the **Silhouette** method:

1. Select a mesh.
2. Select a planar entity or linear entity for the projection direction
3. Click **OK**.

2D Silhouette – Creates a 2D polyline on a plane.

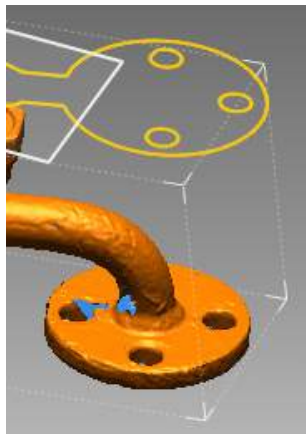


Check 2D Silhouette

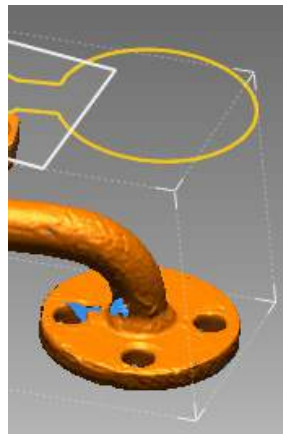


Uncheck 2D Silhouette

Outer Loop Only – Creates only outermost silhouette polylines



Check Outer Loop Only

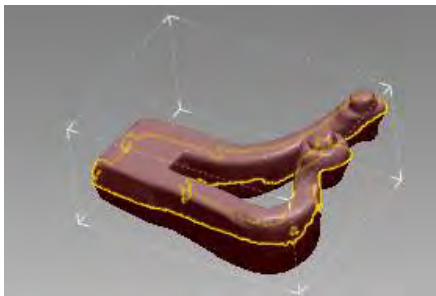


Uncheck Outer Loop Only

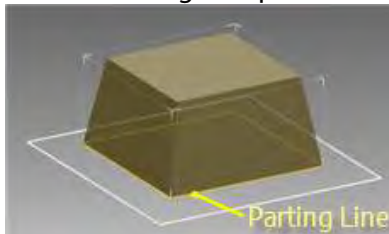
Parting – Creates parting lines from mesh

To make a polyline using the **Parting** method:

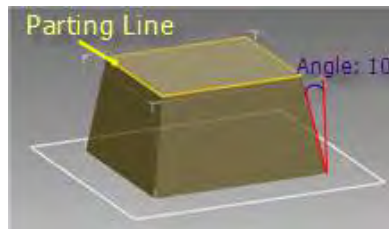
1. Select a mesh.
2. Select a planar entity or linear entity for the projection direction
3. Click **OK**.



Draft Angle – Poly-faces within the draft angle will be removed to calculate a parting line after a draft angle is specified



Draft Angle: 0



Draft Angle: 10

Fillet – Creates polylines in the center of a fillet feature

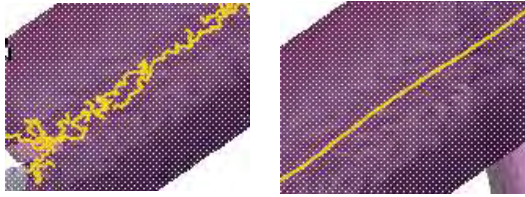
To make a polyline using the **Fillet** method:

1. Select fillet areas on a mesh.
2. Click **OK**.



Sampling Ratio – Samples data points by a specified value. When the ratio is 100%, all selected data will be used. When the ratio is 50%, half of the selected data will be used.

Thinning – Removes noisy polylines to generate a single line. A **Search Depth** value must be specified and is used to clean a polyline with the inputted search depth value.



Uncheck Thinning

Check Thinning

Sweep – Creates a sweep path from a sweep feature
To make a polyline using the **Sweep** method:

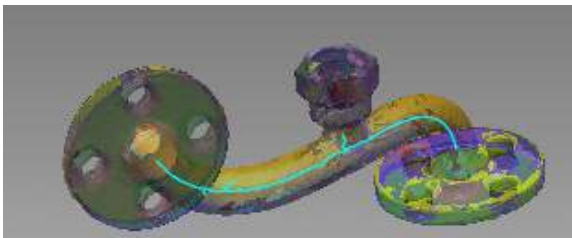
1. Select sweep areas on a mesh.
2. Click **OK**.



Thinning – See the **Fillet** method's sub-option under **Ref. Polyline**.

Pipe – Creates a pipe center path from a pipe feature
To make a polyline using the **Pipe** method:

1. Select pipe areas on a mesh.
2. Click **OK**.

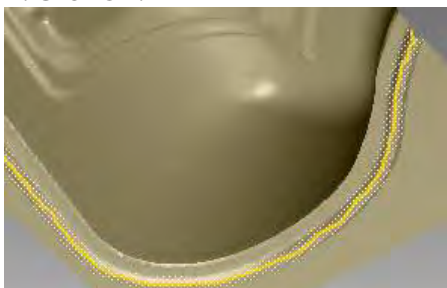


Thinning – See the **Fillet** method's sub-option under **Ref. Polyline**.

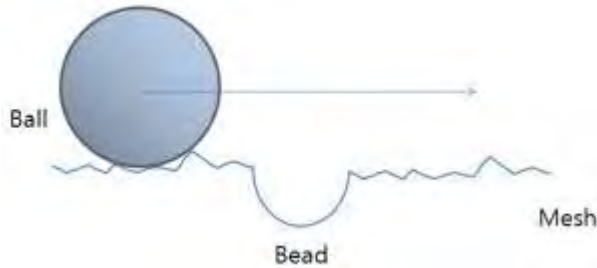
Inner / Outer – Selects the normal direction of a pipe.

Bead Line – Creates bead center polylines from a bead feature.
To make a polyline using the **Bead Line** method:

1. Select bead areas on a mesh.
2. Click **OK**.



Radius – Specifies the radius of a virtual ball that rolls over the surface of a mesh that is used to detect the boundaries of bead lines. The radius of the virtual ball needs to be larger than bead lines because if it is set as a very small value it might follow the shape of bead lines instead of detecting their boundaries. A value of 0 will allow the application to automatically set the radius.



Tolerance – Areas within a given distance will be treated as noise and ignored.



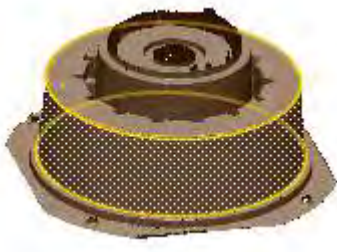
Thinning – See the **Fillet** method's sub-option under **Ref. Polyline**.

Convex / Concave – Chooses a bead shape.

Boundary – Creates boundary polylines from a selected area.

To make a polyline using the **Boundary** method:

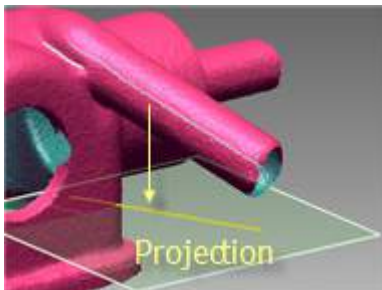
1. Select areas on a mesh.
2. Click **OK**.



Projection – Projects a polyline onto a planar entity.

To make a polyline using the **Projection** method:

1. Select an existing polyline.
2. Select a planar entity such as a plane or face.
3. Click **OK**.

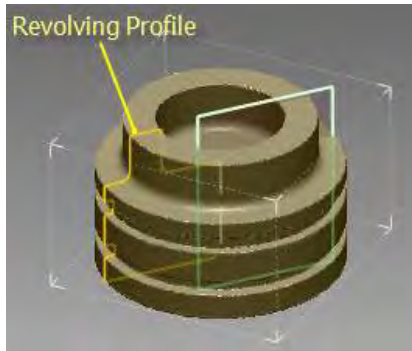


Revolving Profile – Creates a revolving profile from a selected area.

To make a polyline using the **Revolving Profile** method:

1. Select a mesh or areas on a mesh.
2. Select a ref. vector as the revolution axis.

3. Select a planar entity such as a plane or face for the sketch plane.
4. Click **OK**.



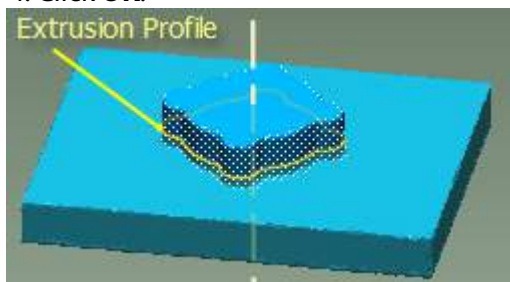
Use Normal – Excludes points that have a normal angle larger than the normal of fitted ref. geometry. After removing points, the application will internally generate fitted ref. geometry again from the refined points.

Thinning – See the **Fillet** method's sub-option under **Ref. Polyline**.

Extrusion Profile – Creates an extrusion profile from a selected area.

To make a polyline using the **Extrusion Profile** method:

1. Select a mesh or areas on a mesh.
2. Select a ref. vector which will be the extrusion direction.
3. Select a planar entity such as a plane or a face for the sketch plane.
4. Click **OK**.



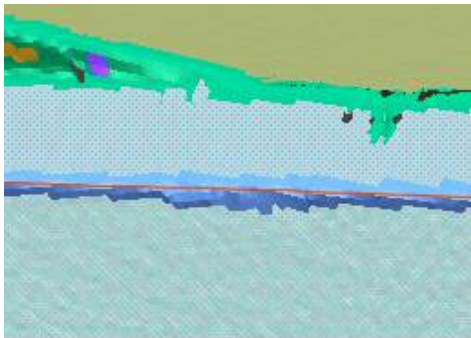
Use Normal – See the **Revolving Profile** method's sub-option under **Ref. Polyline**.

Thinning – See the **Fillet** method's sub-option under **Ref. Polyline**.

Extract Virtual Sharp Edge – Creates sharp edges on rounded features on a mesh or a body

To make a polyline using the **Extract Virtual Sharp Edge** method:

1. Select a side area around a rounded feature.
2. Select an opposite side area around the rounded feature.
3. Click **OK**.



Fit Surface From Region – Internally creates fitting surfaces to find intersections between

surfaces

Expand Ratio – Sets a value for the expand ratio used to find virtual edges. This option expands a selected region. To find a virtual edge between far distance regions, set a large enough value to make an intersection.

More Options

Delete Small Cluster – Removes noisy polylines that have less than a specified number of connected lines.

See Also

[Insert > Ref.Geometry > Plane](#)

[Insert > Ref.Geometry > Vector](#)

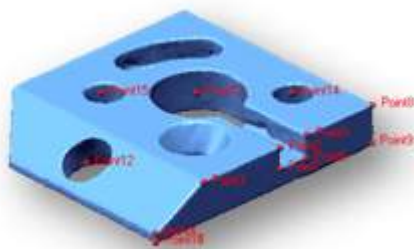
[Insert > Ref.Geometry > Coordinate](#)

[Insert > Ref.Geometry > Point](#)

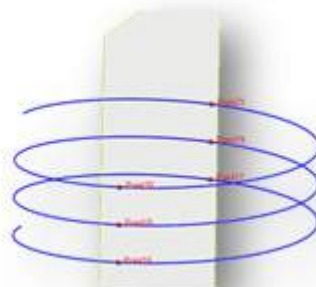
Insert > Ref. Geometry > Point

What

A reference point is a zero-dimensional entity. A reference point is used to mark a specific position on a model or in space.



Ref. Point on Mesh



Intersection between curve and plane

Why

The **Ref. Point** command is useful for:

- Finding a the center of a circle, slot, rectangle, polygon, or sphere
- Finding an average point from multiple points
- Finding the intersection point of two lines
- Finding the intersection point of a line and a face
- Finding the intersection point of three planes
- Importing positions

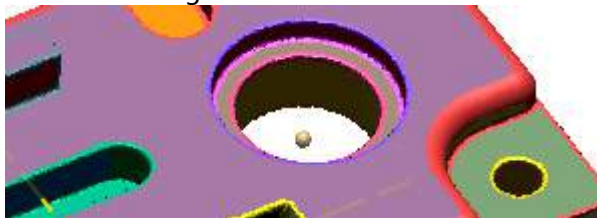
How to Use

How to make an average point from a region

1. Go to **Insert > Ref. Geometry > Point** or click the icon in the toolbar.
2. Change the method to **Extract**.
3. Select a region.



4. Click **OK** to complete the command. The application will extract an average point from the selected region.



Options in Detail

Target Entities

Entities – Selects target entities.

Methods

The following methods are available in this command:

- Definition
 - Extract
 - Find Circle Center
-

-
- Find Slot Center
 - Find Rectangle Center
 - Find Polygon Center
 - Find Sphere Center
 - Projection
 - Pick Multiple Points
 - Convert
 - N Division
 - Between 2 Points
 - Intersect 2 Lines
 - Intersect Line(Curve, Axis) & Face
 - Intersect 3 Planes
 - Import
-

Definition – Creates a point by using the mathematical definition of a point. The value can be input numerically or interactively picked by a point in the **Model View**.

Extract – Creates a point from selected entities by using a fitting algorithm.

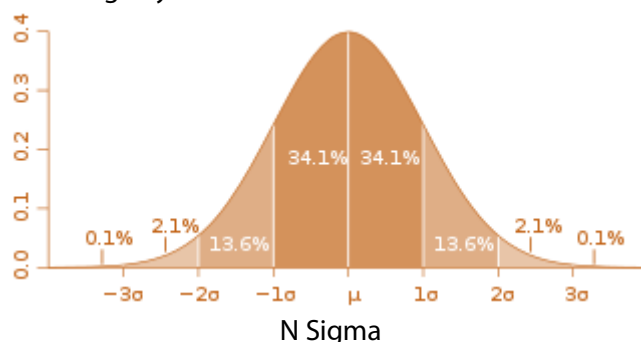


Remove Outlier – Removes outlier data to achieve more accurate fitting results.

The **N*Sigma** method requires an N value and excludes data that is larger than N times the standard deviation, also known as the sigma.

The **Ratio** method requires a ratio value and only data that falls into the selected value will be used.

The **Absolute Distance** method requires a distance and the fitting will be done by excluding any data outside of that distance.



Sampling Ratio – Samples data points by a specified value. When the ratio is 100%, all selected data will be used. When the ratio is 50%, half of the selected data will be used.

Use Only Effective Data – Uses only effective data when fitting. Effective data is the actual data points used in fitting after sampling and outlier removal.

Find Circle Center – Creates a circle center point from selected entities.



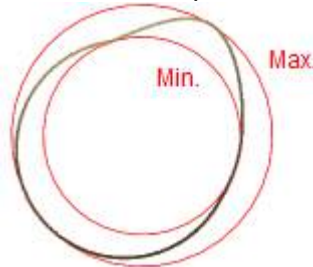
Fitting Type – When **Best Fit** is selected, the least squares fitting method is used. Minimum separation is a fitting method that uses a pair of concentric fitting shapes that contains all data points to extract both a Min. Bound and Max Bound.

When **Min. Bound** is selected for a circle, the smallest value of the radial distances between a pair of concentric circles of minimum separation is used.

When **Max. Bound** is selected the largest value from minimum separations is used. Min. Bound and Max. Bound share the same center point.



Best Fit



Min / Max Boundary
from Minimum Separation

The **Hole** method is available under the **Find Circle Center** option. This method is useful for finding the center of a hole in a model by locating a candidate area around a hole from a selection and generating a fitting circle based on the center of the hole.



1. Selection



2. Candidate area for
circle fitting



3. Find a center of a hole

Remove Outlier, Sampling Ratio, Use Only Effective Data – Identical to the **Extract** method.

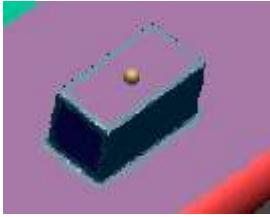
Project On Plane -- Projects a created vector onto a selected planar entity such as a plane, face, region, or sketch.

Fix Radius – Specifies the value of the radius of a cylinder if it is already known.

Find Slot Center – Extracts the center point of a slot from selected entities using a fitting algorithm. The sub-options are the same as the **Find Circle Center** method under **Ref. Point**.

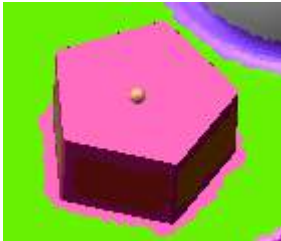


Find Rectangle Center – Extracts the center point of a rectangle from selected entities using a fitting algorithm. The sub-options are the same as the **Find Circle Center** method under **Ref. Point**.



Rounded Corner – Finds the center point of a rounded corner rectangle.

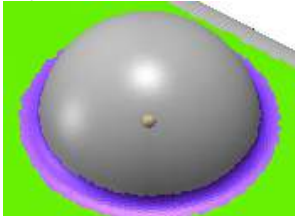
Find Polygon Center – Extracts the center point of a polygon center from selected entities using a fitting algorithm. The sub-options are the same as the **Find Circle Center** method under **Ref. Point**.



Rounded Corner – Used for polygon corners that are round.

Num. Of Edges – Specifies the number of edges of a polygon.

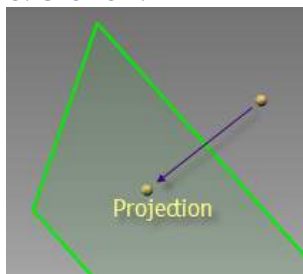
Find Sphere Center - Extracts the center point of a sphere from selected entities using a fitting algorithm. The sub-options are the same as the **Find Circle Center** method under **Ref. Point**.



Projection – Extracts a point by projecting onto another entity.

To make a point using the **Projection** method:

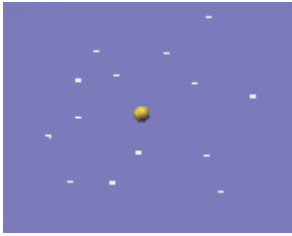
1. Select points such as ref. points, curve points or vertices.
2. Select a target entity that selected points will be projected onto.
3. Click **OK**.



Pick Multiple Points – Creates an average point by choosing one or more points.

To make a point using the **Pick Multiple Points** method:

1. Pick points on scan data or select points such as ref. points, vertices or circles.
2. Click **OK**.



Convert – Create a center point from a selected entity.

To make a point using the **Convert** method:

1. Select a circle or an arc.
2. Click **OK**.



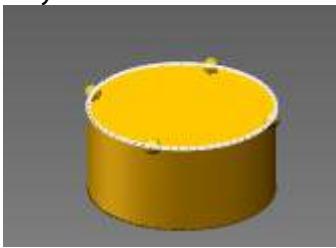
N Division – Creates multiple points by dividing a selected curve, vector, or mesh data.

To make a point using the **N Division** method:

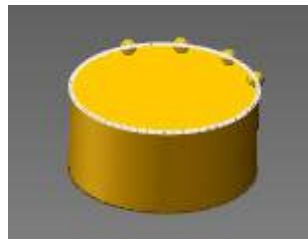
1. Select a curve.
2. Click **OK**.

Uniform – Creates planes by evenly dividing a selected entity by a specified number.

Offset – Creates planes by a specified offset distance and total number on a selected entity.



Uniform

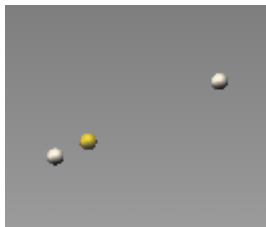


Offset

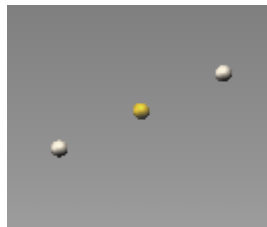
Between 2 Points – Creates a point between two points using a **Ratio** value to decide the position.

To make a point using the **Between 2 Points** method:

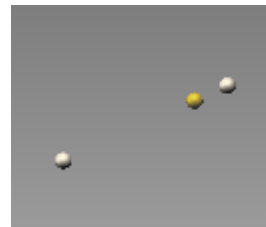
1. Select two points.
2. Set the ratio of position between two selected points.
3. Click **OK**.



Ratio: 0.2



Ratio: 0.5

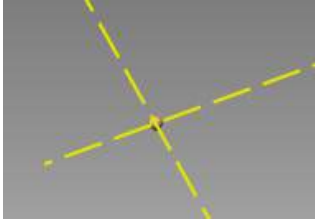


Ratio: 0.8

Intersect 2 Lines – Creates a point from the intersection of two linear entities.

To make a point using the **Intersect 2 Lines** method:

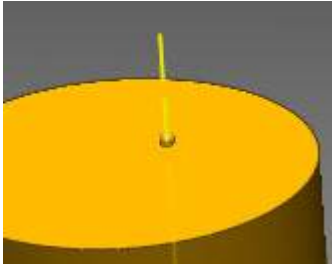
1. Select two linear entities such as a line, edge, or vector.
2. Click **OK**.



Intersect Line (Curve, Axis) & Face - Creates a point from the intersection of a curve or an axis of a geometric entity and a face.

To make a point using the **Intersect Line(Curve, Axis) & Face** method:

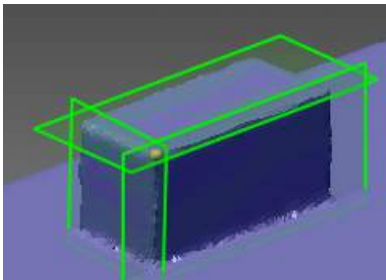
1. Select a curve such as a line, edge, vector, 2D or 3D curve.
2. Select a face.
3. Click **OK**.



Intersect 3 Planes – Creates a point from the intersection of three selected planar entities.

To make a point using the **Intersect 3 Planes** method:

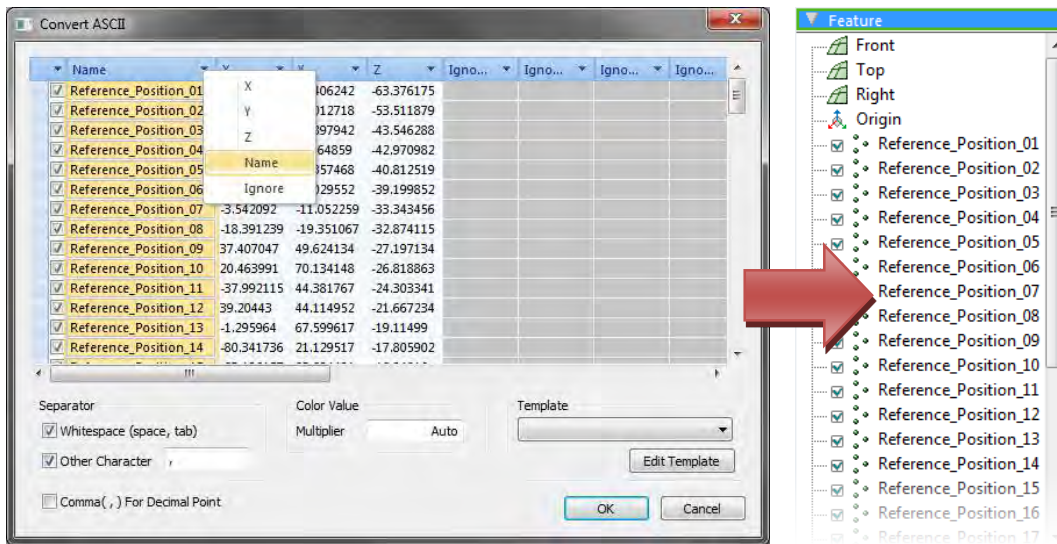
1. Select three planar entities such as a face, region, or plane.
2. Click **OK**.



Check Orthogonality – Allows the creation of a point only if three planes are orthogonal.

Import – Creates points by importing an ASCII file. A text file containing X, Y and Z coordinates separated by tabs or commas can be imported through the [ASCII converter](#).

When a text file is opened the ASCII Converter will appear. Column definition, separators such as white space or user defined characters, and commas for decimal point in the ASCII Convert dialog box can be set.



See Also

- [Insert > Ref.Geometry > Plane](#)
- [Insert > Ref.Geometry > Vector](#)
- [Insert > Ref.Geometry > Coordinate](#)
- [Insert > Ref.Geometry > Polyline](#)

7.4.6. Insert > Solid

The **Insert Solid** menu features various commands used to generate and edit solid bodies and surfaces.

Generating Solid Bodies from Sketches

- **Extrude**
Creates extruded bodies with a sketch, direction and length
- **Revolve**
Creates revolved bodies with a sketch, axis and revolution angle
- **Sweep**
Creates swept bodies with a sketch and path
- **Loft**
Creates lofted bodies with a set of sketches

Generating Solid Bodies from Other Bodies

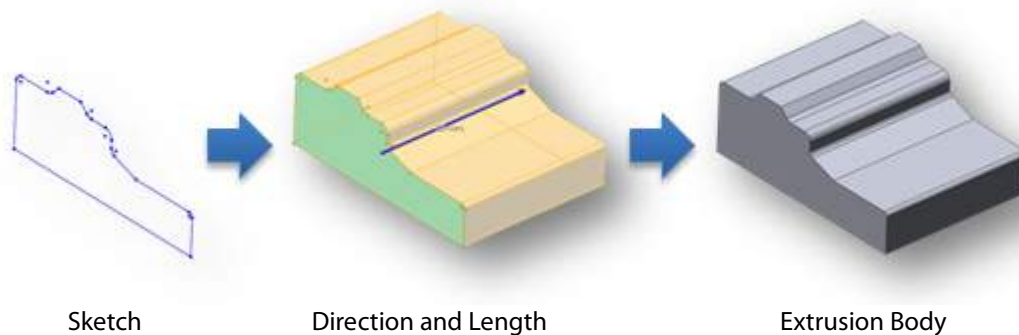
- **Offset**
Creates offset faces from existing body faces

- **Thicken**
Creates a constant thickness solid body from a surface body
- **Hollow**
Creates a constant thickness solid body from a solid body
- **Cut**
Creates a solid body by cutting an existing solid body
- **Boolean**
Creates a solid body by merging, cutting or intersecting with another body

Insert > Solid > Extrude

What

The **Extrude** command stretches an object section in a straight direction and then generates a closed solid body. The object section is expressed by a single profile or multiple profiles on a sketch. A profile is drawn by sketch entities such as a single circle or a spline or the combination of lines, arcs or splines.



Note:

To generate an extruded solid feature, a profile must be closed. The **Surface > Extrude** command can be used to generate an extruded feature by using an open profile.

Why

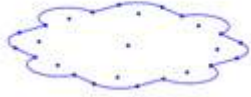
The **Extrude** command is useful for:

- Creating a body for a straight object
- Creating a boss or rib on a plastic part
- Creating a hole in an object
- Creating a straight pipe
- Creating the outside of a building
- Creating a pillar for a building or bridge

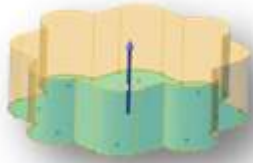
How to Use

A. How to generate an extrusion solid body

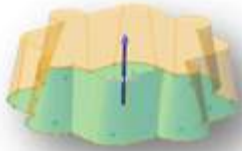
1. Draw a sketch or a mesh sketch.



2. Go to **Insert > Solid > Extrude** or click the icon in the toolbar. Select the sketch as the **Base Sketch**. Specify the **Length** for the extrusion.



3. The **Draft** option generates a tapered body. Specify the value as **10** degrees.

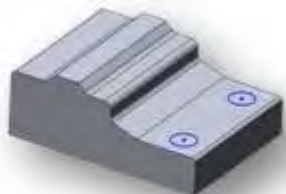


4. Click **OK** to complete the command.

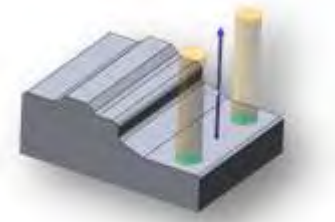


B. How to generate holes on a solid body

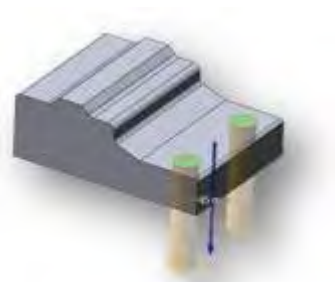
1. Draw a sketch or a mesh sketch.



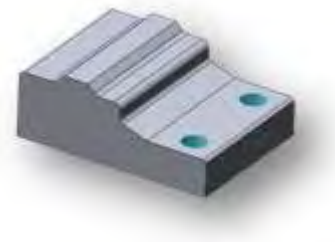
2. Go to **Insert > Solid > Extrude** or click the icon in the toolbar and then select the sketch as the **Base Sketch**.



3. Click the **Flip Direction** button to reverse the extrusion. Choose the **Cut** option under **Result Operator** to cut holes in the main solid body.



4. Click **OK** to complete the command.



Options in Detail

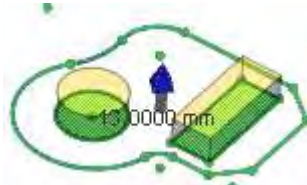
Select Entities

Base Sketch – Selects a **Sketch** of the **Mesh Sketch** as a base sketch. The base sketch is a 2D sectional profile of an extrusion object.

Profiles – Select or deselect loops used to extrude a desired profile. Multiple profiles can be drawn on a sketch.



Extrude Loop 1



Extrude Loop 2 & 3

Custom Direction – Specifies a different extrusion direction as the default direction for a linear entity with a normal. The default extrusion direction will be the normal of the sketch plane.



Default Extrusion



Custom Direction Extrusion

Methods

The following methods are available in this command:

Blind

Defines the extrusion height.

Through All

Defines the extrusion height to the end of another body.

Up To Vertex

Defines the extrusion height to a vertex.

Up To Region

Defines the end condition to a region.

Up To Surface

Defines the end condition to a surface.

Up To Body

Defines the end condition to a body.

Mid Plane

Defines the symmetry extrusion feature from a sketch.

Blind – Extends a profile to a specified distance.



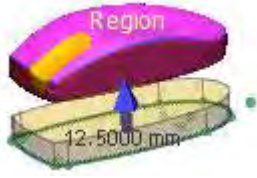
Through All – Extends a profile through all existing solid or surface bodies in the extrusion direction.



Up To Vertex – Extends a profile to a selected vertex.



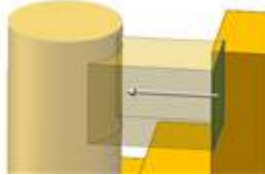
Up To Region – Extends a profile to a selected region on a mesh. A fitting surface is created from the region during the process and cuts the extruded body at the intersection. The sub-option **Region Fitting Option** can be used in this command: see [Insert > Surface > Mesh Fit](#).



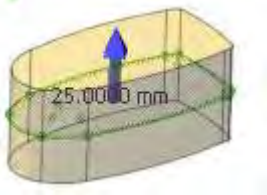
Up To Surface – Extends a profile to a selected surface body that cuts the extruded body at the intersection.




Up To Body – Extends a profile to a selected surface or solid body used when extruding an assembly part to another body.

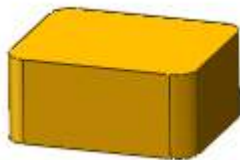


Mid Plane – Extends a profile in both directions leaving a selected sketch in the middle of the extrusion.

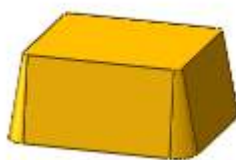


More Options

Draft – Applies a draft angle to a model that can be flipped by using the **Flip Direction** button . Draft angles are used for a mold to ease in the removal of a molded part.

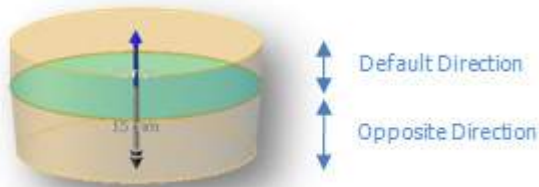


Draft: 0



Draft: 10

Opposite Direction – Extrudes a body shape from a base sketch in the opposite direction of the default direction.



Result Operator

The **Result Operator** option is available when one or more bodies exist. The **Boolean** command can be used to cut or merge with other bodies later.



Without Cut and Merge

Cut – Cuts bodies that intersect with a created extruded body.



With Cut Option

Merge – Merges bodies that intersect with a created extruded body.



With Merge Option

See Also

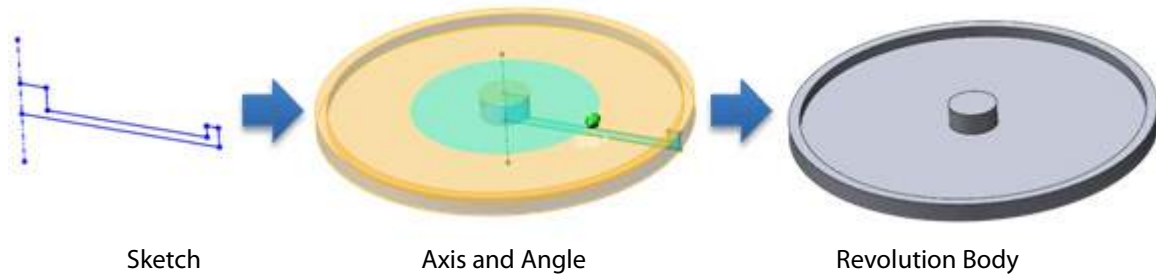
[Insert > Modeling Wizard > Extrusion Wizard](#)

[Insert > Surface > Extrude](#)

Insert > Solid > Revolve

What

The **Revolve** command turns an object section around a revolution axis and generates a closed solid body. The object section can be expressed by a single or multiple profiles on a sketch and is drawn by circle, spline, lines and arc sketch entities.



Note:

To generate a revolved solid feature, a profile must be closed. To generate a revolved feature by using an open profile, use the **Surface > Revolve** command.

Why

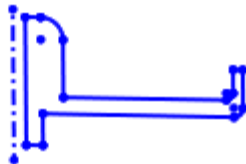
The **Revolve** command is useful for:

- Creating a fully revolved object
- Creating a half or angled revolved object

How to Use

How to generate a revolved body using a revolution angle

1. Draw a sketch or a mesh sketch that includes a profile and a revolution axis.



Note:

A revolution axis can be a construction line in a sketch or other linear entities out of a sketch such as a vector or an edge on the sketch.

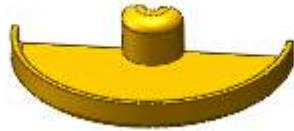
2. Go to **Insert > Solid > Revolve** or click the icon in the toolbar.
3. Select a sketch as a **Base Sketch**. If a sketch has a construction line, or axis, the application will automatically recognize it as the **Axis**.



4. Select the desired **Method**. Enter **180** degrees as the **Revolving Angle**.



5. Click **OK** to complete the command.

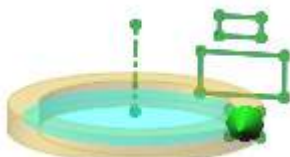


Options in Detail

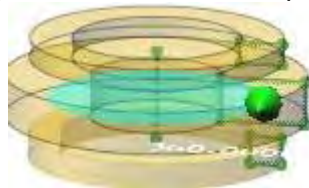
Select Entities

Base Sketch – Selects a sketch to revolve.

Profiles – Selects desired loops to revolve from multiple loops.



Revolve Loop 1



Revolve Loop 1, 2 & 3

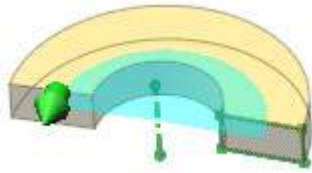
Axis – Selects a linear entity to revolve a sketch around an axis.

Methods

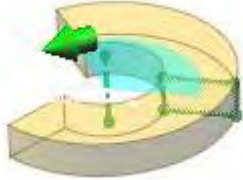
The following methods are available for this command:

- **One Direction**
Revolution direction in a single direction
 - **Mid Plane**
Symmetrical revolution directions to both side of a sketch
 - **Two Direction**
Two different revolution directions
-

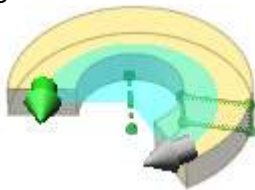
One Direction – Creates a revolved feature in one direction from a sketch.



Mid Plane – Creates a symmetry revolved feature in both the clockwise and counterclockwise directions from a base sketch.



Two-Direction – Creates a revolved feature in the clockwise and counterclockwise directions from a sketch plane with different angle values. The total of the two angles cannot exceed 360 degrees.



More Options

Angle – Specifies the revolution angle from a base sketch.

Opposite Angle – Specifies the revolution angle of the side opposite to the main angle and is available only using the **Both Directions** method.

Result Operator

The **Result Operator** option is available when one or more bodies exist. The **Boolean** command can be used to cut or merge with other bodies later.



Without Cut and Merge

Cut – Cuts bodies that intersect with a created extruded body.



With Cut Option

Merge – Merges bodies that intersect with a created extruded body.



With Merge Option

See Also

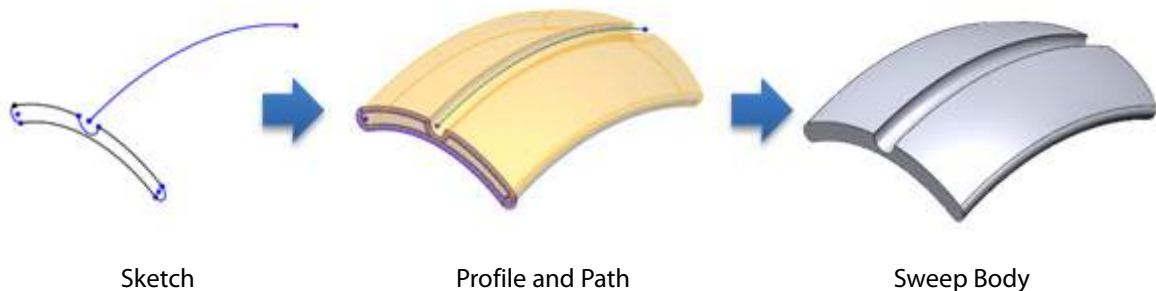
[Insert > Modeling Wizard > Revolution Wizard](#)

[Insert > Surface > Revolve](#)

Insert > Solid > Sweep

What

The **Sweep** command traces the path of an object section and then generates a closed solid body. The object section can be expressed by a single or multiple profiles on a sketch and is drawn by circle, spline, lines and arc sketch entities.



Note:

To generate a sweep solid feature, a profile must be closed. To generate a revolved feature by using an open profile, use the **Surface > Sweep** command.

Why

The **Sweep** command is useful for:

- Creating a bent object with a constant section
- Creating a twisted object with a constant section

How to Use

How to generate a sweep feature using a profile and a path

1. Draw a sketch or a mesh sketch.



2. Go to **Insert > Solid > Sweep** or click the icon in the toolbar.
3. Select a **Profile** and **Path**.



4. Click **OK** to complete the command.



Options in Detail

Select Entities

Profiles – Selects a profile such as a sketch, 3D sketch, or edge that will be used to create a sweep body. The profile should be closed profile. To make a composite profile, select edges or curves with the **Shift** button. A profile should not self-intersect.

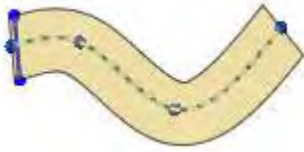
Path – Selects a path for a profile to follow. An opened or closed loop can be used. The starting point of a path must lie on the plane of the profile and a path should not self-intersect.

Methods

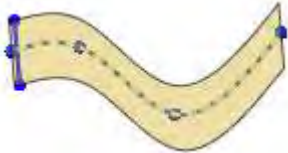
The following methods are available for this command:

- Follow Path
 - Keep Normal Constant
 - Follow Path And First Guide Curve
 - Follow First And Second Guide Curves
 - Twist Along Path
 - Twist Along Path With Normal Constant
-

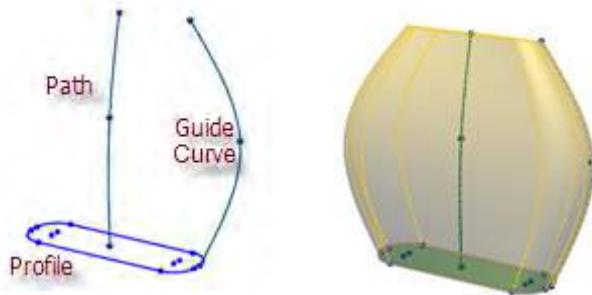
Follow Path – Keeps a profile at the same angle as a path.



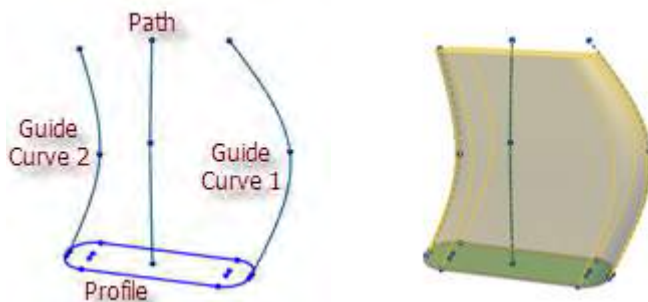
Keep Normal Constant – Keeps a profile parallel to the beginning of a section.



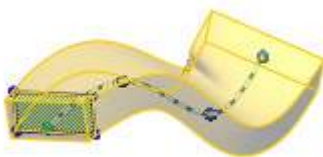
Follow Path and First Guide Curve – Selects a guide curve to control the intermediate shape of a profile. Click the **Curves** button and select a guide curve.



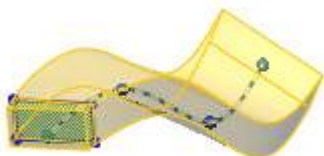
Follow First and Second Guide Curve – Selects two guide curves to control the intermediate shape of a profile. Click the **Curves** button and select guide curves.



Twist Along Path – Twists a section along a path by a specified number of degrees.



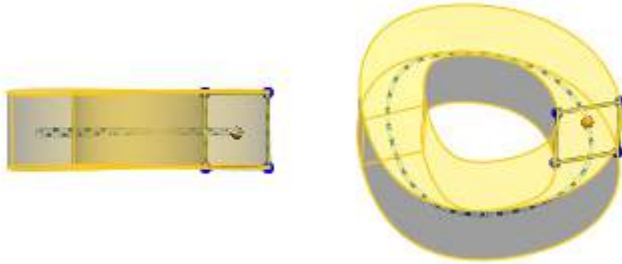
Twist Along Path With Normal Constant – Twists a section along a path keeping the section parallel to the beginning section as it twists along a path.



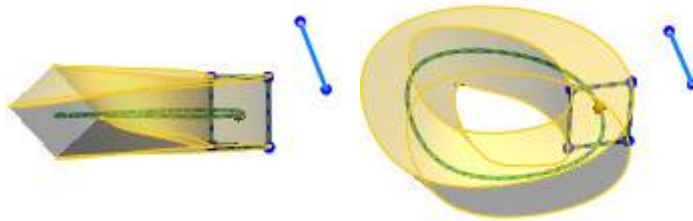
Path Options

Path Alignment Type – Stabilizes a profile when small and uneven curvature fluctuations along a path causes a profile to misalign. This option is available when the **Follow Path** option is selected

None – Aligns a profile normal to a path with no corrections applied.



Direction Vector – Aligns a profile in the direction of a selected linear entity vector.



Guide Curves

Guide Curves – Guides a profile as it sweeps along a path.

Click the **Curves** button and select a guide curve.

Options

Improve Curvature Flow Along Sweep Path – Smoothly connects the curvature flow of faces along a sweep path

Merge Tangent Faces – Merges tangentially connected faces to reduce the total number of faces.



Without Merge Tangent
Faces

With Merge Tangent Faces

Result Operator

The **Result Operator** option is available when one or more bodies exist. The **Boolean** command can be used to cut or merge with other bodies later.



Without Cut and Merge

Cut – Cuts bodies that intersect with a created extruded body.



With Cut Option

Merge – Merges bodies that intersect with a created extruded body.



With Merge Option

See Also

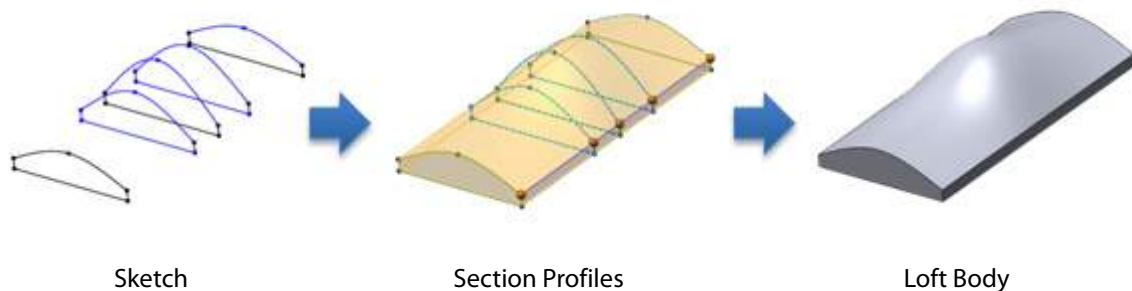
[Insert > Modeling Wizard > Sweep Wizard](#)

[Insert > Surface > Sweep](#)

Insert > Solid > Loft

What

The **Loft** command connects multiple object sections and generates a closed solid body from them. The object sections are expressed by a single profile or multiple profiles on a 2D or 3D sketch and can be drawn by circles, splines, lines and arc sketch entities.



Note:

To generate a lofted solid feature, a profile must be closed. To generate a lofted surface feature using an open profile, see [Surface > Loft](#).

Why

The **Loft** command is useful for:

- Creating an object connected by multiple sections
- Creating different starting and ending profile objects

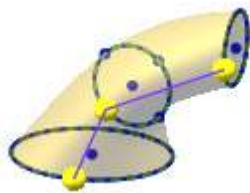
How to Use

How to generate a loft feature from profiles

1. Go to **Insert > Solid > Loft** or click the icon in the toolbar.

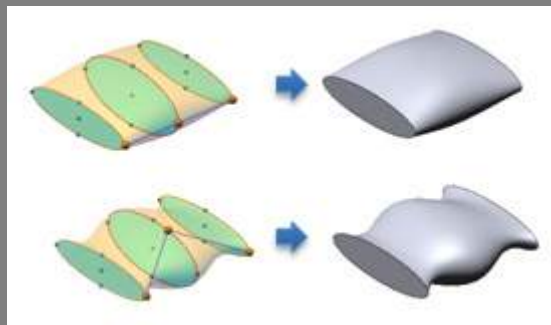


2. Select **Profiles** to loft.





Note:

The alignment between loft profiles can be modified by manipulating the handles (spheres) that appear on the selected profiles as part of the connector. The connector is the poly line connecting corresponding points in both directions.



Note:

The lofting order of profiles in the dialog tree can be changed by using the **Up**  and **Down**  buttons after selecting a profile in the list.



3. Click **OK** to complete the command.



Options in Detail

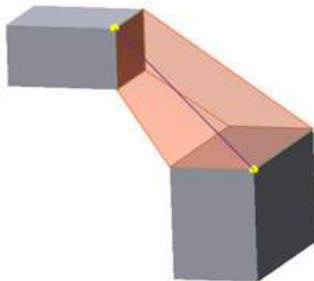
Select Entities

Profiles – Selects a profile such as a sketch, 3D sketch, or edge that will be used to create a sweep body. The profile should be closed profile. To make a composite profile, select edges or curves with the **Shift** button. A profile should not self-intersect.


Constraints

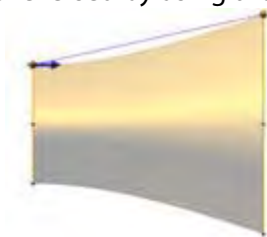
Start Constraint – Selects the starting constraint type.

None – Applies no tangency constraints or zero curvatures.

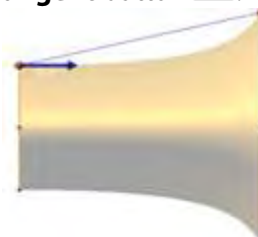


Direction Vector – Applies a tangency constraint based on a selected entity. Click the **Direction Vector** button and select a linear entity such as a vector, line, edge, etc, or a planar entity such as a plane, region, or face.

Tangent Length – Controls the amount of tangency influence on a loft by setting the value or manipulating the blue arrow in the Model View. The tangency direction can be reversed by using the **Flip Tangent** button .

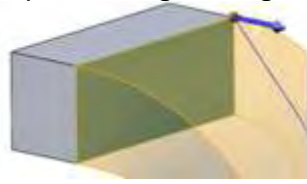


Tangent Length: 1

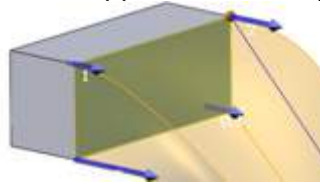


Tangent Length: 2

Use Same Tangent Length To All Vertices - Applies the same tangency to all vertices on a profile. Tangent length can also be applied individually on each vertex.



Use Same Tangent Length



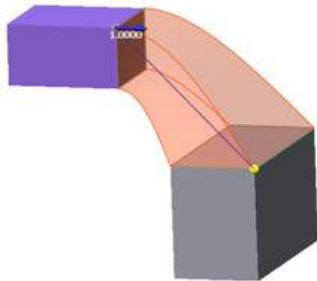
Individual Tangent Length

Normal To Profile - Applies a tangency constraint normal to a starting profile.

Tangent Length – Refer to the **Direction Vector** option.

Use Same Tangent Length To All Vertices – Refer to the **Direction Vector** option.

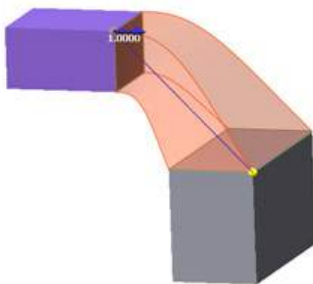
Tangency To Face – Creates a tangency constraint that follows a selected face and is available if a face is selected as a profile. Clicking the **Next Face** button will find the next candidate face.



Tangent Length – Refer to the **Direction Vector** option.

Use Same Tangent Length To All Vertices – Refer to the **Direction Vector** option.

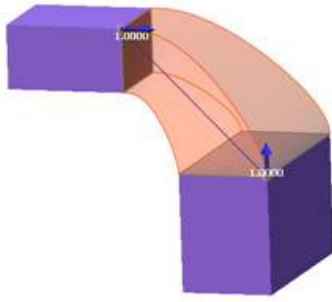
Curvature To Face – Creates a curvature constraint that follows a selected face and is available if a face is selected as a profile. Clicking the **Next Face** button will find the next candidate face.



Tangent Length – Refer to the **Direction Vector** option.

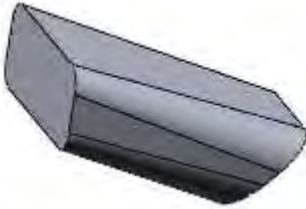
Use Same Tangent Length To All Vertices – Refer to the **Direction Vector** option.

End Constraint – Selects the ending constraint type. The sub-options are the same as the **Start Constraint** options.



Guide Curves

Guide Curves – Selects a guide curve to control the intermediate shape of a sweep body.



Without Guide Curve



With Guide Curve

Options

Merge Tangent Faces – Merges tangentially connected faces to reduce the total number of faces.

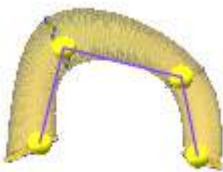


Without Merge Tangent Faces

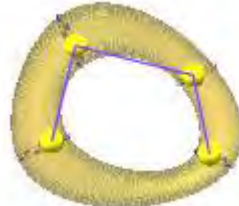


With Merge Tangent Faces

Close Loft – Creates a closed body along a loft direction.



Without Close Loft



With Close Loft

Result Operator

The **Result Operator** option is available when one or more bodies exist. The **Boolean** command can be used to cut or merge with other bodies later.



Without Cut and Merge

Cut – Cuts bodies that intersect with a created extruded body.



With Cut Option

Merge – Merges bodies that intersect with a created extruded body.



With Merge Option

See Also

[Insert > Modeling Wizard > Loft Wizard](#)

[Insert > Surface > Loft](#)

Insert > Solid > Offset

What

The **Offset** command enlarges or shrinks a solid body with a constant distance.



Why

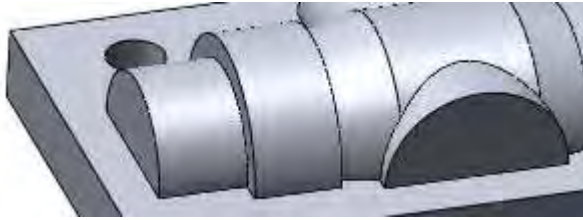
The **Offset** command is useful for:

- Creating a model in a different size

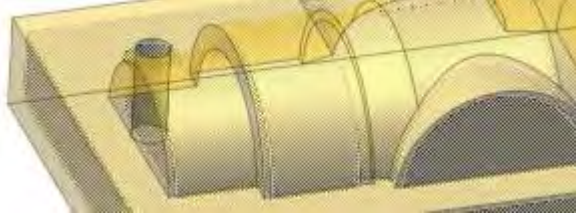
How to Use

How to offset a solid body

1. Go to **Insert > Solid > Offset** or click the icon in the toolbar.



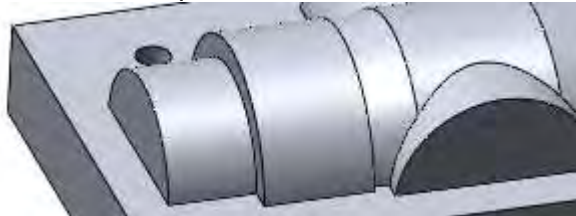
2. Select a solid body and specify the **Offset** distance.



Note:

To reverse the offset direction, use the **Flip Direction** button .

3. Click **OK** to complete the command.



Options in Detail

Bodies – Selects target solid bodies to offset.

Offset Distance – Specifies the distance to offset from a solid body. To reverse the offset direction, use the **Flip Direction** button.



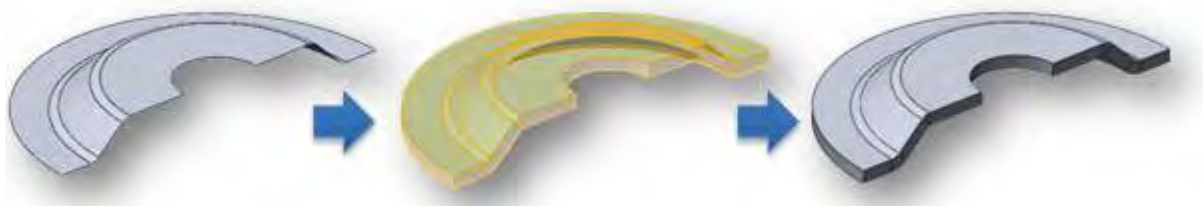
See Also

[Insert > Surface > Offset](#)

Insert > Solid > Thicken

What

The **Thicken** command offsets a surface body and generates a solid body with a constant thickness.



Why

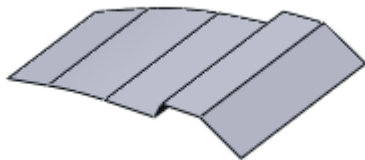
The **Thicken** command is useful for:

- Creating an object with constant thickness
- Creating a sheet metal part

How to Use

How to generate a constant wall thickened feature from a surface body

1. Prepare a surface body.

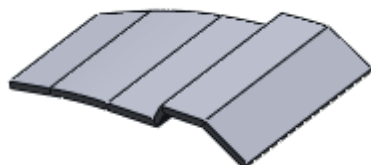


2. Go to **Insert > Solid > Thicken** or click on the icon in the toolbar.

3. Select a surface body to thicken. Enter **3 mm** as the **Thickness** and set the offset direction.



4. Click **OK** to complete the command.



Options in Detail

Select Entities

Body– Selects a surface body.

Note:

When a surface body is comprised of multiple adjacent faces, the surfaces must be sewn together before applying the **Thicken** command.

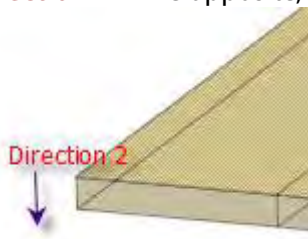
Thickness – Specifies a thickness value.

Direction

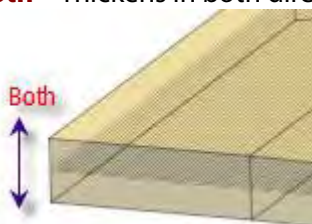
Direction 1 – The default positive direction to thicken.



Direction 2 – The opposite, negative direction from **Direction 1**.



Both – Thickens in both directions.



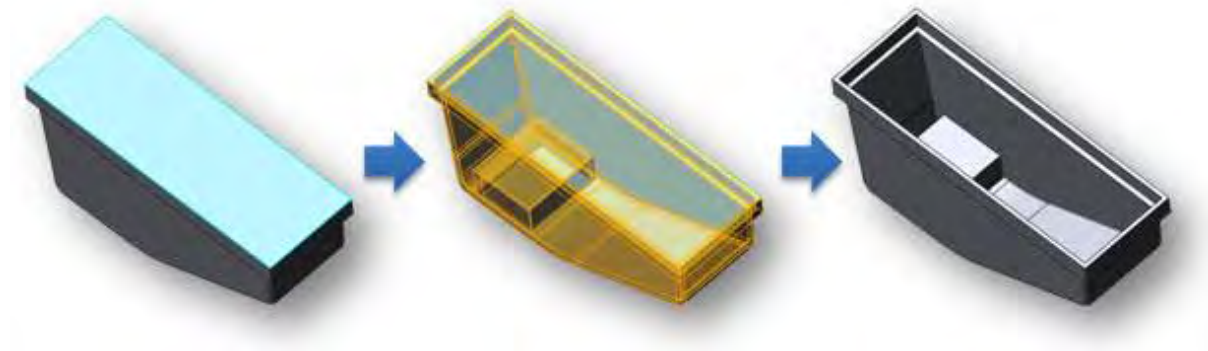
See Also

[Insert > Solid > Hollow](#)
[Insert > Surface > Offset](#)

Insert > Solid > Hollow

What

The **Hollow** command removes selected faces from a solid body and generates a constant thin-walled model with the remaining faces.



Why

The **Hollow** command is useful for:

- Creating a constant thickness object
- Creating an injection molded part

How to Use

How to generate a constant wall thickened feature from a solid body

1. Prepare a solid body.



2. Go to **Insert > Solid > Hollow** or click on the icon in the toolbar.
3. Select a body to **Hollow** and a bottom face for **Remove Faces**. Specify a **Depth**.



4. Click **OK** to complete the command.

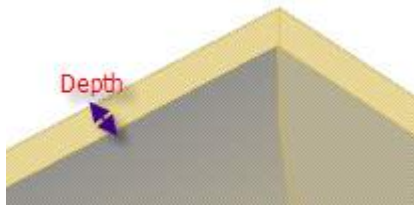


Options in Detail

Select Entities

Body – Selects a body to hollow.

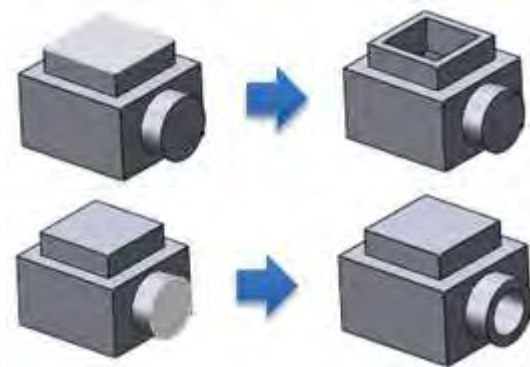
Depth – Specifies a global wall-thickness of a hollow feature.





Hollow Outward – Applies hollowing to the outward direction from a solid body.

Remove Faces

Remove Faces – Selects faces to remove.



Multiple Thickness

Multi-Thickness Faces – Specifies multiple thicknesses for each face by adding a face using the **Add Face** button  and setting its depth. To remove the last added face setting, click the **Remove Face** button .

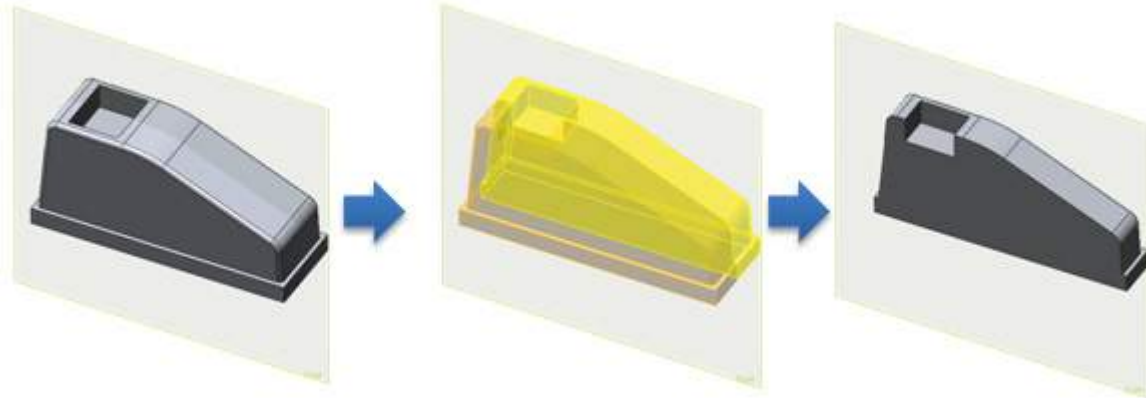
See Also

[Insert > Solid > Thicken](#)

Insert > Solid > Cut

What

The **Cut** command cuts a solid body by removing material with a surface or plane. The remaining material can be chosen manually.



Why

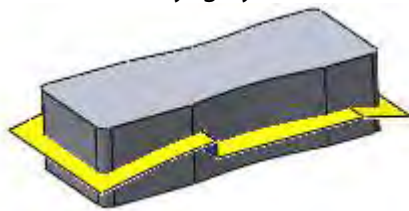
The **Cut** command is useful for:

- Creating a feature cut using a surface or plane
- Dividing an object

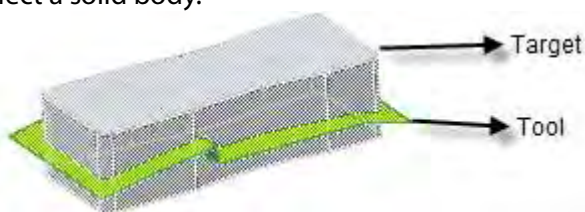
How to Use

How to cut a solid body with a surface body

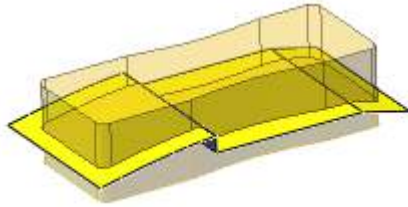
1. Prepare a solid body (gray color) and a surface body (yellow color).



2. Go to **Insert > Solid > Cut** or click the icon in the toolbar.
3. Click the **Tool Entities** button and select a surface body. Click the **Target Bodies** button and select a solid body.



4. Click the **Next Stage** button and select a remaining body.



5. Click **OK** to complete the command and hide the surface body.



Options in Detail

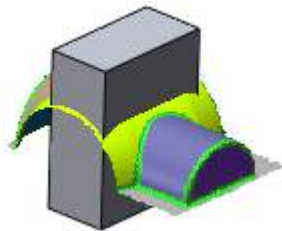
Select Entities

Tool Entities – Selects entities as tool entities to cut a solid body. A ref. plane, region, sketch, sketch chain, curve, 3D sketch, edge and surface body can be selected as tool entities.

Target Bodies – Selects target bodies. Only solid bodies can be selected as target bodies.

Result

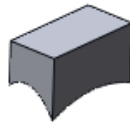
Remaining Bodies – Select bodies to remain after cutting.



Solid Body (Target) and Regions (Tool)



Remaining Body: Bottom



Remaining Body: Top

See Also

[Insert > Solid > Boolean](#)

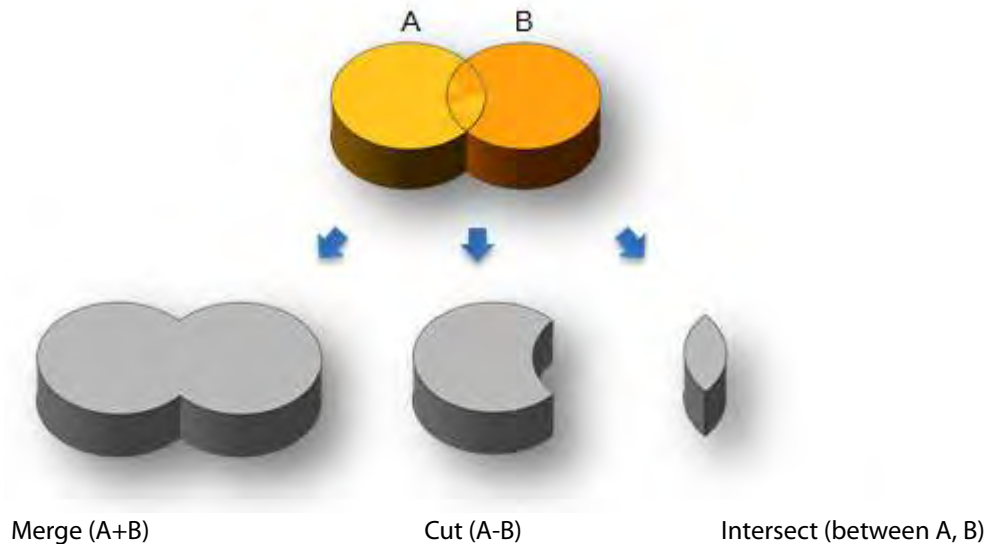
[Insert > Surface > Trim](#)

[Insert > Surface > Trim & Merge](#)

Insert > Solid > Boolean

What

The **Boolean** command combines two or more solid bodies to generate a single body or multiple bodies by using one of three methods: **Merge**, **Cut** and **Intersect**.



Note:

The cut and merge methods can be used while generating a solid body. When a solid body already exists and a new solid body is generated using the **Extrude**, **Revolve**, **Sweep** or **Loft** commands, the **Result Operator** option will appear at the bottom of the command dialog tree.

Why

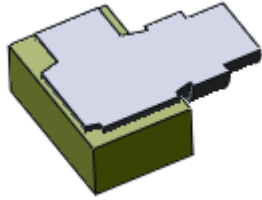
The **Boolean** command is useful for:

- Combining solid bodies
- Splitting solid bodies
- Obtaining a shared area between overlapped solid bodies

How to Use

How to create a negative solid body

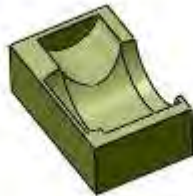
1. Prepare two overlapped solid bodies.



2. Go to **Insert > Solid > Boolean** or click the icon in the toolbar.
3. Choose **Cut** as the **Operation Method**. Select bodies as **Tool Bodies** and **Target Bodies**.



4. Click **OK** to complete the command.



Options in Detail

Methods

Merge – Combines all selected bodies, creating a single body.

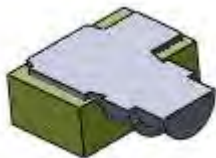


Before Merge



After Merge

Cut – The tool bodies remove overlapping material from the target bodies.

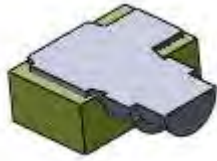


Before Cut



After Cut

Intersect – Removes all material between the tool and target bodies except the area of shared material.



Before Intersect



After Intersect

7.4.7. Insert > Surface

The **Insert Surface** menu features various commands used to generate and edit surfaces. The **Surface** menu features functions such as **Surface Primitives**, **Mesh Fit**, **Boundary Fit**, **Auto Surfacing**, **Extrude**, and, **Revolve**.

Generating Surface Bodies by Fitting a Mesh

- **Mesh Fit**
Generates fitting surfaces based on regions of a mesh
- **Boundary Fit**
Generates a fitting surface inside of a curve network based on a mesh
- **Auto Surfacing**
Generates a curve network and a fitting surface based on a mesh

Generating Surface Bodies from Sketches

- **Extrude**
Extrudes bodies using a sketch, direction and length
- **Revolve**
Revolves bodies using a sketch, axis and revolution angle
- **Sweep**
Sweeps bodies using a sketch and path
- **Loft**
Lofts bodies using a set of sketches

Generating Surface Bodies from Other Entities

- **Offset**
Offsets faces from existing body faces

- **Fill Face**
Creates surfaces inside of boundary curves or edges
- **Trim & Merge**
Creates a solid body by completely cutting a material with other surfaces

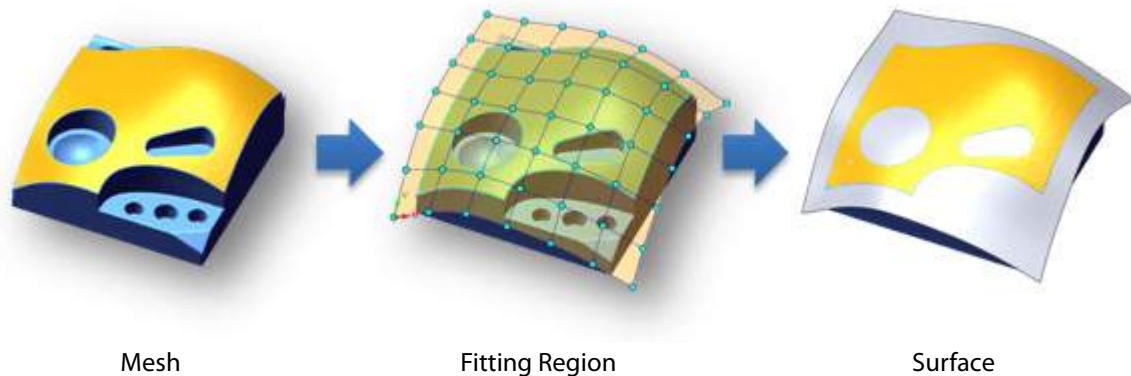
Editing Surface Bodies

- **Extend**
Creates a surface body by extending existing bodies
- **Trim**
Creates a surface body by cutting surfaces
- **Untrim**
Restores a surface previously trimmed by other surfaces or curves
- **Sew**
Merges two or more adjacent surfaces
- **Reverse Normal**
Flips surfaces

Insert > Surface > Mesh Fit

What

The **Mesh Fit** command generates surfaces by using a fitting algorithm based on a mesh. Surface Fitting Technology is a unique technique in the Reverse Design process that provides an effective way to easily and quickly create a 3D freeform surface body from a freeform mesh shape.



Why

The **Mesh Fit** command is useful for:

- Creating a smooth surface from a coarse mesh surface
- Creating a freeform surface (not analytic shape) on a part
- Creating an aesthetic model

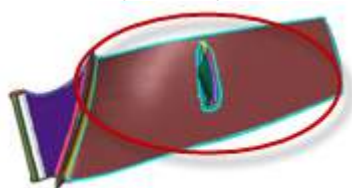
How to Use

How to generate freeform surface on a mesh

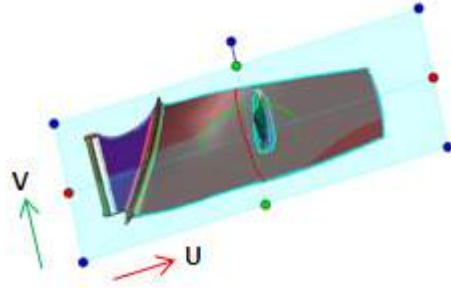
1. Prepare a mesh with regions.



2. Go to **Insert > Surface > Mesh Fit** or click the icon in the toolbar.
3. Select a region to generate a fitting surface.



4. Change the **Resolution** option to **By No. Of Control Points** to define U and V directional control points on the surface. Set the **U** control points to **5** and **V** control points to **10**.

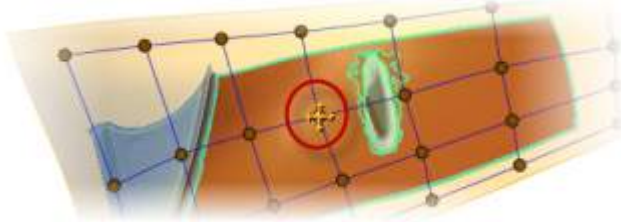


points to **10**.

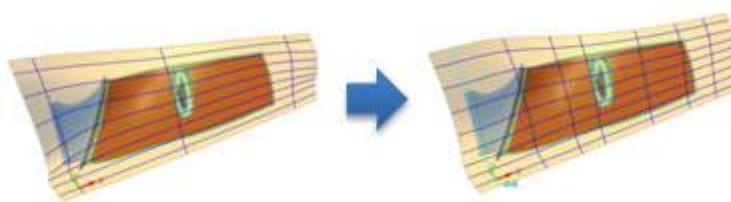
Note:

The preview has controllers at each edge and corner. They can be resized and rotated by dragging them to completely cover the target region.

5. Click the **Next Stage** button.
6. In the second stage, the flow of iso-lines can be changed. Set the **Control Net Density** option and deform controllers on the preview to enhance the iso-line flow.

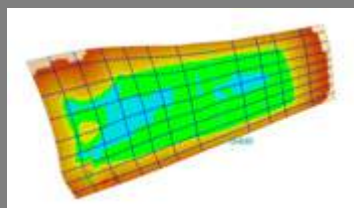


7. Click the **Next Stage** button.
8. In the third stage, the density of iso-lines can be changed. Create additional iso-lines on high curvature areas by clicking an existing iso-line while holding down the **Ctrl** key to create a copy of that line and dragging it to a desired location.

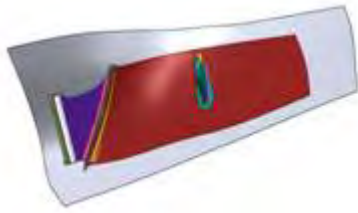


Note:

When the **Deviation** option is used in the **Accuracy Analyzer™** panel, deviations can be checked in real-time whenever iso-lines are modified.



- Click **OK** to complete the command. The fitting surface will be created.



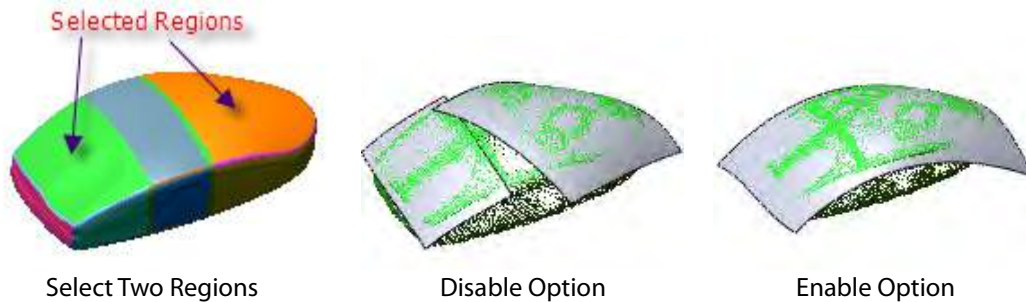
Options in Detail

Stage 1/3: Default Setting

This stage defines the default settings for generating a fitting surface such as target areas on a mesh, resolution, fitting options and surface extension options.

Regions / Poly-Faces – Selects target regions or poly-faces on a mesh to generate a fitting surface.

Create Single Surface – Selects multiple regions to create a single surface.



Select Two Regions

Disable Option

Enable Option

Resolution – Controls resolution to determine the overall accuracy and smoothness of a fitting surface.

By Allowable Deviation – Sets the resolution of a fitting surface by the deviation between the mesh and the fitting surface. Use this option when deviation is the most important criteria of a fitting surface.

Note:

When a region classified as a primitive feature such as a plane, cylinder, cone or sphere is selected as a target region and the resolution option is defined as **By Allowable Deviation**, a primitive surface will be generated as a fitting result. For example, when a conical region is selected as a target region and the resolution option is defined as **By Allowable Deviation**, the fitting result will be a conical surface body.



Conical Target Region



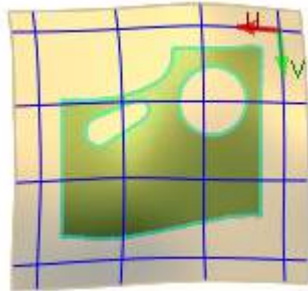
Created Conical Surface Body

Allowable Deviation – Specifies the allowable deviation distance between a mesh and a fitting surface.

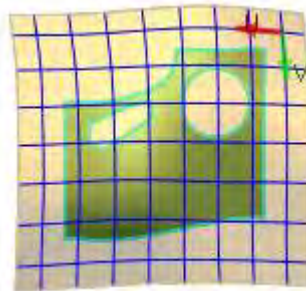


Outlier % For Allowable Deviation – Sets the percentage value used to remove outliers from a mesh. The value can be set under the **Tolerance** tab in **File > Preference**.

Maximum No. Of Control Point – Creates control points by specifying a maximum number. The number relates to the iso-lines in the third stage.

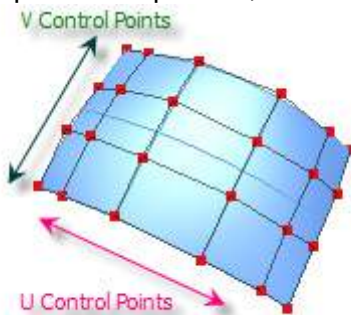


Max. Number: 6

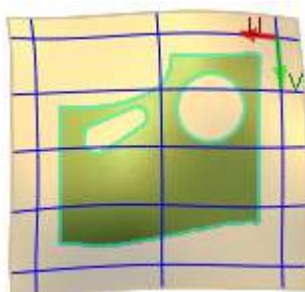


Max. Number: 10

By No. Of Control Points – Specifies the number of control points for the U and V directions which control the resolution of a fitting surface. When a large number of control points are specified, deviation can be minimized but smoothness may also be lost.

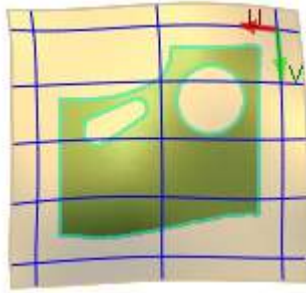


No. Of U Control Points – Specifies the number of control points in the **U** direction on a surface.



U Number: 5

No. Of V Control Points – Specifies the number of control points in the **V** direction on a surface.



V Number: 7

Smoothness – Determines the smoothness level of a fitting surface.



Smoothness: Less

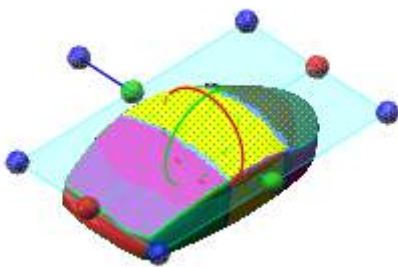


Smoothness: More

Resample Mesh – Creates regular iso-lines of a fitting surface. This option may generate twisted or poorly fitted surfaces on complicate shapes or multiple regions.



Manipulate U-V Axes – Enables the red or green controllers used to resize a fitting area in the U or V directions. The blue controller is used to resize both the U and V directions at the same time. The handle is used to rotate a fitting area.

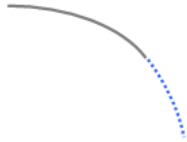


Extend – Extends a fitting surface.

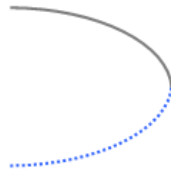
Linear – Linearly extends a surface from the original fitting surface.



Curvature – Extends a surface by maintaining curvature from the original fitting surface.



Same Surface – Extends a surface by mirroring the original fitting surface.



U Extend Ratio – Specifies the extension ratio in the **U** direction.

V Extend Ratio – Specifies the extension ratio in the **V** direction.

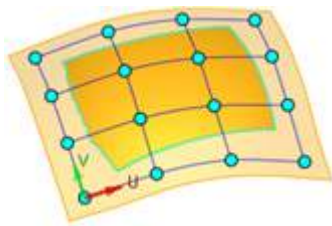
Stage 2/3: Iso-Line Flow Control

This stage controls the flow of Iso-lines which determines the quality of a fitted surface. To make Iso-lines that follow the flow of a surface on a mesh is the purpose in this stage, however, excessive editing may generate twisted fitted surfaces.

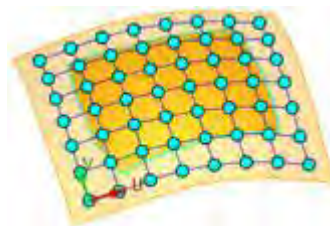
Reset Manipulator – Clears all changes made to manipulators.



Control Net Density – Determines the density of control points for manipulators. A lower density, will allow for control of the overall iso-line flow and a higher density will better control iso-line flow in detail.



Control Density: Less



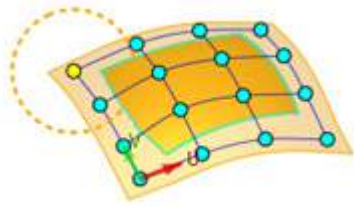
Control Density: More

Note:

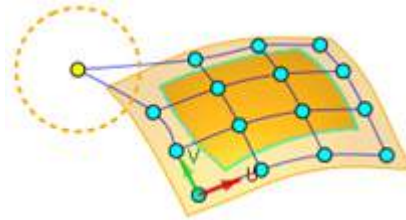
The control net density does not have relationship with the iso-line number. The number of iso-lines is determined by the **Resolution** option in the first stage.

Control Degree Of Deformation – Controls points individually. The editing zone can be resized by holding down the **Alt** key and dragging with the left-mouse button.

Fix Boundary Points – Prevents control points from moving over a boundary.



With Fix Boundary Points



Without Fix Boundary Points

Stage 3/3 Iso-Line Density Control

In the third stage iso-lines can be moved, added and deleted to improve surface quality and fitting accuracy. Additional iso-lines can be added on high curvature areas to satisfy desired fitting deviations. On lower curvature areas, iso-lines can be removed or moved if a few curves are enough to express the flat area.

The following actions can be applied to iso-lines:

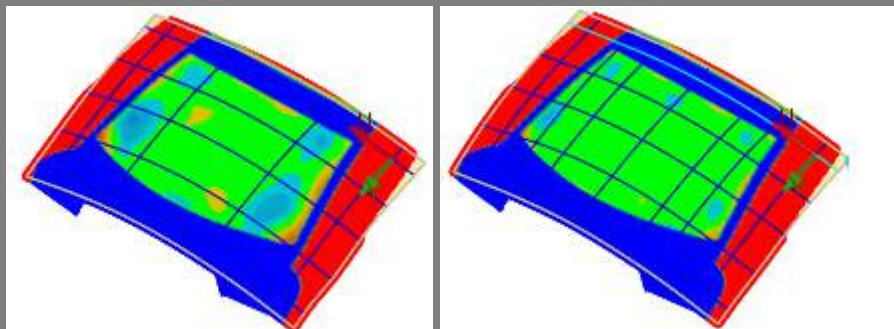
Move – Select and drag.

Add – Hold down the **Ctrl** key and drag.

Delete – Select and press the **Delete** key.

Note:

While manipulating control curves, the **Accuracy Analyzer™** can be used. The image below shows real-time deviation maps as control curves are adjusted.



See Also

[Insert > Surface > Boundary Fit](#)

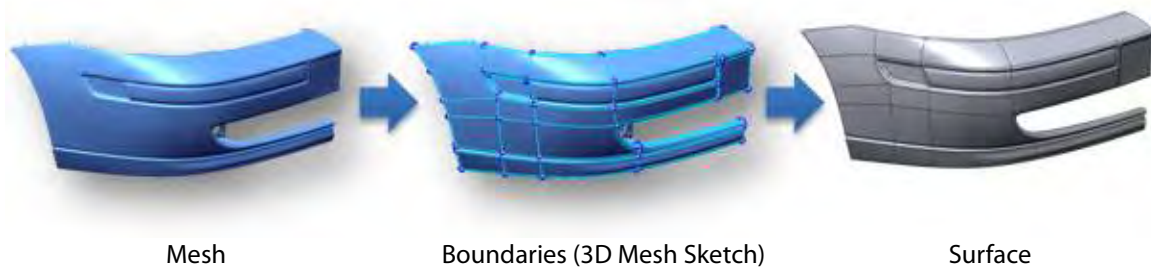
[Insert > Surface > Auto Surfacing](#)

Insert > Surface > Boundary Fit

What

The **Boundary Fit** command generates surfaces inside of boundaries using a fitting algorithm based on a mesh. The boundaries are defined by a 3D spline curve network in the **3D Mesh Sketch** mode. Surface Fitting Technology is a unique technique in the Reverse Design process that provides an

effective way to easily and quickly create a 3D freeform surface body from a freeform mesh shape.



Why

The **Mesh Fit** command is useful for:

- Creating CAD-like fitting surfaces
- Creating higher quality fitting surfaces than auto surfacing
- Quickly making CAD data from a mesh

How to Use

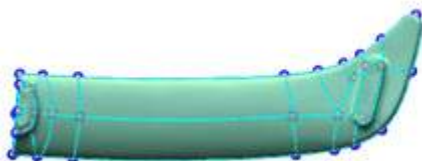
How to generate fitting surface using a curve network

1. Prepare a mesh and curve network.



2. Go to **Insert > Surface > Boundary Fit** or click the icon in the toolbar.

3. Select all curves or select the 3D Sketch (Mesh) under the Feature or Model Trees as Mesh **Curves**.



Note:

Clicking the **Curve Loops** button and picking loops on the curve network will remove them from a fitting surface. Selecting a curve again will re-add it.



4. Click the **Next Stage** button. Red coloring represents the U direction and the green color represents the V direction.

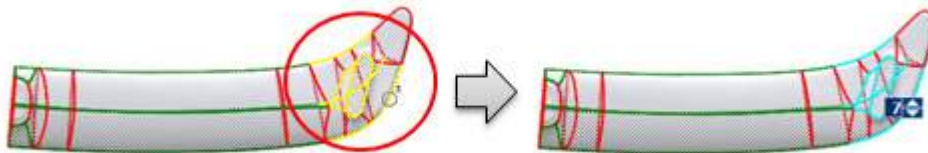


5. Set the surface resolution method to **By Allowable Deviation** or **By No. Of Control Points**.

If **By No. Of Control Points** is selected then set the values for the U and V points.



6. To change control point numbers locally, click the **Set Manually** button, and select a curve to change and set the value in the box that appears near the curve.



7. Click **OK** to complete the command.



Note:

The quality of a boundary fit surface is directly affected by the mesh quality and curve network.

To improve mesh quality:

Remove errors in the mesh such as folded, dangling, small, non-manifold and crossing poly-faces. See [Tools > Mesh Tools > Healing Wizard](#) (hyper link)

Remove unnecessary features from the mesh. See

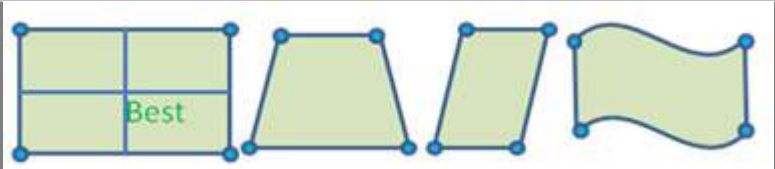
Remove unnecessary holes. See

Improve the balance of poly-faces. See

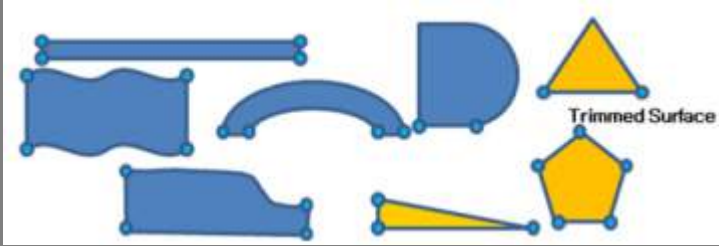
To optimize a curve network:

Create a curve network that equalizes the formation of the structure.

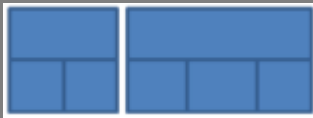
A rectangular feature with four similar curve boundaries will result in a better fitting surface.



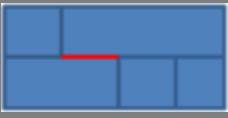
If the shape of a feature has a sudden narrow area or is twisted then a twisted and self-intersected face could be created when the surface is generated.



Avoid creating T-Junctions when creating curve networks.



No T-Junctions



Undesired T-Junction

Anything except a 4 sided curve boundary will create a trimmed fitting surface.

Adjust the same number of control point in each matching boundary for surface patches to improve surface matching continuity.

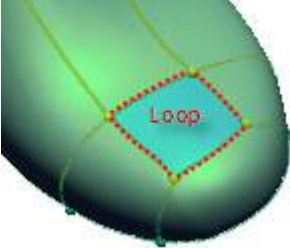
Set a curve boundary using appropriate control point.

Options in Detail

Stage 1/2: Select Entities and Loop Options

Mesh Curves – Selects a curve network.

Curve Loops – Selects loops in a curve network.



Use Mesh Normal To Evaluate Loop Winding – Evaluates a mesh normal to decide a loop to be filled when boundaries create two loops.

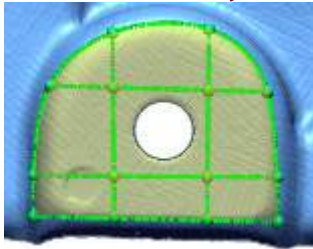


Enable Option

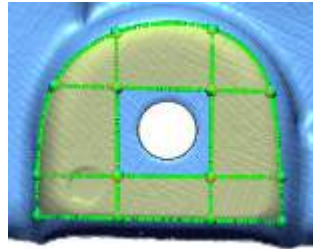


Disable Option

Allow Hole (Boundary) – Fills a loop that includes holes with a trimmed surface.

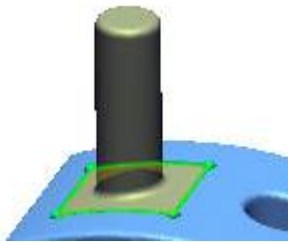


Enable Option

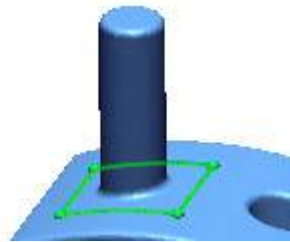


Disable Option

Allowable Convex Ratio – Sets the ratio for an allowable convex or concave ratio for a loop used to wrap a feature between a loop and inside a mesh shape.



Ratio: 10: possible to wrap the feature



Ratio: 2: impossible to wrap the feature

Stage 2/2: Control Surface Quality

Resolution – Controls resolution to determine the overall accuracy and smoothness of a fitting surface.

By Allowable Deviation – Sets the resolution of a fitting surface by the deviation between the mesh and the fitting surface. Use this option when deviation is the most important criteria of a fitting surface.

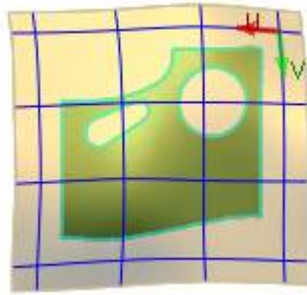
Allowable Deviation – Specifies the allowable deviation distance between a mesh and a fitting surface.



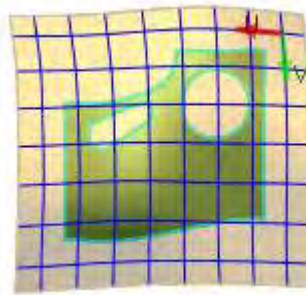
Outlier % For Allowable Deviation – Sets the percentage value used to remove

outliers from a mesh. The value can be set under the **Tolerance** tab in **File > Preference**.

Maximum No. Of Control Point – Creates control points by specifying a maximum number. The number relates to the iso-lines in the third stage.

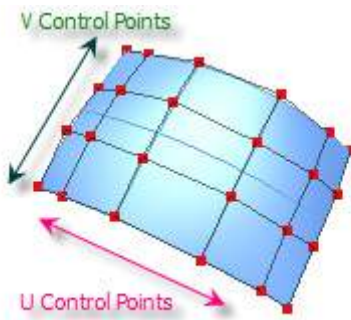


Max. Number: 6

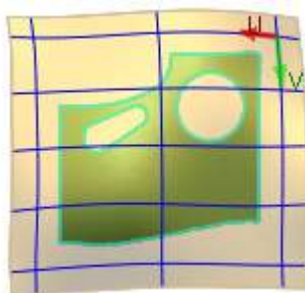


Max. Number: 10

By No. Of Control Points – Specifies the number of control points for the **U** and **V** directions which control the resolution of a fitting surface. When a large number of control points are specified, deviation can be minimized but smoothness may also be lost.

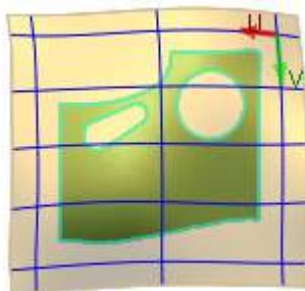


No. Of U Control Points – Specifies the number of control points in the **U** direction on a surface.



U Number: 5

No. Of V Control Points – Specifies the number of control points in the **V** direction on a surface.



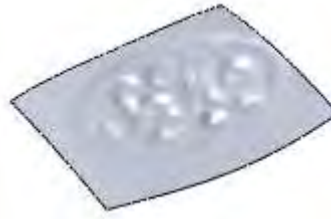
Set Manually – Changes control point values locally on a curve network. Click the **Set Manually** button, select a curve to change, and set a value in the box that will appear near it.



Smoothness – Determines the smoothness level of a fitting surface. Moving the slider towards **More** will create a smoother surface but can diminish accuracy.

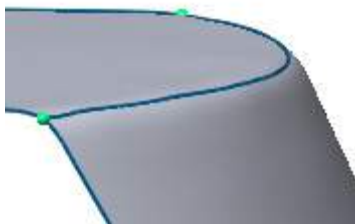


Smoothness: Less



Smoothness: More

Set Sharp Edge – Removes the tangent constraint between adjacent surfaces of selected curves to create a sharp edge.

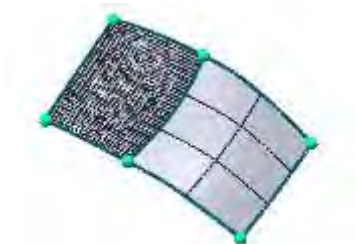


Disable Sharp Edge

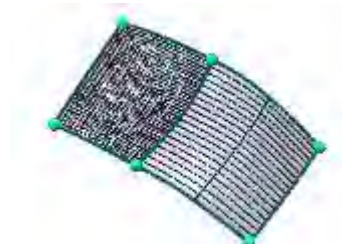


Enable Sharp Edge

Regularize Overall Control Point Count – Attempts to propagate the control point number to a neighboring surface and is only available if the **By Allowable Deviation** method is selected.

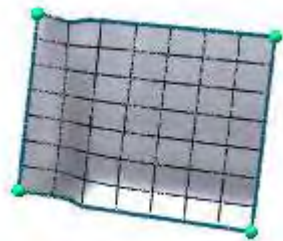


Disable Option



Enable Option

Resample Mesh – Creates iso-lines of equal values and works best for simple surfaces.



See Also

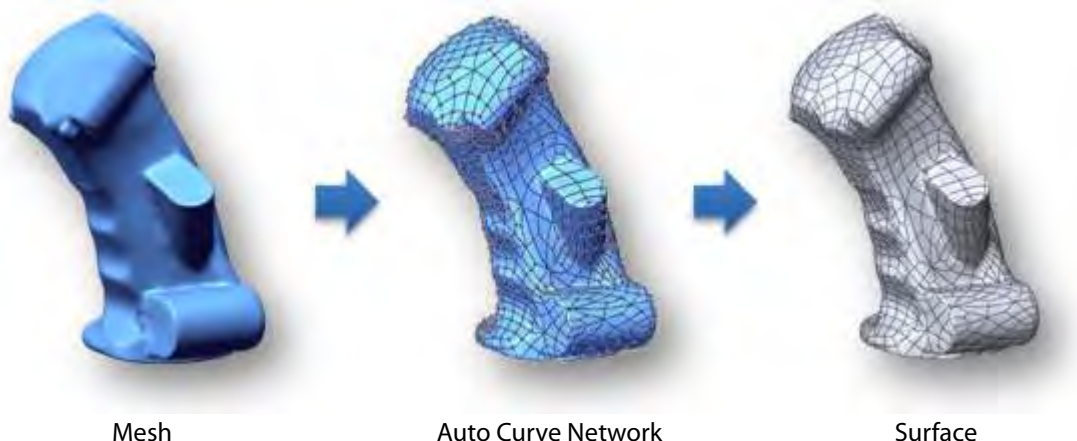
[Insert > Surface > Mesh Fit](#)

[Insert > Surface > Auto Surfacing](#)

Insert > Surface > Auto Surfacing

What

The **Auto Surfacing** command generates fitting surfaces that mathematically fit CAD surfaces to mesh. During the auto surfacing process the application will automatically create a curve network that envelops the entire mesh and fills fitting surfaces for each patch of the curve network.



Mesh

Auto Curve Network

Surface

This command is available if a mesh is already imported or has been generated from a point cloud. As the fitting surface is affected by the quality of a mesh, it should be pre-processed without erroneous poly-faces.

Note:

When a mesh is modified after generating auto surfaces, the surfaces will be updated in the **Feature Tree** but may take time to rebuild.

Why

The **Auto Surfacing** command can be used to:

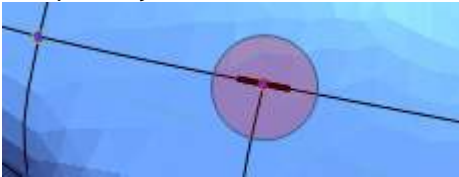
- Generating surfaces from a mesh
- Assisting in surfacing a complicated part
- Defines surface models that cannot be defined in CAD such as organic or natural objects

How to generate quality auto surfaces

The quality of the **Auto Surfacing** command directly depends on the mesh and curve network. The following guidelines can be utilized to successfully generate auto surfaces:

- Remove sharp patches
- Remove T-junctions*
- Maintain similar mesh densities
- Remove noise around boundaries by using the Tools > Mesh Tools > Edit Boundaries command
- Fill holes
- Split patches to make the size of patches standard

***T-junctions** are junctions where one spline joins another but does not cross it and is marked by thick lines at intersecting node such as the image below. Surface matching around T-junction may not be satisfactory for auto surfacing, so it is better to remove them or add a spline to make a completed junction.



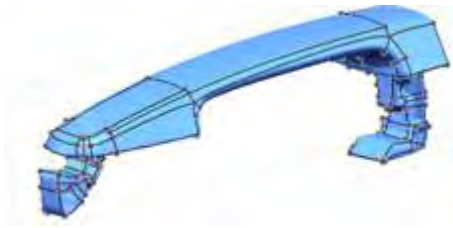
How to use Auto Surfacing command

A. How to generate CAD like auto surfaces

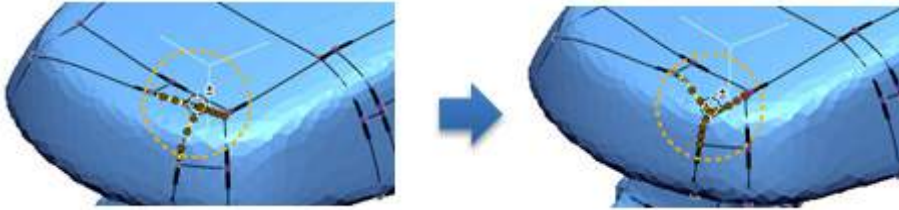
1. Go to **Insert > Surface > Auto Surfacing** or click the icon in the toolbar.



2. Choose the **Feature Following Network** option.
3. Adjust the slider for the **Level of Patch Complexity** option to the middle.
4. Click the **Next** button. The curve network will be modified in the second stage.



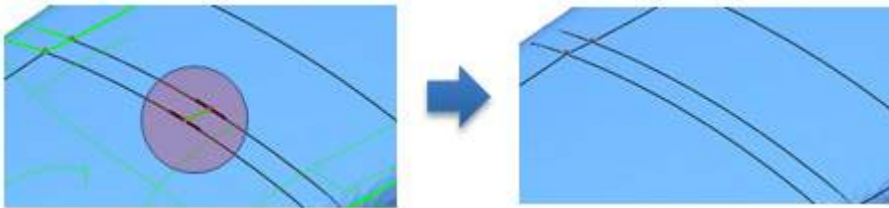
5. A node position can be relocated by using the **Deform** option and click-dragging node points.



Note:

To resize the editing zone represented by circle comprised of yellow dots, press the **Alt** key and move the mouse left or right while holding down the left-click button.

6. To remove a spline from a T-junction, choose the **Remove** option and removable splines will be highlighted in green.



7. Click **OK** to complete the command.



B. How to generate auto surfaces on an organic model

1. Go to **Insert > Surface > Auto Surfacing** or click the icon in the toolbar.



2. Choose the **Evenly Distributed Network** option.
3. Specify the total number of surface for the **No. Of Surfaces** option. In this example **300** is used.

4. Click the **Next** button. The curve network will be modified in the second stage.



5. Click **OK** to complete the command.



Options in Detail

Stage 1

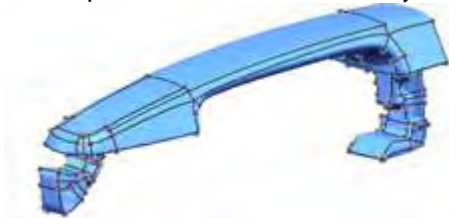
Mesh – Select a mesh.

Methods


Evenly Distributed Network – Generates a curve network with evenly sized patches. This method can apply to any object shape, especially those that cannot be defined as CAD objects like organic or natural objects. To generate a CAD like network, use the **Feature Following Network** option.



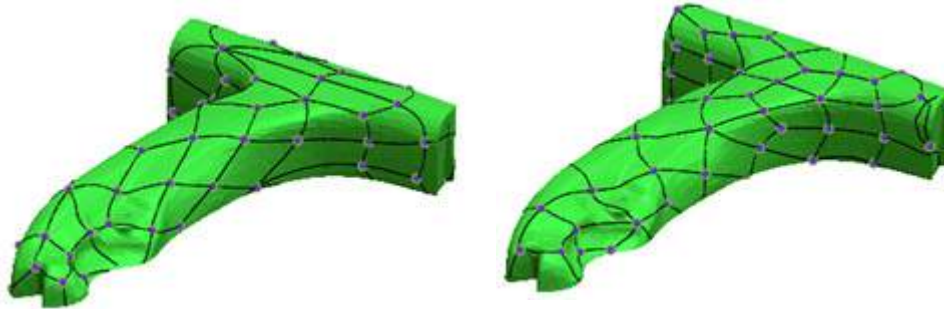
Feature Following Network – Follows character lines on high curvature areas of a mesh. This method creates a smaller number of patches than the **Evenly Distributed Network** method and expresses the model similarly to CAD data.



Patch Network Options

No. of Surfaces – Controls the total number of patches in a network. A higher value more accurately represents a mesh but increases the number of patches and the file size. The **Estimate** button  will automatically recommend a value.

Patch Size Regularity – Controls the regularity of patches. When the slider is moved to **Adaptive**, the application will generate an appropriate patch shape around areas of high curvature and when the slider is moved to **Uniform**, patches will be regular size regardless of the curvature area.

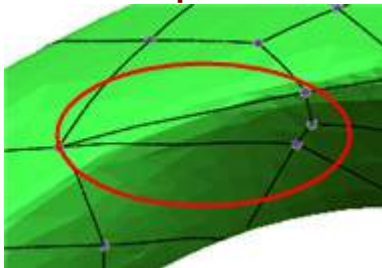


Adaptive

Uniform

Hole Filling By Area – Generates fitting surfaces when small holes that normally disturb the creation of fitting surfaces exist in a mesh. When the slider is moved to **Large**, holes with areas up to 20% of the entire mesh areas will be filled.

Remove Sharp Patches – Removes sharp patches in a curve network.



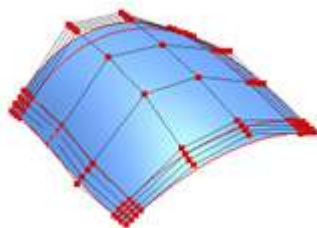
Relax Initial Mesh – Internally modifies a mesh to make regular size poly-faces and generates patches with lines that are close to 90 degree angles.

Level of Patch Complexity – Adjusts the level of patch complexity in a mesh using the slider. The number of patches will be increased by adjusting towards "More".

Optimize Patch Network – Minimizes irregular patches such as narrow and long patches.

Fitting Options

No. of Control Points / Surface – Determines the number of control points along the U and V directions for each surface. The values are simultaneously changed and the control points are internally set and cannot be displayed. The higher the number of control points, the more degrees of freedom for changing the shape and accurately following areas of high curvature areas but at the loss of surface smoothness.



Control Points

Smoothness – Controls the smoothness of a fitting surface. A strong smoothness will remove

spikes and wrinkles but will also lose fitting accuracy.

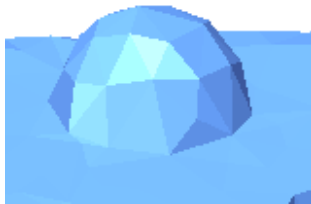


Smoothness: Less



Smoothness: More

Geometry Capture Accuracy – Defines geometry capture accuracy. When the slider is moved towards **Loose**, less poly-faces will be used to create a shape. When the slider is moved towards **Tight**, more poly-faces will be used to create a shape.



Loose



Tight

Remove Self-intersecting Patches – Remove self-intersecting patches.

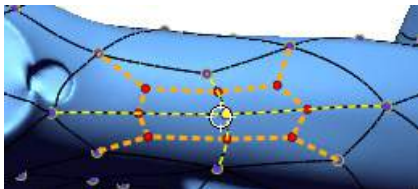
Stage 2 of Evenly Distributed Network

The following editing methods appear when the **Evenly Distributed Network** method is chosen in Stage 1. You also can quickly access the editing methods from the pop-up menu.

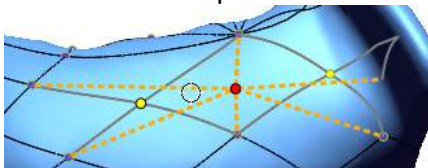
Deform Curve Network (D) – Deforms a preview network by moving a node of a curve within an editing zone. To adjust the editing zone, hold down the **Alt** key and left-click and drag the mouse left or right. The shortcut key of this method is **D**.



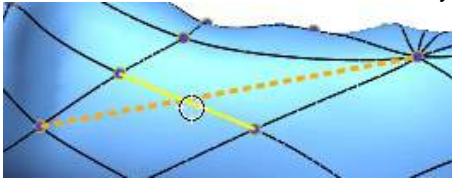
Subdivide Vertex (S) – Previews subdivided curves when the mouse positions on an intersection node. The shortcut key of this method is **S**.



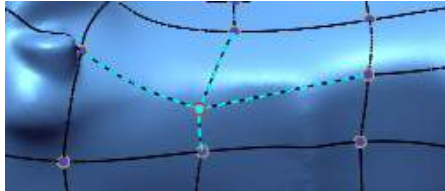
Merge Vertex (G) – Previews merged curves to help reduce the number of network curves when the mouse positions inside of a network area. The shortcut key of this method is **G**.



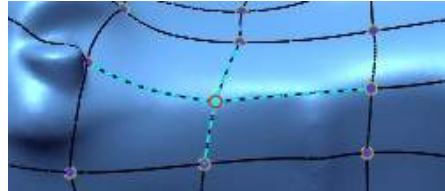
Rotate Edge (R) – Previews rotated curves to flip an edge direction when the mouse positions on a network curve. The shortcut key of this method is **R**.



Relax Patches – Smooths entire patches.



Before Relax Patches



After Relax Patches

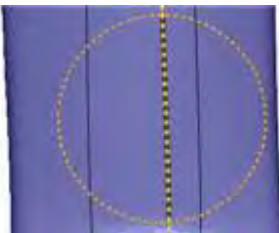
Stage 2 of Feature Following Network

The following editing methods appear when the **Feature Following Network** method is chosen in **Stage 1**. You also can quickly access the editing methods from the pop-up menu.

Deform (D) – Deforms a spline by moving a node of a curve within in an editing zone. To adjust the editing zone, hold down the **Alt** key and left-click and drag the mouse left or right. The shortcut key of this method is **D**.

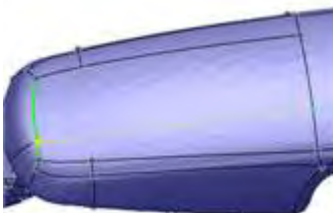


Before Deform

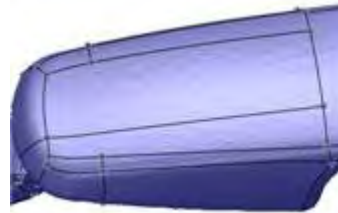


After Deform

Split (S) – Adds a spline and splits a patch to generate an additional patch by picking a point on a spline and then picking a second point on another spline that will be displayed in green. The shortcut key of this method is **S**.

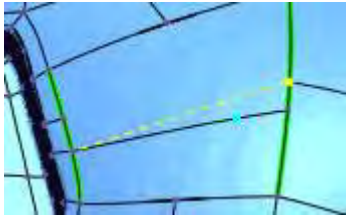


Before Split



After Split

Slide (E) – Repositions the start or end point of a spline to another position. Pick a spline first and then pick the position on another green spline. The shortcut key of this method is **E**.

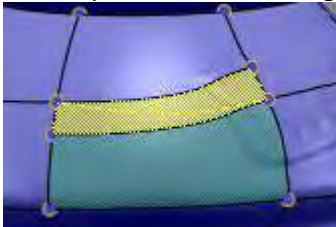


Before Slide

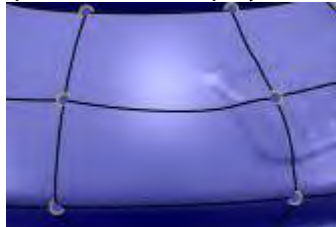


After Slide

Merge (G) – Removes a patch in a network by merging patches together. Patches that are able to be merged will be displayed in green and when the mouse cursor is placed over a green patch, a preview of the merged patch will be displayed. The shortcut key of this method is **G**.

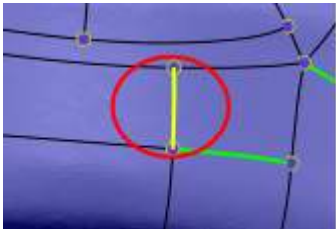


Before Merge

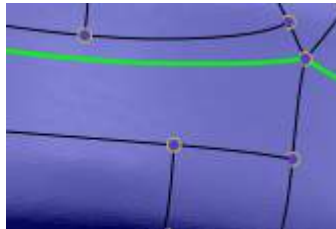


After Merge

Remove (R) – Deletes a spline. Only highlighted splines can be removed. The shortcut key of this method is **R**.



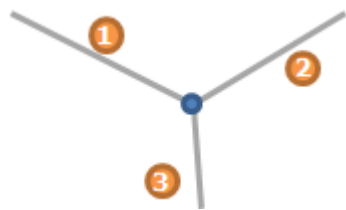
Before Remove



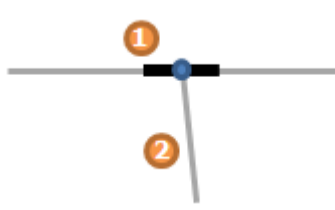
After Remove

Swap Y and T Junction (W) – Swaps the properties of a Y junction and a T junction. Splines able to be swap will be displayed in green. When a green spline is selected, the T junction mark will be swap to the Y junction. The shortcut key of this method is **W**.

Y junction consists of three splines and T junction consists of two splines.

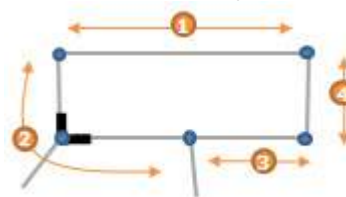


Y Junction

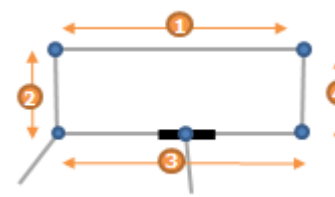


T Junction

If a patch is consist of Y and T junction but T junction is located at a corner of a rectangle, the rectangle does not consist of right four edges. By swapping Y and T junction this rectangle can have correct four edges.

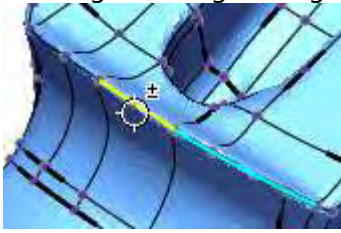


Before Swap

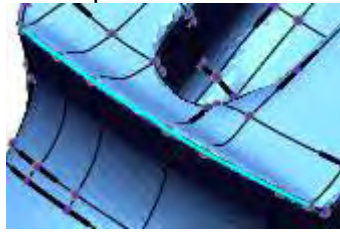


After Swap

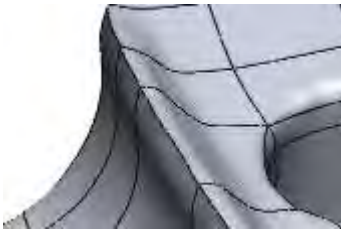
Define Sharp Edges (H) – Defines a spline as a sharp edge which prevents a spline from matching with neighboring surfaces. Splines defined as sharp edges will be highlighted.



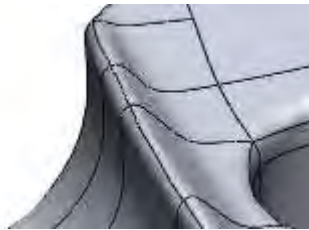
Defining Sharp Edges



Defined Sharp Edges





Fitting Result with Sharp Edge

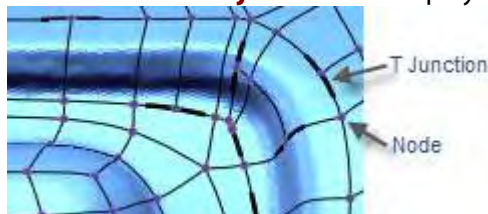


Fitting Result without Sharp Edge

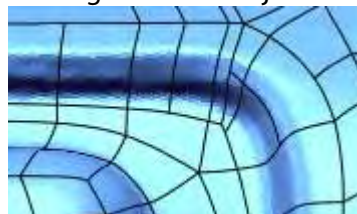
Quick Preview – Displays preview results without matching and sewing.

Show Erroneous Patches – Highlights erroneous patches (gaps between patches or missed patches after generating fitting surfaces) in red. The thin red patch shows errors created when fitting or sewing. The thick red patch shows errors created when generating surfaces. Erroneous patches can be navigated through by clicking the **Previous**  or **Next**  buttons.

Show Nodes And T-junctions – Displays interesting nodes and T junction on patches.



Show Nodes and T Junctions



Hide Nodes and T Junctions

Show Defined Sharp Edge – Displays sharp edges defined by the editing method “Define Sharp Edges (H)”.

Deform With Stiffening Around Crossing Nodes – Maintains tangency after deforming splines around an intersecting node, otherwise, sharp edges will be formed.



With Option



Without Option

See Also

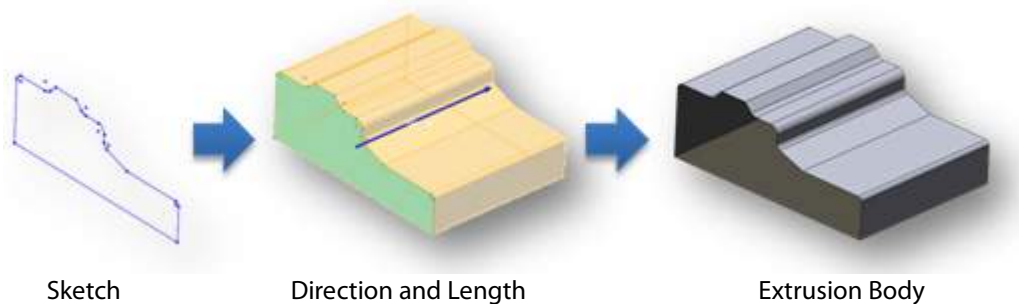
[Insert > Surface > Mesh Fit](#)

[Insert > Surface > Boundary Fit](#)

Insert > Surface > Extrude

What

The **Extrude** command stretches an object section in a straight direction and then generates an open solid body. The object section is expressed by a single profile or multiple profiles on a sketch. A profile is drawn by sketch entities such as a single circle or a spline or the combination of lines, arcs or splines.



Note:

To generate an extruded surface feature, it is not necessary for a profile to be closed. If an extruded solid feature generated from a closed profile is desired use the **Solid > Extrude** command.

Why

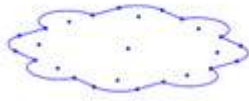
The **Extrude** command is useful for:

- Creating a body for a straight object
- Creating a boss or rib on a plastic part
- Creating a hole in an object
- Creating a straight pipe
- Creating the outside of a building
- Creating a pillar of a building or bridge

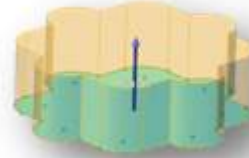
How to Use

How to generate an extrusion surface body

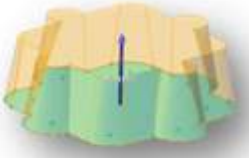
1. Draw a sketch or a mesh sketch.



2. Go to **Insert > Surface > Extrude** or click the icon in the toolbar and select the sketch as the **Base Sketch**. Specify the desired **Length** for the extrusion.



3. Check the **Draft** option and specify the value as **10** degrees to generate a tapered body.



4. Click **OK** to complete the command.

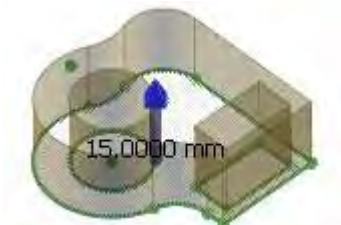


Options in Detail

Select Entities

Base Sketch – Select a base sketch, or a 2D sectional profile of an extrusion object.

Profiles – Selects closed-loop profiles in a sketch to extrude.



Extrude Loop 1, 2 & 3



Extrude Loop 2 & 3

Custom Direction – Specifies a different extrusion direction as the default direction for a linear entity with a normal. The default extrusion direction will be the normal of the sketch plane.



Default Extrusion



Custom Direction Extrusion

Methods

The following methods are available in this command:

- **Blind**
Defines the height of an extrusion
 - **Through All**
Defines the height of an extrusion to the end of all bodies
 - **Up To Vertex**
Defines the height of an extrusion to a vertex.
 - **Mid Plane**
Defines the symmetry extrusion feature of a sketch
-

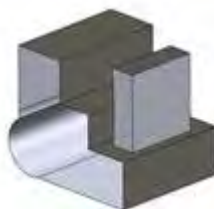
Blind – Extends a profile to a specified distance.



Through All – Extends a profile through all existing solid or surface bodies in an extrusion direction.




Up To Vertex – Extends a profile until a selected vertex.

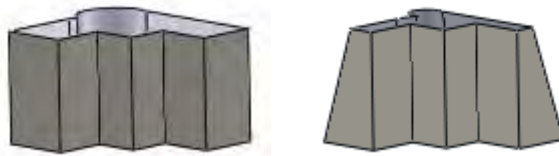


Mid Plane – Extends a profile in two directions from a sketch. The sketch will become the middle of the extrusion.



More Options

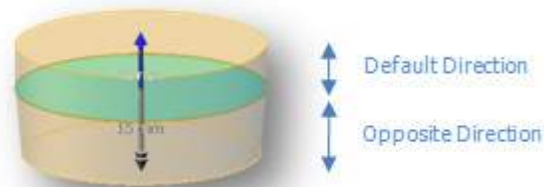
Draft – Applies a draft angle to a model that can be flipped by using the **Flip Direction** button . Draft angles are used for a mold to ease in the removal of a molded part.



Draft: 0

Draft: 10

Opposite Direction – Extrudes a body shape from a base sketch in the opposite direction of the default direction.



See Also

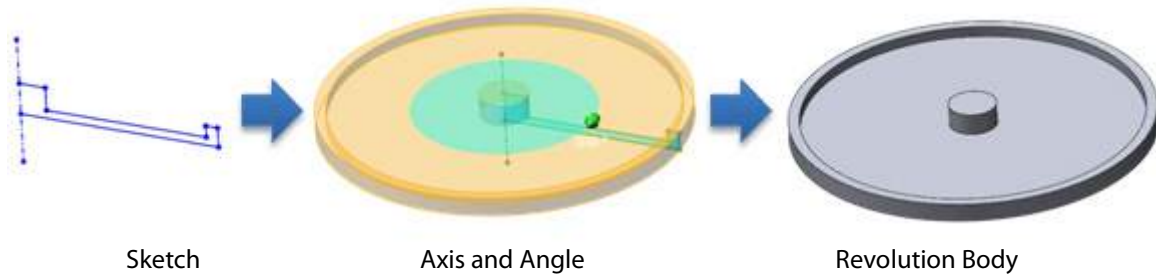
[Insert > Modeling Wizard > Extrusion Wizard](#)

[Insert > Solid > Extrude](#)

Insert > Surface > Revolve

What

The **Revolve** command turns an object section around a revolution axis and then generates a surface body. The object section is expressed by a single profile or multiple profiles on a sketch. A profile is created by sketch entities such as a circle, spline, or the combination of lines, arcs and splines.



Note:

To generate a revolved surface feature, it is not necessary for a profile to be closed. If a revolved solid feature generated from a closed profile is desired use the **Solid > Revolve** command.

Why

The **Revolve** command is useful for:

- Creating a fully revolved object
- Creating a half revolved or angled object

How to Use

How to generate a revolved surface from a revolution angle

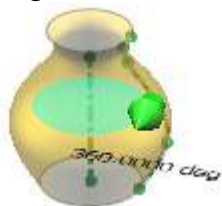
1. Draw a sketch or a mesh sketch that includes profile and revolution axis.



Note:

A revolution axis can be a construction line or another linear entity from a sketch such as a vector or an overlapping edge.

2. Go to **Insert > Surface > Revolve** or click the icon in the toolbar.
3. Select a **Base Sketch**. If a sketch has an axis as a construction line the application will automatically recognize it as the **Axis**.



4. Click **OK** to complete the command.

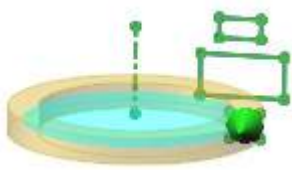


Options in Detail

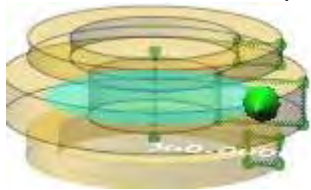
Select Entities

Base Sketch – Selects a sketch to revolve.

Profiles – Selects desired loops to revolve from multiple loops.



Revolve Loop 1



Revolve Loop 1, 2 & 3

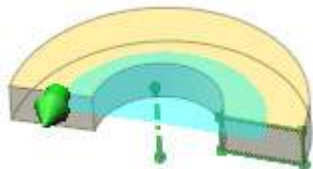
Axis – Selects a linear entity as an axis to revolve a sketch around.

Methods

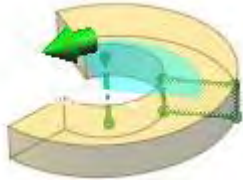
The following methods are available for this command:

- One Direction
Sets the revolution direction to a single direction
 - Mid Plane
Sets symmetrical a revolution direction for both side of a sketch
 - Two Direction
Creates a revolution in both direction with different angle values
-

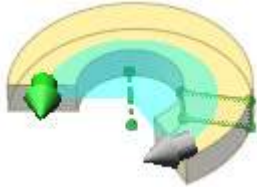
One Direction – Creates a revolved feature in one direction from a sketch.



Mid Plane – Creates a symmetrical revolved feature in both the clockwise and counterclockwise directions from a base sketch.



Two-Direction – Creates a revolved feature in the clockwise and counterclockwise directions from a sketch plane with different angle values. The total of the two angles cannot exceed 360 degrees.



More Options

Angle – Specifies the revolution angle from a base sketch.

Opposite Angle – Specifies the revolution angle of the side opposite the default side. This option is available only for the **Both Directions** method.

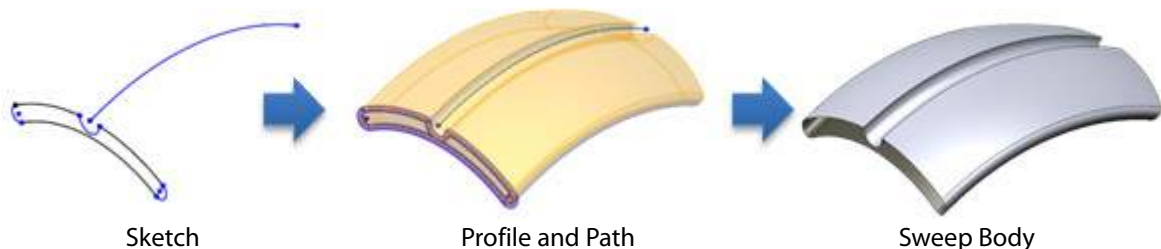
See Also

[Insert > Modeling Wizard > Revolution Wizard](#)
[Insert > Solid > Revolve](#)

Insert > Surface > Sweep

What

The **Sweep** command traces the path of an object section and generates a surface body from it. The object section is expressed by a single profile or multiple profiles on a sketch. A profile is created by sketch entities such as a circle, spline, or the combination of lines, arcs and splines.



Note:

To generate a swept surface feature, it is not necessary for a profile to be closed. If a swept solid feature generated from a closed profile is desired use the **Solid > Sweep** command.

Why

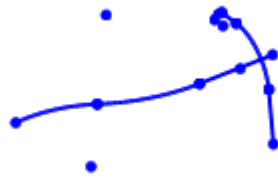
The **Sweep** command is useful for:

- Creating a bent object with a constant section
- Creating a twisted object with a constant section

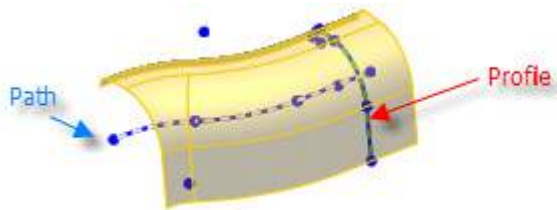
How to Use

How to generate a sweep surface feature using a profile and a path

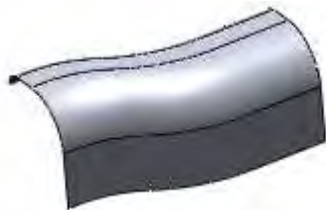
1. Prepare sketches that include a profile and a path.



2. Go to **Insert > Surface > Sweep** or click the icon in the toolbar.
3. Select a **Profile** and a **Path**.



4. Change the **Method** to **Keep Normal Constant**.
5. Click **OK** to complete the command.



Options in Detail

Select Entities

Profiles – Selects a profile such as a sketch, 3D sketch, or edge that will be used to create a sweep body. To make a composite profile, select edges or curves with the **Shift** button. A profile should not self-intersect.

Paths – Selects a path on which the profile follows. You can use the opened or closed loop. The start point of the path must lie on the plane of the profile. The path should not be self-

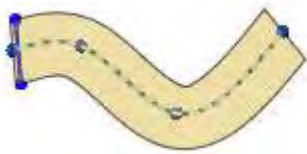
intersection.

Methods

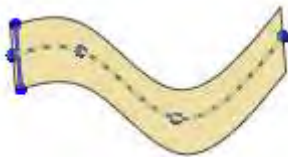
The following methods are available in this command:

- Follow Path
 - Keep Normal Constant
 - Follow Path AND First Guide Curve
 - Follow First And Second Guide Curves
 - Twist Along Path
 - Twist Along Path With Normal Constant
-

Follow Path – Keeps a profile at same angle with respect to path at all times.



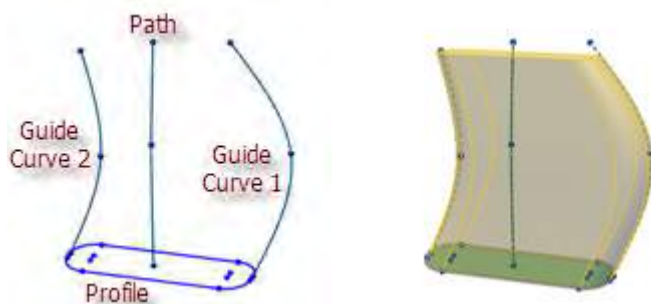
Keep Normal Constant – Keeps a profile parallel to the beginning section at all times.



Follow Path and First Guide Curve – Selects a guide curve to control the intermediate shape of a profile. Click the **Curves** button and select a guide curve.



Follow First and Second Guide Curve – Selects two guide curves to control the intermediate shape of a profile. Click the **Curves** button and select guide curves.



Twist Along Path – Twists a section along a path. Define the twist angle by degrees.



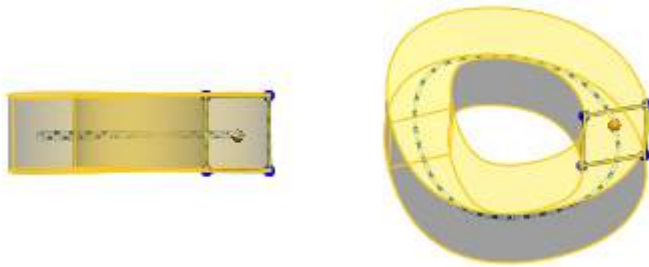
Twist Along Path With Normal Constant – Twists a section along a path keeping the section parallel to the beginning section as it twists along the path.



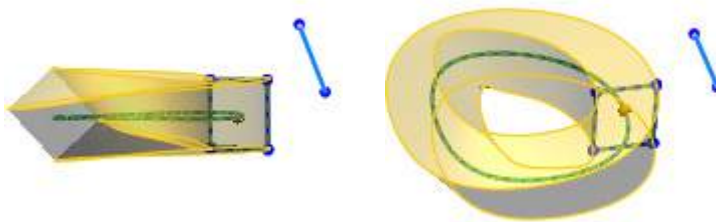
Path Options

Path Alignment Type – Stabilizes a profile when small and uneven curvature fluctuations along the path cause the profile to misalign. This option is available when **Follow Path** is selected.

None – Aligns a profile normal to a path. No correction is applied.



Direction Vector – Aligns a profile in a selected direction. Select a linear entity to set the direction vector.



Guide Curves

Guide Curves – Guides a profile as it sweeps along a path. Click the **Curves** button and select a guide curve on the screen.

Options

Improve Curvature Flow Along Sweep Path – Smoothly connects curvature flow of faces along a sweep path.

Merge Tangent Faces – Merges tangentially connected faces and reduces the total face numbers.



Without Merge Tangent Faces



With Merge Tangent Faces

See Also

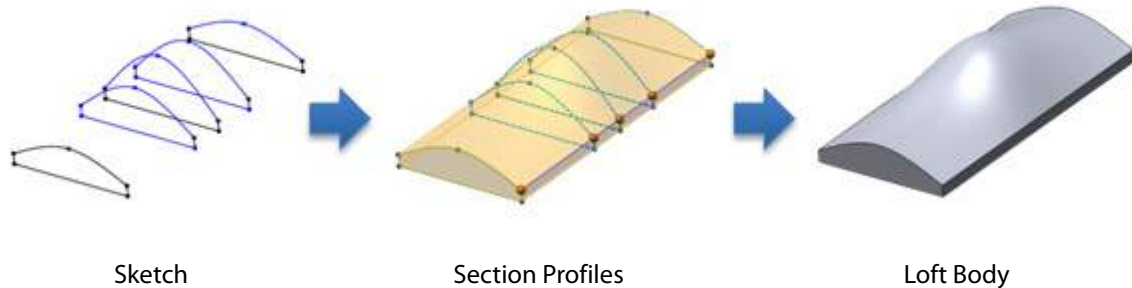
[Insert > Modeling Wizard > Sweep Wizard](#)

[Insert > Solid > Sweep](#)

Insert > Surface > Loft

What

The **Loft** command connects multiple object sections and generates a surface body from them. The object sections are expressed by a single profile or multiple profiles on a 2D or 3D sketch and can be drawn by circles, splines, lines and arc sketch entities.



Note:

To generate a lofted solid feature, a profile must be closed. For more information, see

Why

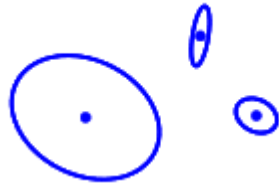
The **Loft** command is useful for:

- Creating an object connected by multiple sections
- Creating different starting and ending profile objects

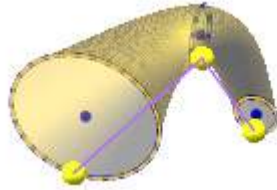
How to use Loft command

How to generate a loft feature using profiles

1. Go to **Insert > Surface > Loft** or click the icon in the toolbar.

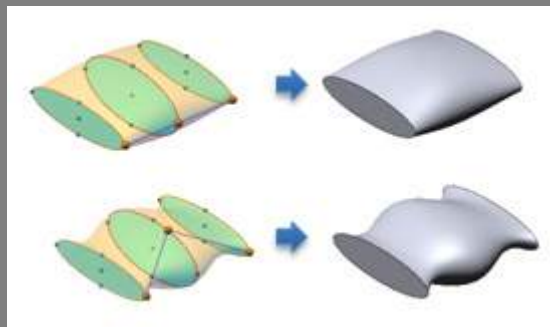


2. Select **Profiles** to loft.





Note:

The alignment between loft profiles can be modified by manipulating the handles (spheres) that appear on the selected profiles as part of the connector. The connector is the poly line connecting corresponding points in both directions.



Note:

The lofting order of profiles in the dialog tree can be changed by using the **Up**  and **Down**  buttons after selecting a profile in the list.



3. Click **OK** to complete the command.



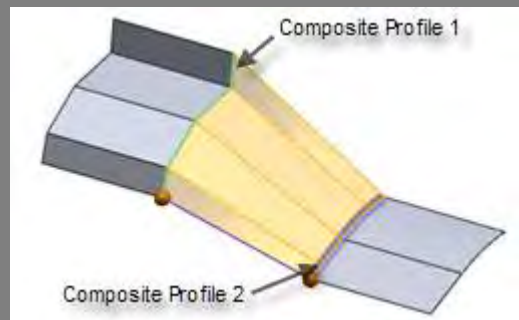
Options in Detail

Select Entities

Profiles – Selects a profile such as a sketch, 3D sketch, or edge that will be used to create a sweep body. To make a composite profile, select edges or curves with the **Shift** button. A profile should not self-intersect.

Note:

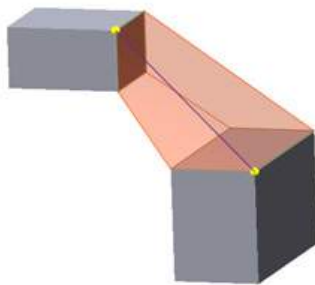
- Non-planar profiles can be used for a loft surface.
- To make a composite profile, select multiple edges or curves with the **Shift** key.




Constraints

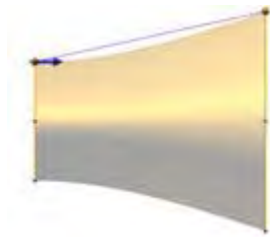
Start Constraint – Selects the starting constraint type.

None – Applies no tangency constraints or zero curvatures.

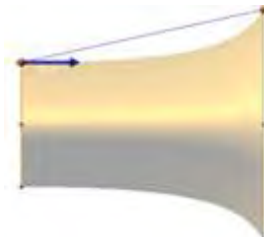


Direction Vector – Applies a tangency constraint based on a selected entity. Click the **Direction Vector** button and select a linear entity such as a vector, line, edge, etc, or a planar entity such as a plane, region, or face.

Tangent Length – Controls the amount of tangency influence on a loof by setting the value or manipulating the blue arrow in the Model View. The tangency direction can be reversed by using the **Flip Tangent** button .

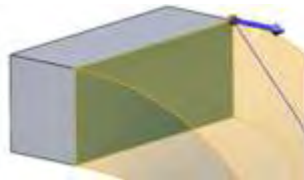


Tangent Length: 1

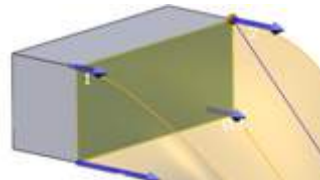


Tangent Length: 2

Use Same Tangent Length To All Vertices - Applies the same tangency to all vertices on a profile. Tangent length can also be applied individually on each vertex.



Use Same Tangent Length



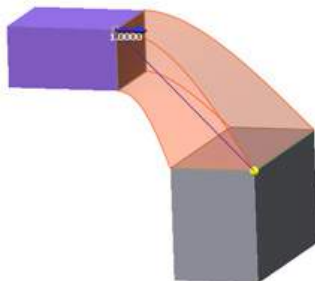
Individual Tangent Length

Normal To Profile - Applies a tangency constraint normal to a starting profile.

Tangent Length – Refer to the **Direction Vector** option.

Use Same Tangent Length To All Vertices – Refer to the **Direction Vector** option.

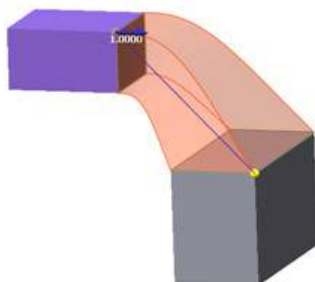
Tangency To Face – Creates a tangency constraint that follows a selected face and is available if a face is selected as a profile. Clicking the **Next Face** button will find the next candidate face.



Tangent Length – Refer to the **Direction Vector** option.

Use Same Tangent Length To All Vertices – Refer to the **Direction Vector** option.

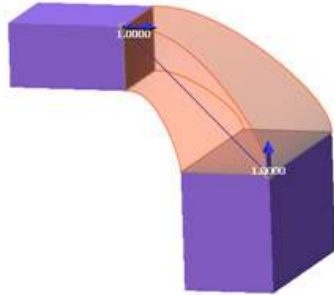
Curvature To Face – Creates a curvature constraint that follows a selected face and is available if a face is selected as a profile. Clicking the **Next Face** button will find the next candidate face.



Tangent Length – Refer to the **Direction Vector** option.

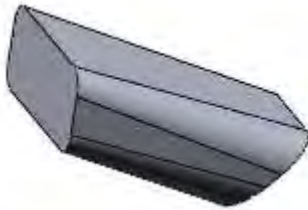
Use Same Tangent Length To All Vertices – Refer to the **Direction Vector** option.

End Constraint – Selects the ending constraint type. The sub-options are the same as the **Start Constraint** options.



Guide Curves

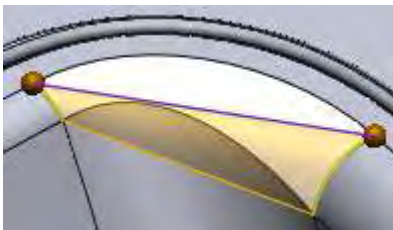
Guide Curves – Selects a guide curve to control the intermediate shape of a sweep body.



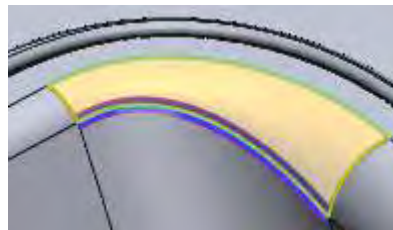
Without Guide Curve



With Guide Curve



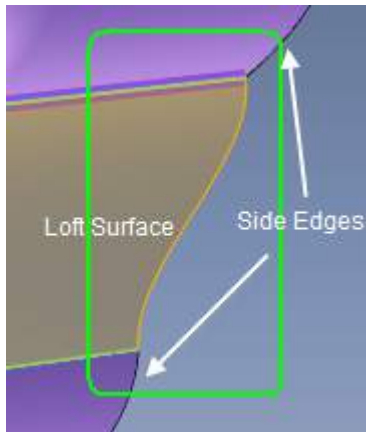
Without Guide Curve



With Guide Curve

Options

Add Side Curve Influence – Sets the side edges of a loft surface tangent to the side edges of references. This option is available for **Direction Vector**, **Tangency To Face**, or **Curvature To Face** constraints.



Without Side Curve Influence



With Side Curve Influence

Merge Tangent Faces – Merges tangentially connected faces to reduce the total number of faces.

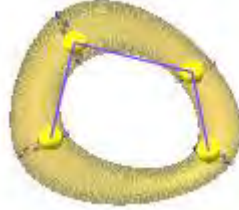
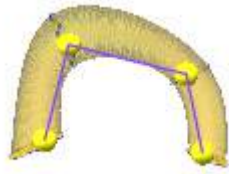


Without Merge Tangent Faces



With Merge Tangent Faces

Close Loft – Closes a lofted body.



See Also

[Insert > Modeling Wizard > Loft Wizard](#)

[Insert > Solid > Loft](#)

Insert > Surface > Offset

What

The **Offset** command enlarges or shrinks a surface body by a constant distance.



Why

The **Offset** command is useful for:

- Creating a model in a different size
- Offsetting faces on a surface or solid body
- Copying faces on a surface or solid body

How to Use

A. How to offset faces on a surface or a solid body

1. Prepare a solid or surface body.



2. Go to **Insert > Surface > Offset** or click the icon in the toolbar.
3. Select faces on a body and specify the **Offset** distance.



4. Click **OK** to complete the command.

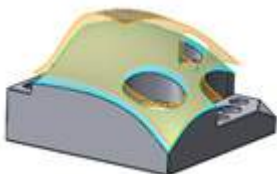


B. How to copy faces on a body

1. Go to **Insert > Surface > Offset** or click the icon in the toolbar.



2. Select a face to copy.



3. Change the offset distance to **0**.



4. Click **OK** to complete the command and then hide the body. A copied face can be seen.



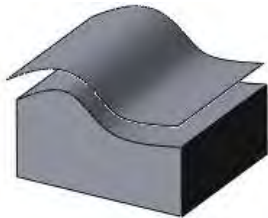
Options in Detail

Faces – Selects faces on a body or an entire surface body.

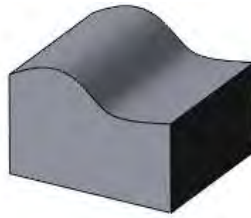
Offset Distance – Specifies the distance to offset from a solid body. To reverse the offset direction, use the **Flip Direction** button.



Make Side Wall – Creates side walls between an offset face and a body.



Disable Option

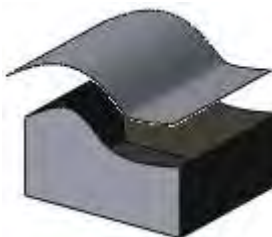


Enable Option

Remove Original Faces – Removes original faces used to offset.



Disable Option



Enable Option

See Also

[Insert > Solid > Offset](#)

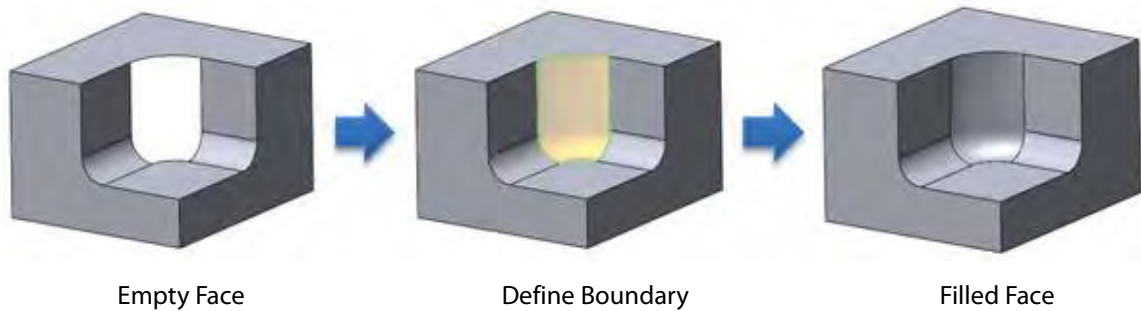
[Insert > Surface > Fill Face](#)

[Insert > Surface > Trim](#)

Insert > Surface > Fill Face

What

The **Fill Face** command generates a surface patch with any number of sides within a boundary defined by edges, sketches or curves.



Why

The **Fill Face** command is useful for:

- Filling in missing faces of an imported body
- Healing faces
- Modifying local features
- Filling holes for machining

How to Use

How to fill a missing face

1. Prepare a body with a missing face.



2. Go to **Insert > Surface > Fill Face** or click the icon in the toolbar.
3. Select the edges to fill with a face.



4. Click **OK** to complete the command.



Options in Detail

Select Entities

Edges – Selects edges on a body to fill with a face.

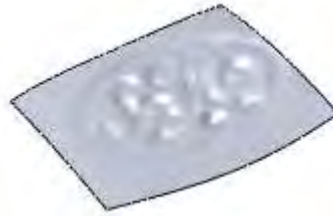
Fit Face To Underlying Mesh – Fits a face to an underlying mesh. This option is useful for real parts that are different from their originally designed CAD parts.

Geometry Capture Accuracy – Defines geometry capture accuracy. When the slider is moved towards **Loose**, less poly-faces will be used to create a shape. When the slider is moved towards **Tight**, more poly-faces will be used to create a shape.

Smoothness – Determines the smoothness level of a fitting surface. Moving the slider to **More** will increase smoothness but can also lower accuracy.



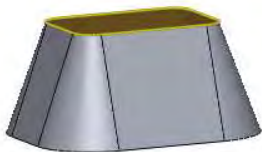
Smoothness: Less



Smoothness: More

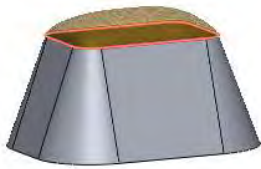
Matching Options

Set Continuity Constraint – Defines a tangent or curvature constraint with adjacent faces. If this option is not used, sewing will be executed with the G0 position constraint.



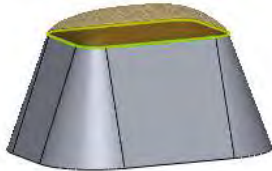
Without Continuity

Tangent Continuous Edges – Selects edges to connect with the G1 tangent constraint with adjacent faces.



Tangent

Curvature Continuous Edges – Selects edges to connect with the G2 curvature constraint with adjacent faces. This option is available if the **Create Single Patch** option is checked.

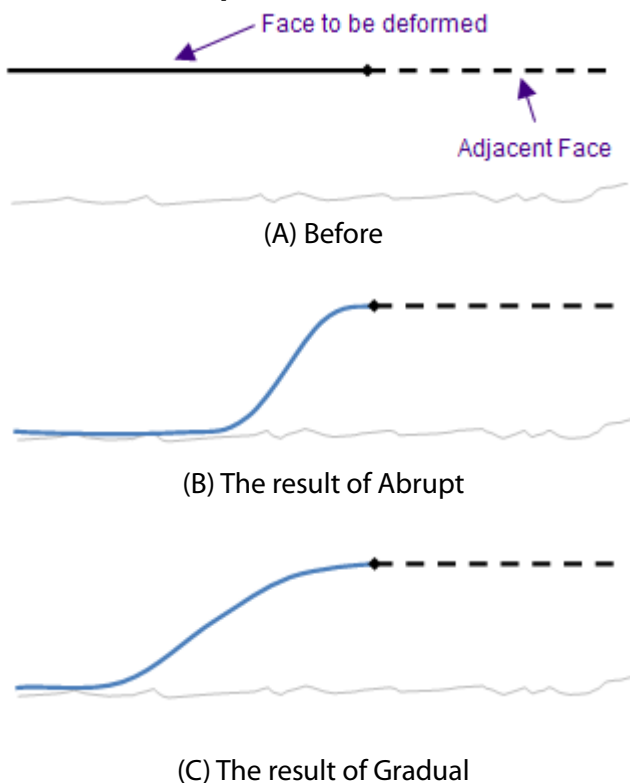


Curvature

Advanced Continuity Options

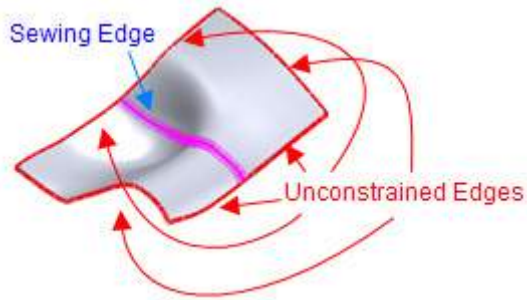
This option is available when the **Create Single Patch** or **Fit Face To Underlying Mesh** options are used.

Shape Transition To Adjacent Faces – Centralizes a modification around an edge when the slider is moved to **Abrupt**.



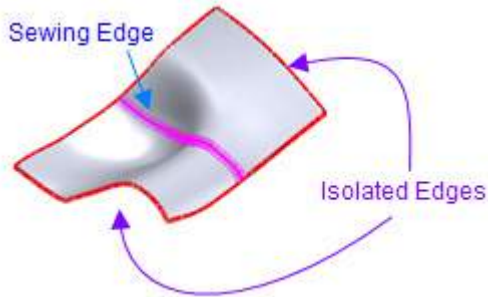
Boundary Accuracy – Determines accuracy when sewing edges. When the slider is moved to **Tight** more tightly joined points will be created on an edge.

Unconstrained Edge – Determines the deformation method for non-sewing edges that are unconstrained.



Do Not Preserve – Does not preserve unconstrained edges which may deform them.

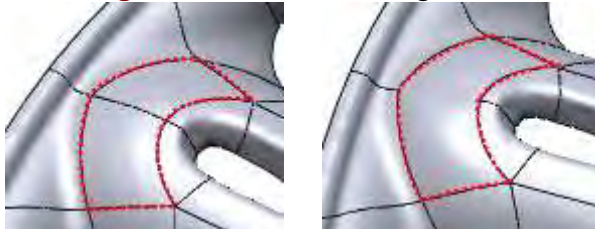
Preserve Isolated Edges Only – Preserves isolated edges that are not adjacent to sewing edges.



Preserve All – Preserves all edges without deformations.

More Options

Create Single Patch – Creates a single surface.



Disable Option

Enable Option

Merge Result – Merges a created single surface to a body to create a single body.

See Also

[Insert > Modeling Feature > Delete Face](#)

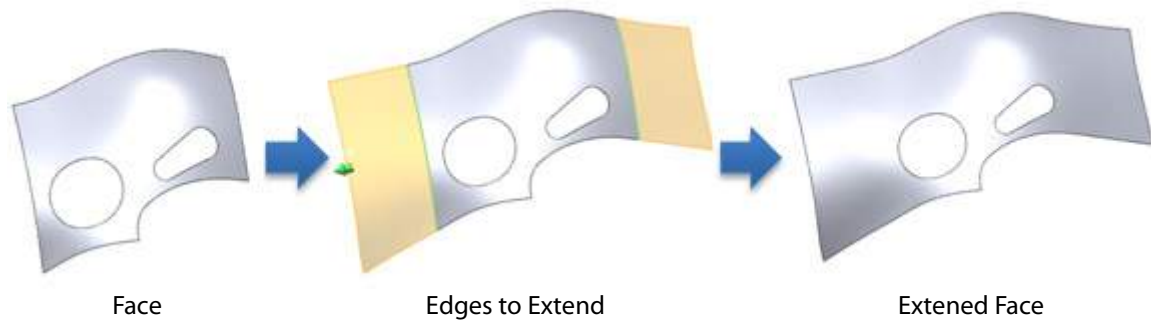
[Insert > Modeling Feature > Split Face](#)

[Insert > CAD Correct > Refit](#)

Insert > Surface > Extend

What

The **Extend** command expands a surface boundary by an edge, multiple edges, or a face.



Why

The **Extend** command is useful for:

- Expanding a face by preserving its shape
- Creating intersections with other faces

How to Use

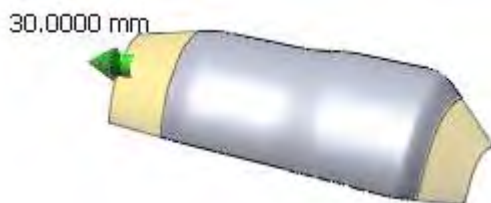
How to extend a face

1. Prepare a surface or solid body.

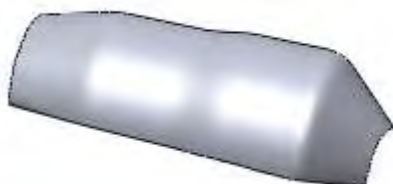


2. Go to **Insert > Surface > Extend** or click the icon in the toolbar.

3. Select edges on a body and specify a **Distance** value or click-drag the green arrow on the graphics area to extend the face.



4. Click **OK** to complete the command.



Options in Detail

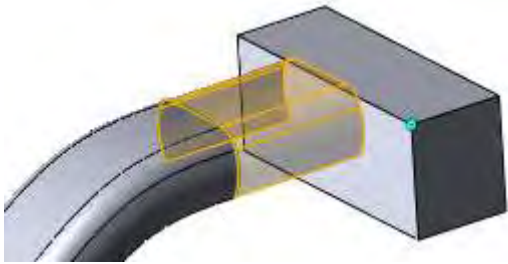
Select Entities

Edges/Faces – Selects edges or faces to extend.

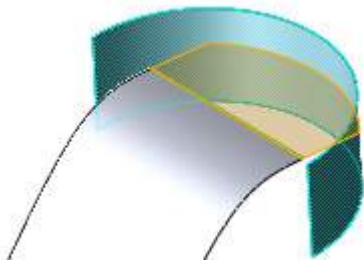
End Conditions

By Distance – Specifies an extension distance.

Up To Point – Selects a vertex for an end position.

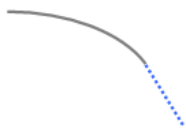


Up To Body/Region – Select a face on a body or a region. The end will be cut by the selected body or region.

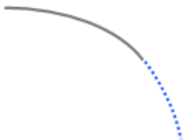


Extension Methods

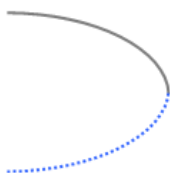
Linear – Linearly extends the surface from the original surface.



Curvature – Extends the surface by keeping curvature from the original surface.



Same Surface – Extends the surface by mirroring the original surface.



See Also

[Insert > Surface > Trim](#)

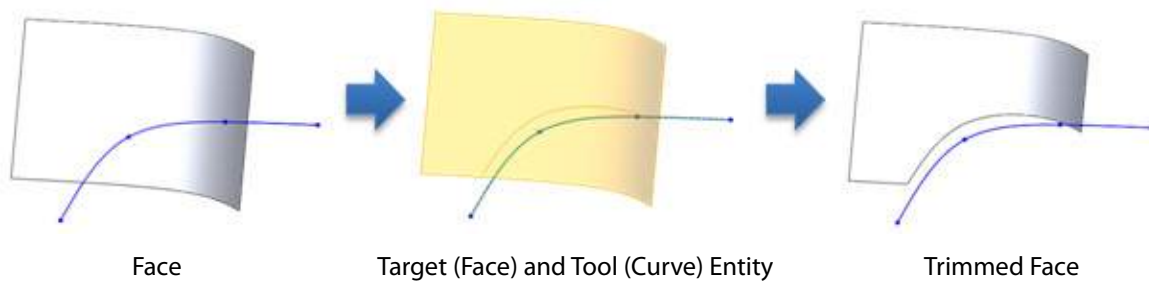
[Insert > Surface > Trim & Merge](#)

[Insert > Solid > Cut](#)

Insert > Surface > Trim

What

The **Trim** command cuts a surface body by removing material using a surface, solid body or curve. The remaining material is chosen manually.



Why

The **Trim** command is useful for:

- Creating a feature by surface modeling method
- Creating a sheet metal part
- Trimming a surface by using a curve or another surface
- Removing surface material with intersecting surface

How to Use

A. How to trim a face by a curve

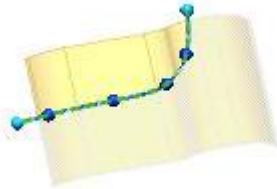
1. Prepare a face and a curve.



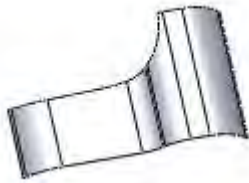
2. Go to **Insert > Surface > Trim** or click the icon in the toolbar.
3. Select a curve as a **Tools Entities** and a face as a **Target Bodies**.



3. Click the **Next Stage** button and select a face to remain after the trim.



4. Click **OK** to complete the command.

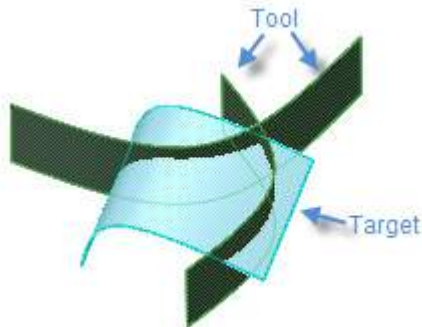


B. How to trim a face by another faces

1. Prepare a face and s curve.



2. Go to **Insert > Surface > Trim** or click the icon in the toolbar.
3. Select two faces as **Tools Entities** and a main face as a **Target Bodies**.



3. Click the **Next Stage** button and select a face to remain after the trim.



4. Click **OK** to complete the command.



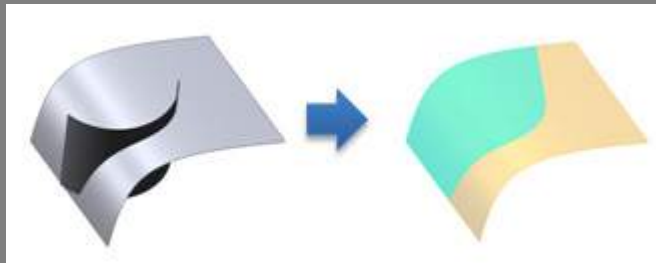
Options in Detail

Select Entities

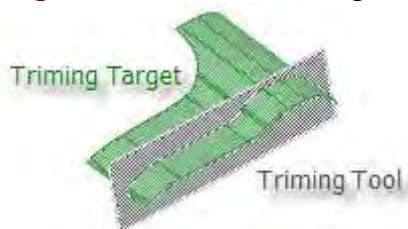
Tool Entities – Selects tool entities such as ref. planes, regions, sketch chains, surface bodies and solid bodies to cut a target face.

Note:

When a tool entity does not make a complete interaction with a target entity, it will be extended and will make an intersection automatically.



Target Bodies – Defines a target surface body.

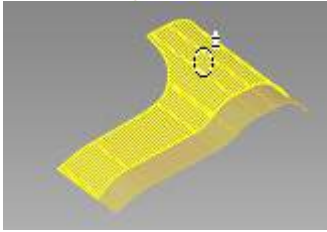


Note:

It is possible to trim between tool entities without defining a target entity. If a target entity is defined then tool entities will be hidden in the following stage to aid in the selection of remaining faces.

Result

Remaining Bodies – Selects remaining areas.



See Also

[Insert > Surface > Untrim](#)

[Insert > Surface > Trim & Merge](#)

[Insert > Solid > Cut](#)

Insert > Surface > Untrim

What

The **Untrim** command extends a surface boundary and returns it to an untrimmed status.



Why

The **Untrim** command is useful for:

- Filling holes
- Re-trimming an original face

How to Use

How to patch a hole by using the Untrim command

1. Prepare a trimmed face.



2. Go to **Insert > Surface > Untrim** or click the icon in the toolbar.
3. Select a face as **Entities**.



4. Click **OK** to complete the command.

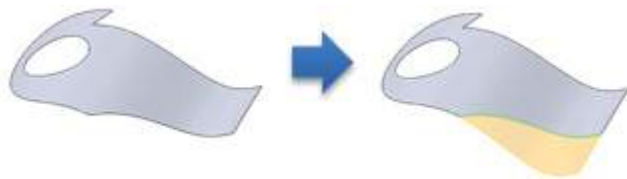


Options in Detail

Entities – Selects entities such as edges, loops and faces to untrim.



Entity: Face



Entity: Edge

See Also

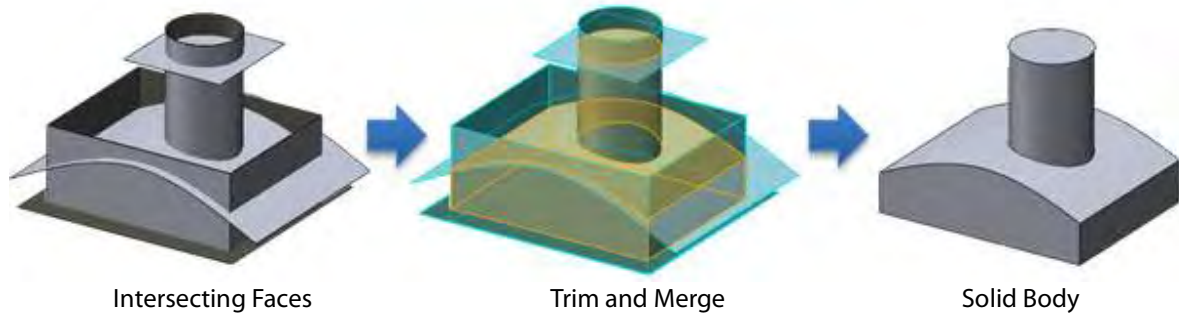
[Insert > Surface > Trim](#)

[Insert > Surface > Trim & Merge](#)

Insert > Surface > Trim & Merge

What

The **Trim & Merge** command trims faces leaving a common area among intersecting faces to generate a solid body.



Why

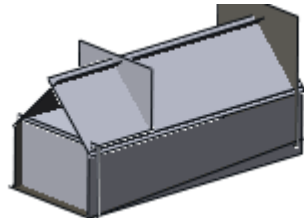
The **Trim & Merge** command is useful for:

- Creating a feature from intersecting faces

How to Use

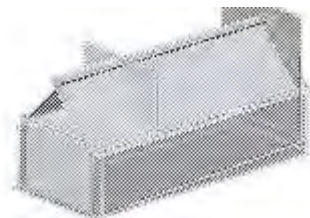
How to cut intersecting faces and leave a common area

1. Prepare a set of faces that can create inner material.

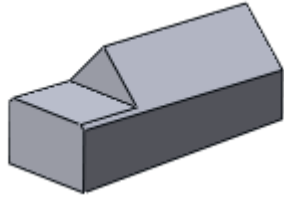


2. Go to **Insert > Surface > Trim & Merge** or click the icon in the toolbar.

3. Select all faces as **Entities**.



4. Click **OK** to complete the command.



Options in Detail

Entities – Selects enclosed faces or regions.

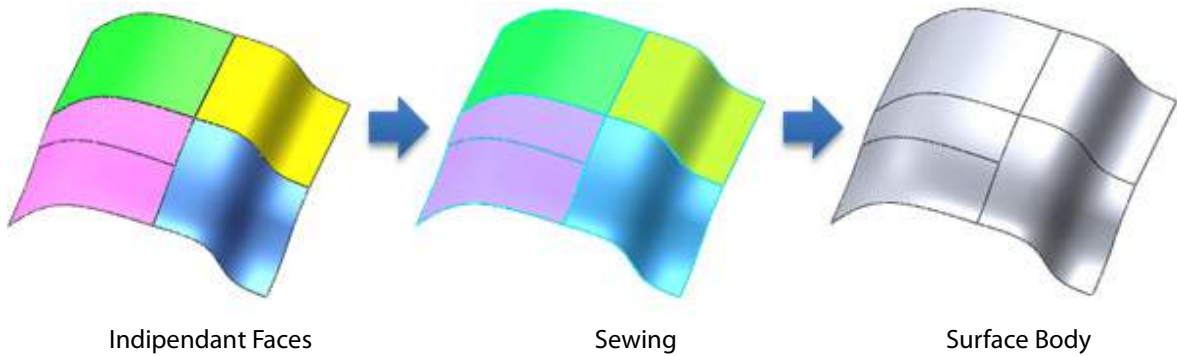
See Also

[Insert > Surface > Trim](#)

Insert > Surface > Sew

What

The **Sew** command combines two or more faces into a single surface body by sewing together boundaries.



Why

The **Sew** command is useful for:

- Combining faces
- Matching faces

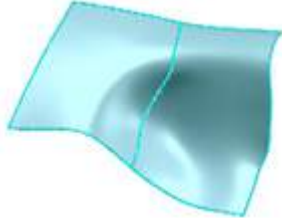
How to Use

How to sew faces with the tangent constraint

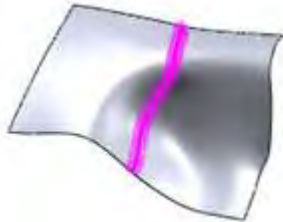
1. Prepare faces.



2. Go to **Insert > Surface > Sew** or click the icon in the toolbar.
3. Select faces as **Surface Bodies**. Click the **Next Stage** button.



4. To tangentially connect faces, check the **Set Continuity Constraint** option. Click the **Tangent Continuous Edges** button and select common edges between faces.



5. Click **OK** to complete the command. Faces will be tangentially connected to form a single surface body.



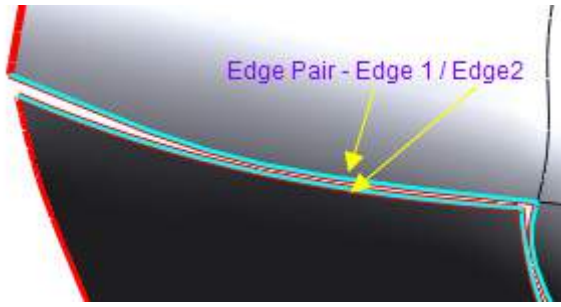
Options in Detail

Select Entities

Surface Bodies – Selects independent surfaces to sew.

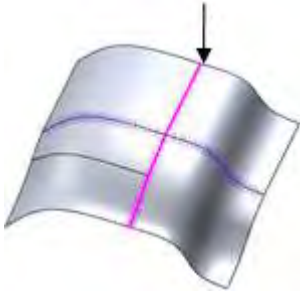
Zip Gaps

Edge Pair – Forces the connection of gaps between faces that are larger than the internal tolerance for the application. While sewing gaps surface geometry can be deformed. To make edge pairs, select two edges in sequence or drag edges together.

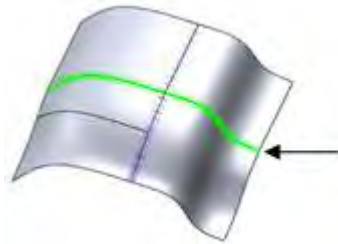


Set Continuity Constraint – Defines a tangent or curvature constraint with adjacent faces. If this option is not used, sewing will be executed with the G0 position constraint.

Tangent Continuous Edges – Selects edges to connect with the G1 tangent constraint with adjacent faces.



Curvature Continuous Edges – Selects edges to connect with the G2 curvature constraint with adjacent faces.



Note:

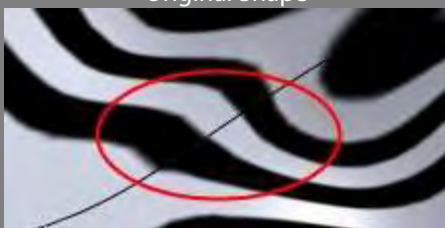
When **Environment Mapping** in **Accuracy Analyzer™** is enabled, surface matching condition will be displayed. When G0 continuity is used the flow of light will be split on an edge. When G1 continuity is used it will be connected without splitting but can have a sharp peak. When G2 continuity is used it will be smoothly connected without splitting and peaking.



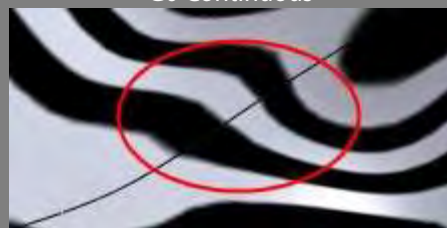
Original Shape



G0 Continuous

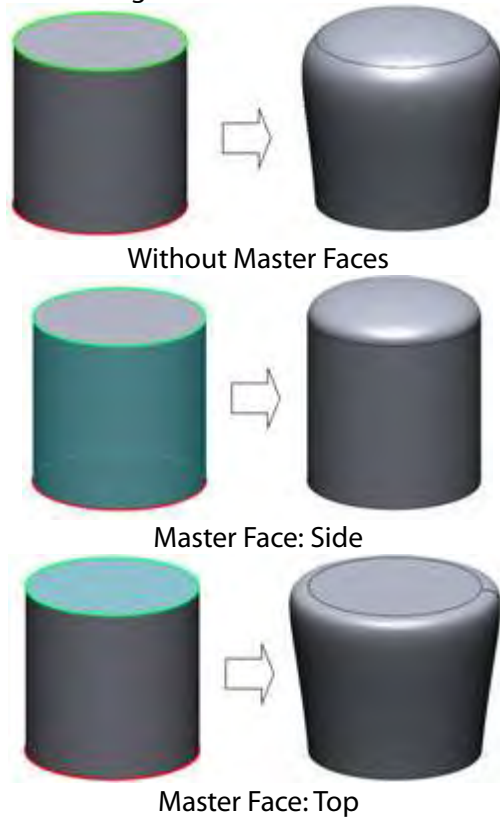


G1 Continuous



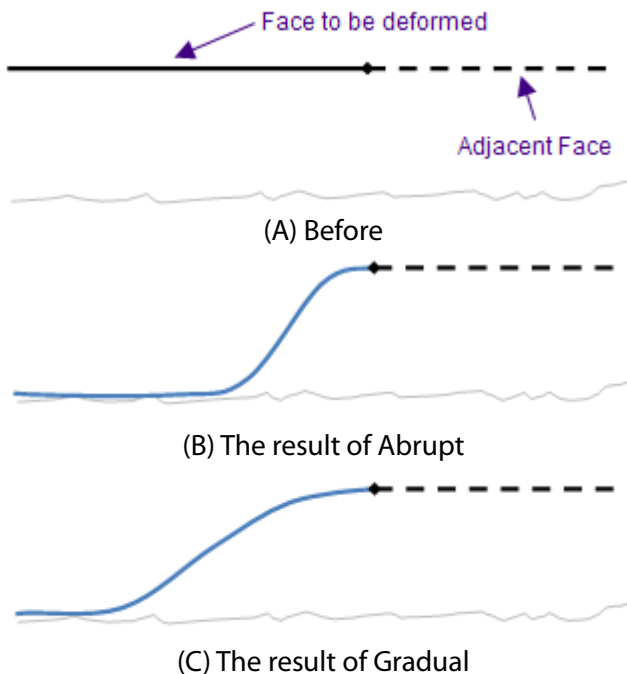
G2 Continuous

Master Faces – When faces are designated as **Master Faces** they will not deform while sewing.



Advanced Continuous Options

Shape Transition To Adjacent Faces – Centralizes the deformation of faces around an edge as the slider is moved to **Abrupt**.



Boundary Accuracy – Determines accuracy when sewing edges. When the slider is moved to **Tight** the application will create more points on an edge to sew. The **Preview** button shows the maximum deviation between edges.

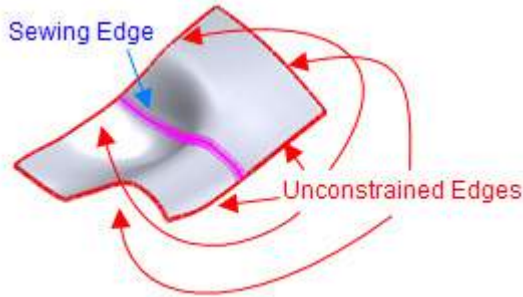


Loose



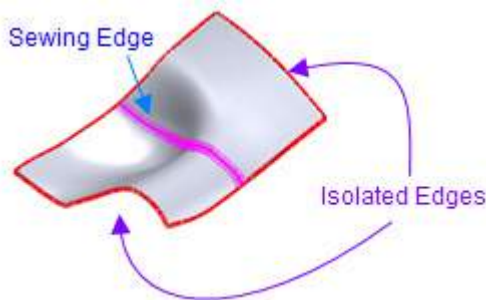
Tight

Unconstrained Edge – Determines the deformation method for non-sewing edges that are unconstrained.



Do Not Preserve – Does not preserve unconstrained edges which may deform them.

Preserve Isolated Edges Only – Preserves isolated edges that are not adjacent to sewing edges.



Preserve All – Preserves all edges without deformations.

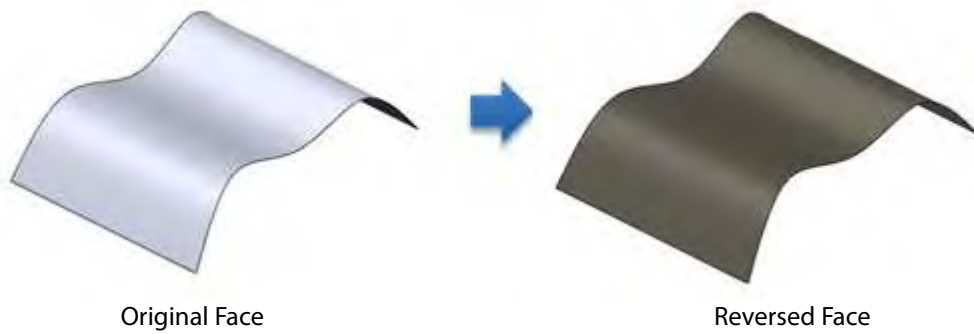
See Also

[Insert > Surface > Trim and Merge](#)

Insert > Surface > Reverse Normal

What

The **Reverse Normal** command flips the normal of faces.



Why

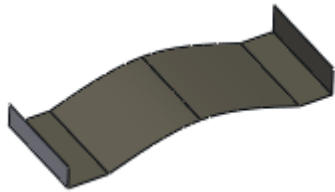
The **Reverse Normal** command is useful for:

- Mating normal between faces

How to Use

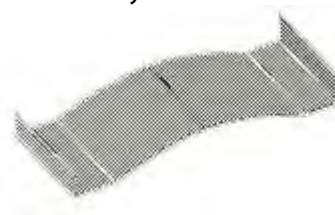
How to flip the normal of a surface body

1. Prepare a surface body.



2. Go to **Insert > Surface > Reverse Normal** or click the icon in the toolbar.

3. Select a surface body.



4. Click **OK** to complete the command.



Options in Detail

Surface Bodies – Selects surface bodies to reverse the normal.

7.4.8. Insert > Modeling Wizard

The **Modeling Wizard** menu features various commands used to extract geometric features from a mesh model. These wizards intelligently analyze features from a mesh and automatically extract feature information such as feature profiles, axes, and paths. After using a wizard, features can be manually edited for more accurate results.

Generating Primitive Features

- **Solid Primitives**

Generates the following solid features: cylinders, cones, spheres, tori and boxes

- **Surface Primitives**

Generates the following surface features: planes cylinders, cones, spheres and tori

Generating CAD Features

- **Extrusion Wizard**

Generates extruded bodies

- **Revolution Wizard**

Generates revolved bodies

- **Sweep Wizard**

Generates swept bodies

- **Loft Wizard**

Generates lofted bodies

- **Pipe Wizard**

Generates swept bodies with circular profiles and paths

Insert > Modeling Wizard > Solid Primitives

What

The **Solid Primitives** command quickly extracts simple geometry objects from a mesh. Solid primitives such as cylinders, cones, spheres, tori and boxes can be extracted.



Note:

Primitive objects are extracted based on regions. To use the **Solid Primitive** function, a mesh must have region groups that can be generated by the **Region Group** mode in the toolbar.



Why

The **Solid Primitives** command is useful for:

- Generating solid primitive features

How to Use

How to generate solid primitive features from a mesh

1. Prepare a mesh.



2. Go to **Insert > Region Group** or click the icon in the toolbar. Click **OK** in the **Auto Segment** command to classify regions.



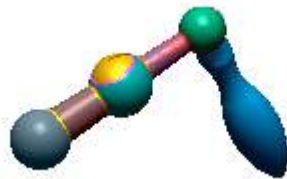
3. Exit the **Region Group** mode.
4. Go to **Insert > Modeling Wizard > Solid Primitives** or click the icon in the toolbar. Choose the **Automatically Extract Shapes** option. Select all regions on the mesh and check the **Cone** and **Sphere** shape types under the **Shapes To Be Extract** option.



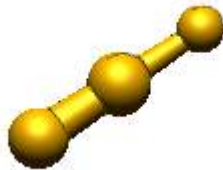
5. Click the **Next Stage** button to move to the next step.



6. Extracted primitives will be displayed and can also be removed under the **Result** option. Click **OK** to confirm remaining primitives.



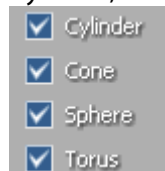
7. Hide the mesh to see the final extracted primitive solid bodies.



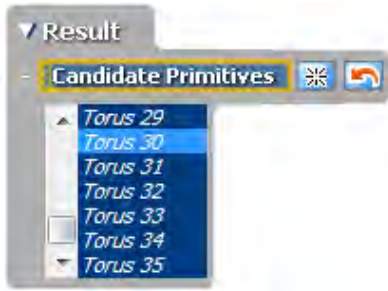
Options in Detail


Methods

Automatically Extract Shapes – Extracts possible primitive features from selected regions. Cylinder, Cone, Sphere and Torus can be selected as candidate features to be extracted.



The next stage will show the candidate primitives. To remove candidates, select them from the list box, and then click the **Delete** key.



Extract Specific Shape – Extracts a specified single primitive feature such as a Cylinder, Cone, Sphere, Torus and Box from a selected region. The **Preview** button  will show extracted primitives without completing the command.

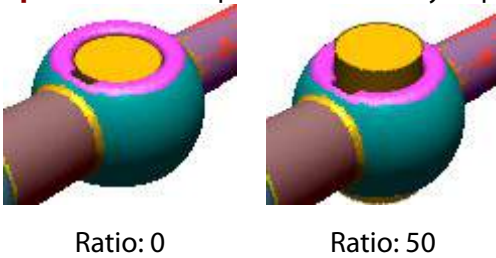


Select Entities

Regions – Selects regions on a mesh to create solid primitives.

More Options

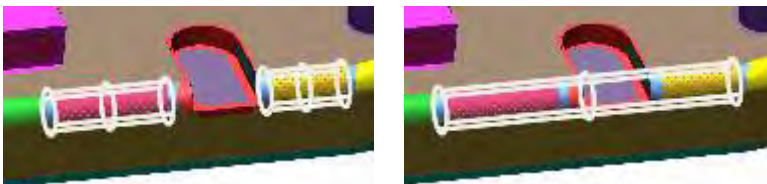
Expand Ratio – Expands a feature by a specified ratio.



Ratio: 0

Ratio: 50

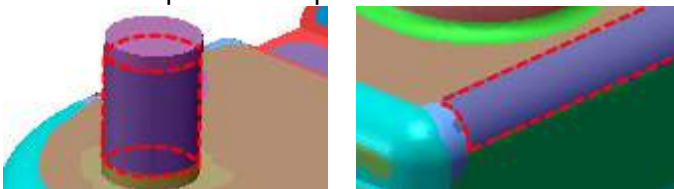
Merge Regions Belong To Single Shape – Generates a single primitive feature from separated regions as a single shape. This option is available in the **Automatically Extract Shapes** method.



Disable Merge

Enable Merge

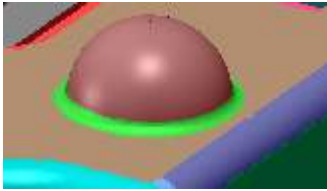
Use Only Regions Covering Whole Shape – Extracts only a complete shape of a primitive region as a primitive feature. When extracting primitive features from partial shape of regions, uncheck this option. This option is available in the **Automatically Extract Shapes** method.



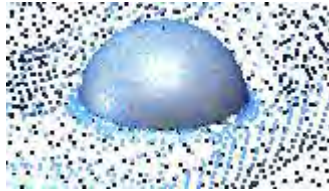
Whole Shape of Cylinder

Partial Shape of Cylinder

Get Partial Shape – Extracts only partial primitive features from partial regions.



Partial Shape of Sphere



Enable Option



Disable Option

See Also

[Insert > Modeling Wizard > Surface Primitives](#)

Insert > Modeling Wizard > Surface Primitives

What

The **Surface Primitives** command quickly extracts simple geometry objects from a mesh. Surface primitives such as cylinders, cones, spheres, and tori can be extracted.



Mesh

Mesh with Region

Surface Primitives

Note:

Primitive objects are extracted based on regions. To use the **Surface Primitive** function, a mesh must have region groups that can be generated by the **Region Group** mode in the toolbar.



Why

The **Surface Primitives** command is useful for:

- Generating surface primitive features

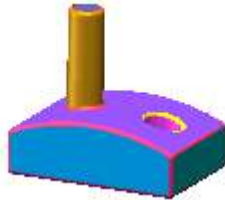
How to Use

How to generate surface primitive features from a mesh

1. Prepare a mesh.



2. Go to **Insert > Region Group** or click the icon in the toolbar. Click **OK** in the **Auto Segment** command to classify regions.

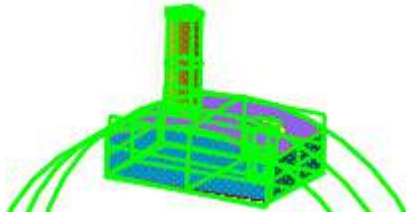


3. Exit the **Region Group** mode.

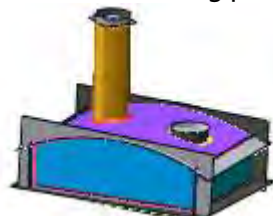
4. Go to **Insert > Modeling Wizard > Surface Primitives** or click the icon in the toolbar. Choose the **Automatically Extract Shapes** option. Select all regions on the mesh and check the **Plane** and **Cylinder** shape types under the **Shapes To Be Extract** option.



5. Click the **Next Stage** button to move to the next step.



6. Extracted primitives will be displayed and can also be removed under the **Result** option. Click **OK** to confirm remaining primitives.



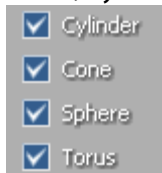
7. Hide the mesh to see the final extracted primitive surface bodies.



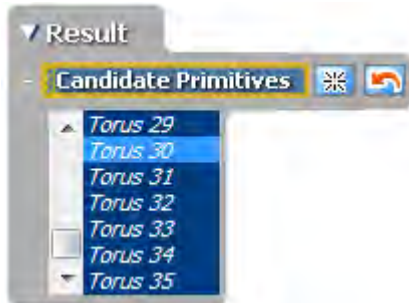
Options in Detail


Methods

Automatically Extract Shapes – Extracts possible primitive features from selected regions. Plane, Cylinder, Cone, Sphere and Torus can be selected as candidate features to be extracted.



The next stage will show the candidate primitives. To remove candidates, select them from the list box, and then click the **Delete** key.



Extract Specific Shape – Extracts a specified single primitive feature such as a Plane, Cylinder, Cone, Sphere, Torus and Box from a selected region. The **Preview** button  will show extracted primitives without completing the command.



Select Entities

Regions – Selects regions on a mesh to create surface primitives.

More Options

Expand Ratio – Expands a feature by a specified ratio.

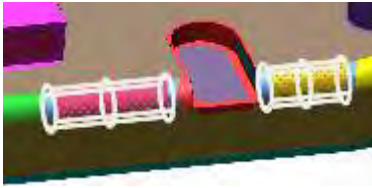


Ratio: 0

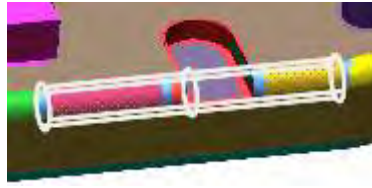


Ratio: 50

Merge Regions Belong To Single Shape - Generates a single primitive feature from separated regions as a single shape. This option is available in the **Automatically Extract Shapes** method.

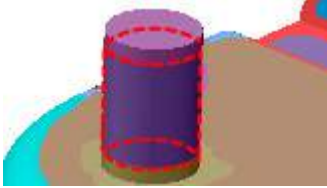


Disable Merge

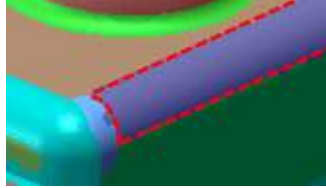


Enable Merge

Use Only Regions Covering Whole Shape - Extracts only a complete shape of a primitive region as a primitive feature. When extracting primitive features from partial shape of regions, uncheck this option. This option is available in the **Automatically Extract Shapes** method.

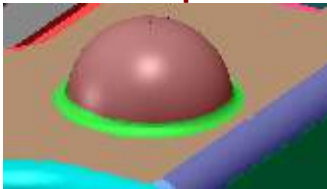


Whole Shape of Cylinder

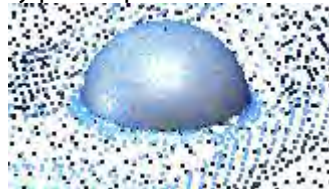


Partial Shape of Cylinder

Get Partial Shape - Extracts only partial primitive features from partial regions.



Partial Shape of Sphere



Enable Option



Disable Option

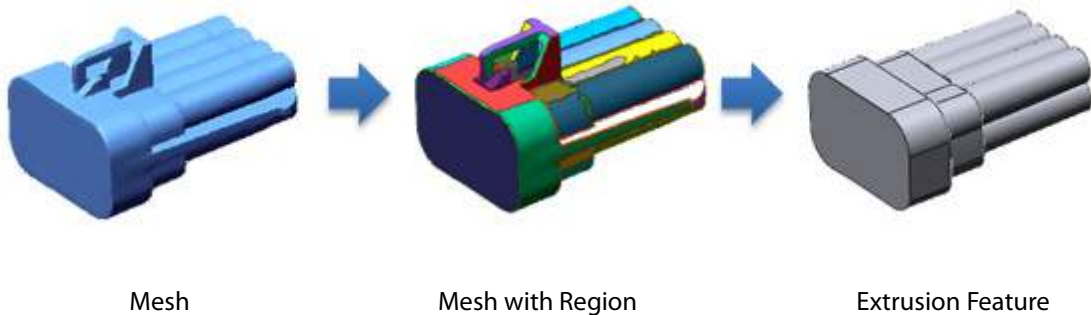
See Also

[Insert > Modeling Wizard > Solid Primitives](#)

Insert > Modeling Wizard > Extrusion Wizard

What

The **Extrusion Wizard** command extracts extruded objects from a mesh. The wizard intelligently computes sectional profiles, extrusion direction, and height based on selected regions.



Why

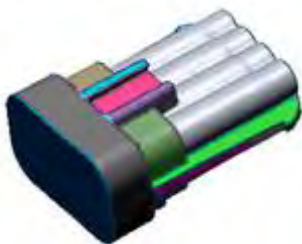
The **Extrusion Wizard** command is useful for:

- Generating extrusion solid or surface features

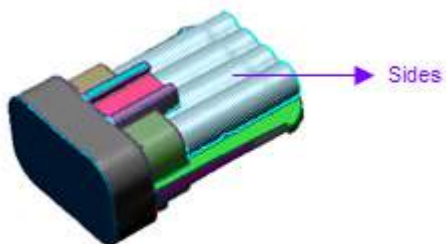
How to Use

How to quickly generate an extrusion feature based on a mesh

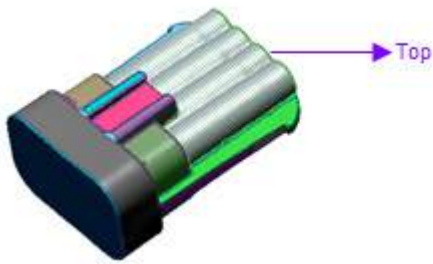
1. Prepare a mesh with regions and go to **Insert > Modeling Wizard > Extrusion Wizard** or click the icon in the toolbar.



2. Click the **Sides** button and choose regions for the side faces of an extrusion body.



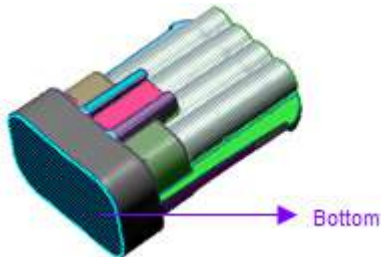
3. Click the **Top** button and choose a region for the top face of an extrusion body.



Note:

The **Extrusion Wizard** can be executed without selecting a top region. The application will automatically define the top.

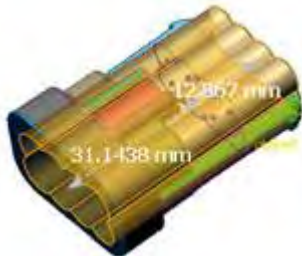
4. Click the **Bottom** button and choose a region for the bottom face of an extrusion body.



Note:

The **Extrusion Wizard** can be executed without selecting a bottom region. The application will automatically define the bottom.

5. Click the **Next Stage** button. To make a solid body, check the **Cap Ends** option.

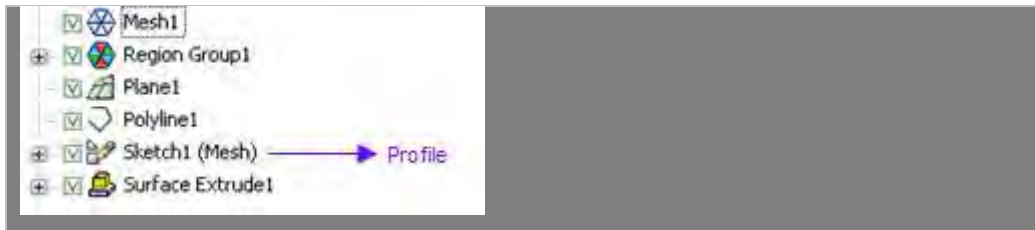


6. Click **OK** to complete the command. Hide the mesh to see the result.



Note:

The profile of an extrusion body is registered in the **Feature Tree**. Curves of the profile can be edited for a more fine and accurate extrusion body.



Options in Detail


Select Entities

Sides – Selects regions or poly-faces on the mesh.

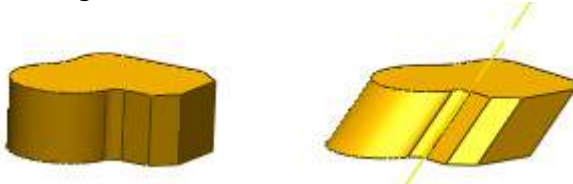
Top – Selects regions, poly-faces, poly-vertices or faces for the top face of an extrusion body. If a top face is not defined, the application will automatically find it.

Bottom – Selects regions, poly-faces, poly-vertices or faces for the bottom face of an extrusion body. If a bottom face is not defined, the application will automatically find it.

Note:

When a seed region is selected, the Estimate button  will be enabled and can be used to automatically find target regions that require the creation of an extruded feature shape.


Custom Extrusion Direction – Selects a custom direction for an extrusion from an entity or linear entity with a normal. The default extrusion direction will be calculated by the direction of selected regions.




Default Extrusion

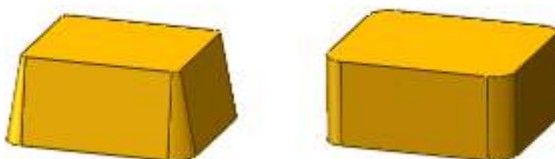
Custom Direction Extrusion

Note:

When target regions are selected, the **Custom Extrusion Direction** can be automatically found from the selected target regions by using the Estimate button .

Fix Sketch Plane With Selection – Fixes a sketch base plane by using a user-defined plane.

Draft Angle – Specifies a draft angle for an extrusion body or automatically applies a value if Auto or 0 is used. The **Estimate** button  will recommend an adaptable draft angle.



Result Operator – Specifies results. This option offers 5 different operation methods: **Solid Insert**, **Solid Merge**, **Solid Cut**, **Surface Insert**, and **Surface Cut**.

Note:

If other bodies exist already, the Boolean operation can be used while creating extruded features.

Solid Insert – Creates a new solid body.


Solid Merge – Merges a resulting body into an existing solid body.

Solid Cut – Cuts an existing solid body by a resulting solid body

Surface Insert – Creates a new surface body.

Surface Cut – Cuts an existing solid body using a resulting surface body

Second Stage Options

Draft Angle – Specifies a draft angle for an extrusion body or automatically applies a value if Auto or 0 is used. The **Estimate** button  will recommend an adaptable draft angle.

Resolution – Specifies the resolution of section segments by adjusting the slider between **Min.** and **Max.** When adjusting the slider towards **Max.**, more segments will be constructed.



Low Resolution



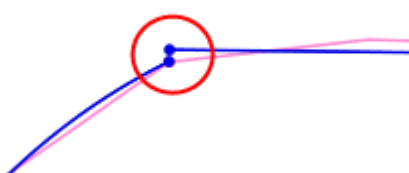
High Resolution

Note:

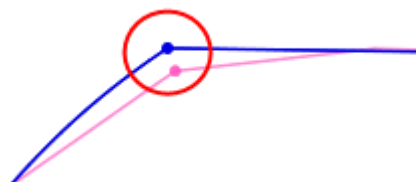
While adjusting the slider, results will be updated in real-time.

Geometry Capture Accuracy – Specifies geometry capture accuracy by adjusting the slider between **Loose** and **Tight**. When moving the slider toward **Tight**, sketches will be extracted from section segments with a tight tolerance.

Sewing Tolerance – Specifies a tolerance range for sewing sketches by adjusting the slider between **Min.** and **Max.** When adjusting the slider toward **Max.**, sketches will be joined with coincidence constraint with a loose tolerance.



With Tight Tolerance

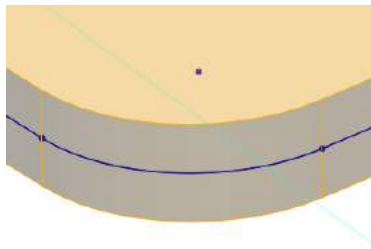


With Loose Tolerance

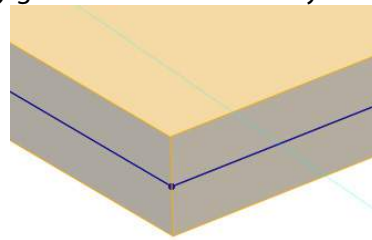
Note:

When moving the slider toward **Min.**, sketches may not be joined with coincidence constraint because of use of tight tolerance.

Remove Small Fillet – Removes automatically generated small fillets by using a radius value.



Filleted Feature



Removed Fillet

See Also

[Insert > Modeling Wizard > Revolution Wizard](#)

[Insert > Modeling Wizard > Sweep Wizard](#)

[Insert > Modeling Wizard > Loft Wizard](#)

[Insert > Modeling Wizard > Pipe Wizard](#)

Insert > Modeling Wizard > Revolution Wizard

What

The **Revolution Wizard** command extracts revolved objects from a mesh. The wizard intelligently computes sectional profiles and revolution axis and the rotation angle based on the selected regions.



Mesh



Mesh with Region



Revolution Feature

Why

The **Revolution Wizard** command is useful for:

- Generating revolved solid or surface features

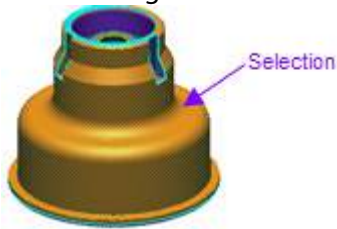
How to Use

How to generate a revolution feature based on a mesh

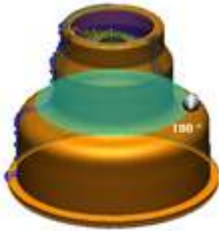
1. Prepare a mesh with regions and go to **Insert > Surface > Revolution Wizard** or click the icon in the toolbar.



2. Choose a region for a revolution body for the **Target** option.



3. Click the **Next Stage** button.



4. Click **OK** to complete the command. Hide the mesh to see the result.



Note:

The profile of an extrusion body is registered in the **Feature Tree**. Curves of a profile can be edited for a more fine and accurate extrusion body.



Options in Detail

Select Entities

Target – Selects regions or poly-faces on a mesh.

Custom Revolution Axis – Finds an optimized revolution axis from a selected target.

Note:

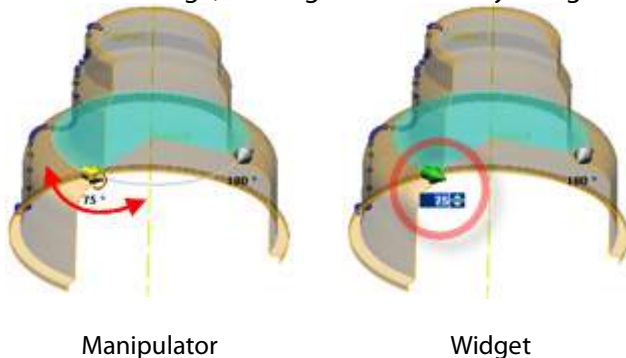
The Custom Revolution Axis can be specified by 2 planes. When specifying a custom revolution axis by using 2 planes, an intersecting line will be specified as the revolution axis and a sketch base plane will be automatically defined on a higher priority plane from the 2 specified planes. The higher priority plane is determined by the order of registration in the Feature Tree

Use Specified Axis Direction As It Is – Uses the direction of a specified axis as it is to set an axis of revolution.

Use Specified Axis Of Rotation As It Is – Uses both the direction and the position of a specified axis to set an axis of revolution.

Fix Sketch Plane With Selection – Fixes a sketch base plane by using a user-defined plane.

Get Partial Shape – Defines a revolution angle to extract a partial shape of a revolution body. In the second stage, the angle can be set by using the appropriate manipulator or widget.



Result Operator – Specifies results. This option offers 5 different operation methods: **Solid Insert**, **Solid Merge**, **Solid Cut**, **Surface Insert**, and **Surface Cut**.

Note:

If other bodies exist already, the Boolean operation can be used while creating extruded features.

Solid Insert – Creates a new solid body.

Solid Merge – Merges a resulting body into an existing solid body.

Solid Cut – Cuts an existing solid body by a resulting solid body

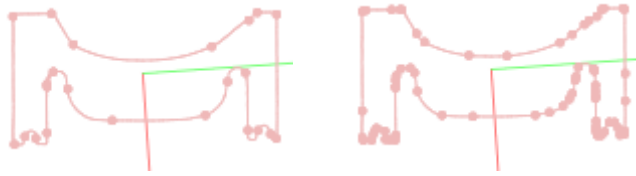
Surface Insert – Creates a new surface body.

Surface Cut – Cuts an existing solid body using a resulting surface body

Profile Auto Generation Options

Full Revolution – Creates a fully revolved feature.

Resolution – Specifies the resolution of section segments by adjusting the slider between **Min.** and **Max.** Adjusting the slider towards **Max.** will construct more segments.



Low Resolution

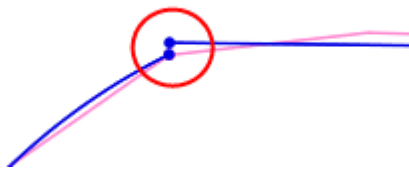
High Resolution

Note:

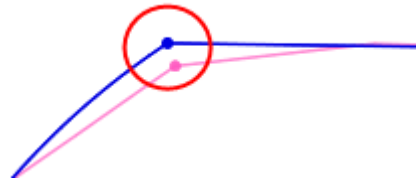
While adjusting the slider, results will be updated in real-time.

Geometry Capture Accuracy – Specifies geometry capture accuracy by adjusting the slider between **Loose** and **Tight**. When moving the slider toward **Tight**, sketches will be extracted from section segments with a tight tolerance.

Sewing Tolerance – Specifies a tolerance range for sewing sketches by adjusting the slider between **Min.** and **Max.** When adjusting the slider toward **Max.**, sketches will be joined with coincidence constraint with a loose tolerance.



With Tight Tolerance

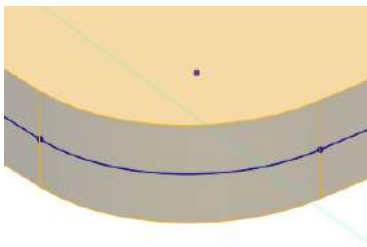


With Loose Tolerance

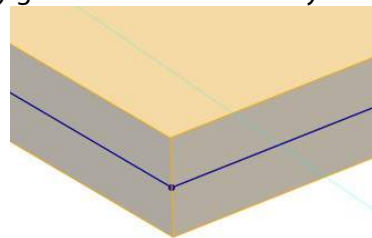
Note:

When moving the slider toward **Min.**, sketches may not be joined with coincidence constraint because of use of tight tolerance.

Remove Small Fillet – Removes automatically generated small fillets by using a radius value.



Filleted Feature



Removed Fillet

See Also

[Insert > Modeling Wizard > Extrusion Wizard](#)

[Insert > Modeling Wizard > Sweep Wizard](#)

[Insert > Modeling Wizard > Loft Wizard](#)

[Insert > Modeling Wizard > Pipe Wizard](#)

Insert > Modeling Wizard > Sweep Wizard

What

The **Sweep Wizard** command quickly extracts sweep objects from a mesh. The wizard intelligently computes sectional profiles and the path based on the selected regions.



Why

The **Sweep Wizard** command is useful for:

- Generating swept solid or surface features

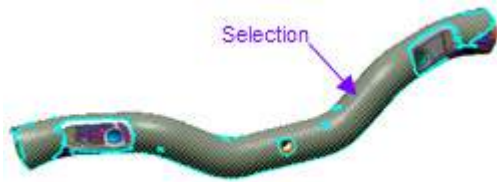
How to Use

How to quickly generate sweep feature based on a mesh

1. Prepare a mesh with regions and go to **Insert > Surface > Sweep Wizard** or click the icon in the toolbar.



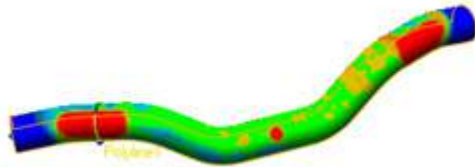
2. Select a swept region as a **Target**.



3. Choose the **Auto Generation** option under **Profile** and **Path**. Click the **Next Stage** button.
4. A preview body will be previewed.



5. To check the accuracy, click **Deviation** in the **Accuracy Analyzer** panel.

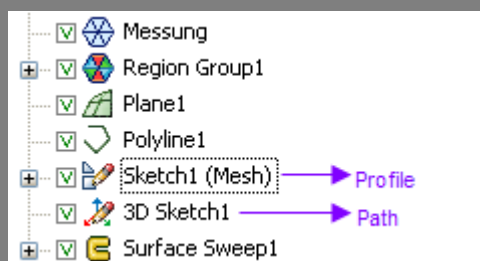


6. Click **Ok** to complete the command.



Note:

The profile of a swept body is registered in the **Feature Tree**. Curves of a profile can be edited for a more fine and accurate extrusion body.



Options in Detail

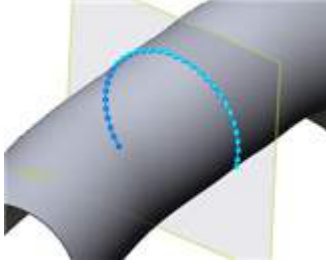
Select Entities

Target – Selects regions or poly-faces on a mesh.

Profile

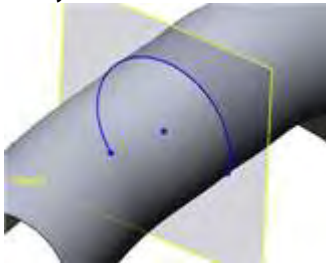
Auto Generation – Automatically finds a profile.

Freeform Curve(s) – Creates 2D spline curves as the result a profile auto generation.

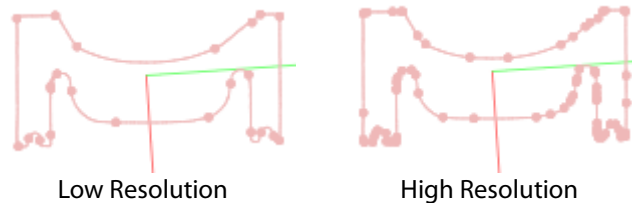


Gap Joining Tolerance – Connects separated sketch entities within a given tolerance.

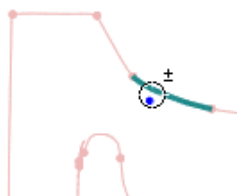
Use Line/Arcs When Possible – Uses 2D lines and arcs when sectional segments are analyzed as lines and arcs for profile entities.



Sensitivity – Selects the tolerance used to determine arcs and lines from sectional segments of a mesh. A smaller value generates a higher segment resolution.



Max. Arc Radius – Recognizes a curve with a value lower than a specified value as an arc. Curves with a value higher than the specified value will be classified as lines.



Gap Joining Tolerance – See the sub-option under the **Freeform Curve(s)** method.

Use Geometry In Specified Sketch – Uses a sketch that already exists outside of the sweep area, by transforming the sketch to the sectional coordinate of the sweep body.

Use Specified Sketch As It Is – Selects a sketch to use as a sweep profile without changing its location.

Path

Auto Generation – Automatically finds a path.

Freeform Curve(s) – Creates 3D spline curves as the result a profile auto generation.



Use Line/Arcs When Possible – Extracts lines and arcs splines when sectional segments are analyzed as lines and arcs for profile entities.



Use Specified Curve As Initial Guess – Specifies a curve to approximately set a correct sweep path. The application will generate several sectional profiles based on a selected curve to find the correct path.

Use Specified Curve As It Is – Selects a curve to use as a sweep profile without changing its location.

Second Stage Options

Cap Ends – Closes top and bottom faces. This option is available if the profile is a closed curve.

Result Operator – Cuts or merges resulting bodies with other intersecting body or bodies. This option is available when one or more bodies already exist. When the **Cut** or **Merge** options are not checked, an independent body will remain.

Profile Auto Generation Options – Refer to the first stage.

Path Auto Generation Options – Refer to the first stage.

Deform Curve – Deforms a preview network by moving a node of a curve within in an editing zone. To adjust the editing zone, hold down the **Alt** key and left-click and drag the mouse left or right.



Editing Zone

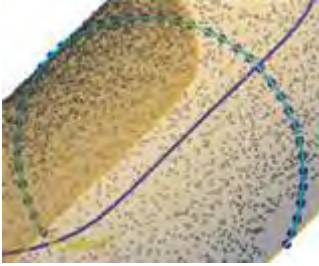


Enlarged Editing Zone

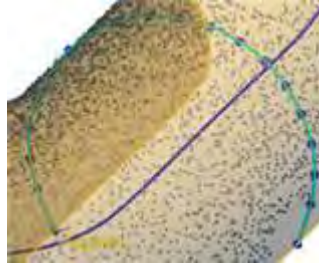
Rebuild Curve Options – Adjusts the number of node points on a freeform curve that has excessive node points.

To edit node points:

1. Click the **Target Curves** button and select splines.
2. Set the total number of points next to **No. Of Interpolation Points**.
3. Click the **Rebuild Curves** button .



No. Of Interpolation Points: 50



No. Of Interpolation Points: 20

See Also

[Insert > Modeling Wizard > Extrusion Wizard](#)

[Insert > Modeling Wizard > Revolution Wizard](#)

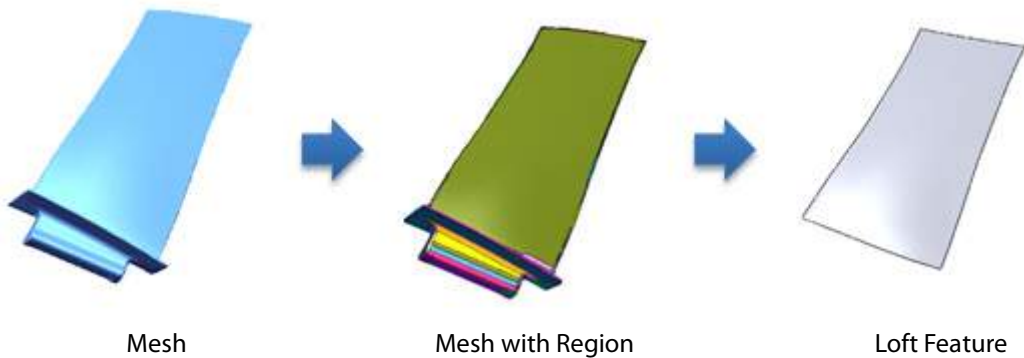
[Insert > Modeling Wizard > Loft Wizard](#)

[Insert > Modeling Wizard > Pipe Wizard](#)

Insert > Modeling Wizard > Loft Wizard

What

The **Loft Wizard** command extracts loft objects from a mesh. The wizard intelligently computes several sectional profiles and creates a loft path based on the selected regions.



Why

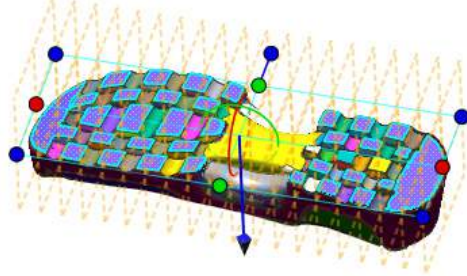
The **Loft Wizard** command is useful for:

- Generating loft surface features

How to Use

How to generate a loft feature from a mesh

1. Prepare a mesh with regions and go to **Insert > Modeling Wizard > Loft Wizard** or click on the icon in the toolbar.
2. Select regions or poly-faces where a lofted surface will be created. Check the **By Plane** option to generate sectional planes on a planar area.



3. Check the **By No. Of Sections** option and set the **No. Of Sections** to **10**. Move the Slider to **More** and manipulate the size and rotation of the preview plane in the **Model View**.

▼ Section

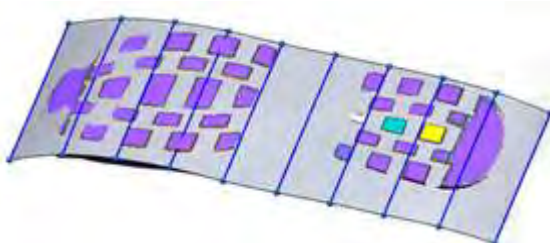
By Allowable Deviation

By No. Of Sections

No. Of Sections

Smoothness

4. Click **OK** to complete the command.



Options in Detail

Select Entities

Regions/Poly-Faces – Selects regions or poly-faces on a mesh.

Path

By Plane – Selects a planar entity for a planar loft. Profile planes will be aligned perpendicular to a selected plane.

By Curve – Selects a curve for a blended loft. Profile planes will be aligned perpendicular to a selected curve.

Section

By Allowable Deviation – Calculates the adaptable number of sections by the deviation between allowable deviation and the limited number sections.

Allowable Deviation – Specifies the allowable deviation distance between a mesh and a loft body.

Max. No. of Sections – Limits the number of created sections by a specified value.

By No. Of Sections – Calculates the loft surface by the total number of sections.

No. Of Sections – Specifies the total number of sections.

Smoothness – Determines the smoothness level of a fitting surface. Moving the slider towards **More** will create a smoother surface but can diminish accuracy.

Profile Type

3D Sketch – Creates 3D spline curves for profile entities and registers resulting sections as a **3D Sketch** in the **Feature Tree**.

Ref. Planes + Mesh Sketches – Creates 2D lines and arcs for profile entities that are analyzed as lines and arcs and registers resulting sections as **2D Mesh Sketch** with sectional planes.

Second Stage

In the next stage, section planes can be manipulated for a more accurate loft surface. The **Accuracy Analyzer™** can be used to check deviation in real-time.

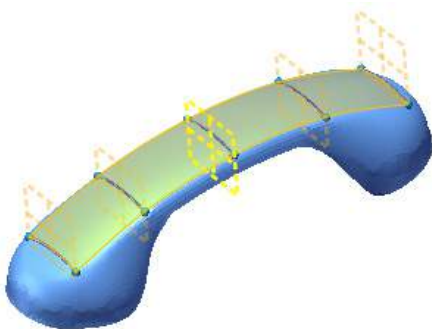
The following actions are available in the second stage:

Move – Select and Drag

Add – Ctrl + Drag

Remove – Select and Delete button

Undo – Ctrl + Z



See Also

[Insert > Modeling Wizard > Extrusion Wizard](#)

[Insert > Modeling Wizard > Revolution Wizard](#)

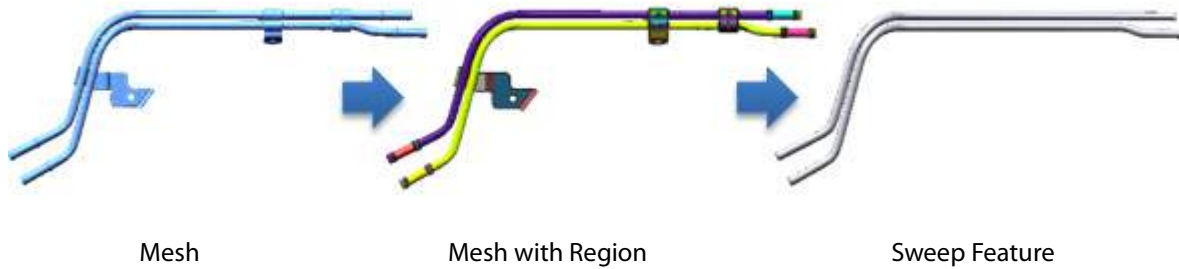
[Insert > Modeling Wizard > Sweep Wizard](#)

[Insert > Modeling Wizard > Pipe Wizard](#)

Insert > Modeling Wizard > Pipe Wizard

What

The **Pipe Wizard** command extracts pipe objects from a mesh or point cloud. The wizard intelligently computes the circular profile, or radius, and the path based on selected poly-faces or poly-vertices.



Note:

The **Pipe Wizard** command, unlike the other wizard commands, can use point clouds as target entities. When extracting pipe features, it is not necessary to generate a mesh from a point cloud.

Why

The **Pipe Wizard** command is useful for:

- Generating pipe features
- As-built modeling for construction

How to Use

How to generate a pipe feature from a mesh

1. Prepare a mesh or point cloud and go to **Insert > Surface > Pipe Wizard** or click the icon in the toolbar.



2. Choose the **Auto Selection** option and select pipe regions in the model as **Target**.



3. Click the **Next Stage** button to see the preview pipe.



4. In the **Adjust Pipe Path** stage, the curve of a pipe path can be edited by choosing the **Rebuild Curve** option and selecting a pipe path. Set the **No. Of Interpolation Points** to **50** and click the **Rebuild Curve** button. The number of nodes on the points will be reduced.

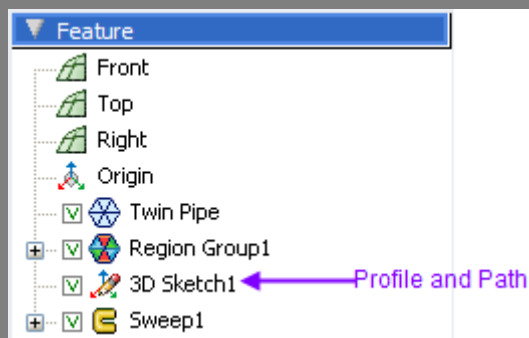


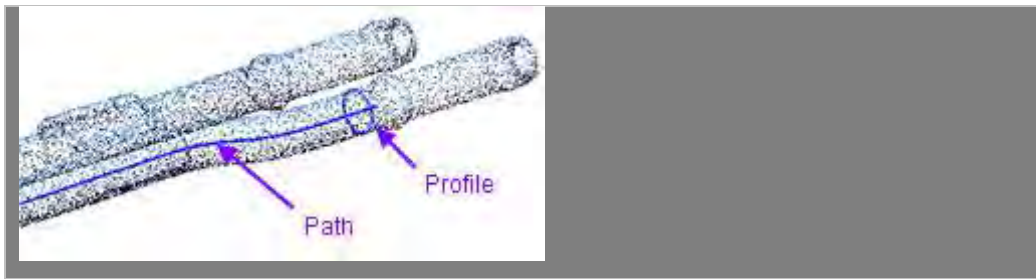
5. Click **OK** to complete the command. A pipe body will be created.



Note:

The profile and the path of a swept body are registered in the **Feature Tree**. Curves of profiles and paths can be edited for more accurate pipe body.





Options in Detail

1/3 Select Target Data

Auto Selection – Uses a selected seed point to automatically find a pipe shape in a mesh or a point cloud. For a mesh, select regions or poly-faces and for a point cloud, select poly-vertices. One depth of undo is allowable from multiple selections.



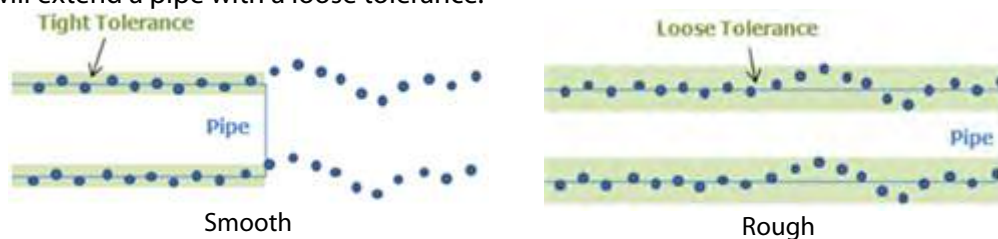
Selection on Point Cloud



Auto Searching Pipe

Data Quality – Defines the quality of scan data. When scan data is clean move the slider towards **Clean** and for noisy scan data move the slider towards **Noisy**.

Roughness – Detects the length of a pipe path. When scan data is smoothly connected without noise, move the slider towards **Smooth** and the application will detect a pipe area with a tight tolerance. When the slider is moved towards **Rough** the application will extend a pipe with a loose tolerance.



Cylinder Selection – Detects a pipe from several picked points and is useful if a pipe is split but a single pipe is desired. One depth of undo is allowable from multiple selections.



Seed Points



Detected Pipe

Target – Selects regions, poly-faces or poly-vertices of a pipe area. If a poly-vertex is picked, the application will automatically find the pipe area.

Pipe Radius – Specifies a pipe radius. When the value is **Auto**, the application will automatically find the radius of a pipe.

2/3 Select Remaining Pipe

When multiple pipe areas are selected the **Select Remaining Pipe** stage will show candidate pipes. Pipes to be generated can be selected by clicking the **Bodies** button.

3/3 Adjust Pipe Path

After generating a preview pipe, the path of the pipe can be edited.

Cap Ends – Closes top and bottom faces and generates a solid body. This option is available if the profile is a closed curve.

Result Operator – Cuts or merges resulting bodies with other intersecting body or bodies. This option is available when one or more bodies already exist. When the **Cut** or **Merge** options are not checked, an independent body will remain.

Freeform Curve(s) – Creates 3D spline curves as the result of a profile auto generation.



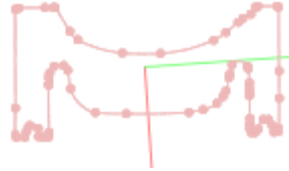
Use Line/Arcs When Possible – Uses lines and arcs splines when sectional segments are analyzed as lines and arcs.



Sensitivity – Selects the tolerance used to determine arcs and lines from sectional segments of a mesh. A smaller value generates a higher segment resolution.

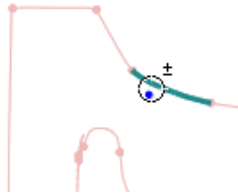


Low Resolution

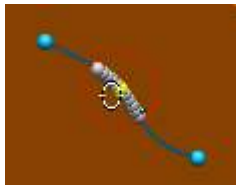


High Resolution

Max. Arc Radius – Recognizes a curve with a value lower than a specified value as an arc. Curves with a value higher than the specified value will be classified as lines.



Deform Curve – Deforms a preview network by moving a node of a curve within in an editing zone. To adjust the editing zone, hold down the **Alt** key and left-click and drag the mouse left or right.



Editing Zone

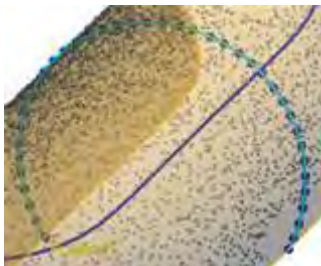


Enlarged Editing Zone

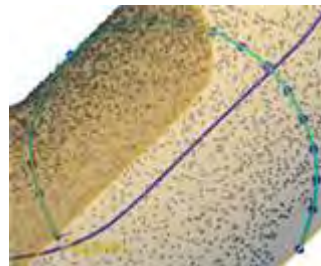
Rebuild Curve – Adjusts the number of node points on a freeform curve that has excessive node points.

To edit node points:

1. Click the **Target Curves** button and select splines.
2. Set the total number of points next to **No. Of Interpolation Points**.
3. Click the **Rebuild Curves** button .



No. Of Interpolation Points: 50



No. Of Interpolation Points: 20

See Also

[Insert > Modeling Wizard > Extrusion Wizard](#)

[Insert > Modeling Wizard > Revolution Wizard](#)

[Insert > Modeling Wizard > Sweep Wizard](#)

[Insert > Modeling Wizard > Loft Wizard](#)

7.4.9. Insert > Modeling Feature

The **Modeling Features** menu features various commands used for applying additional modifications on basic bodies to generate final features.

The **Modeling Features** are divided into groups used to:

Modify Features

- **Fillet**
Generates rounded-edge features
- **Chamfer**
Generates beveled-edge features
- **Draft**
Generates tapered features
- **Emboss**
Sculpts patterns or shapes on a body with curves

Generate Patterned Features

- **Linear Pattern**
Generates linearly copied features
- **Circular Pattern**
Generates circularly copied features
- **Curve Pattern**
Generates copied features by following a curve
- **Mirror**
Generates symmetrical features

Generate Special Features

- **Helix**
Generates helical features

- **Spiral**
Generates spiral features

Edit Bodies

- **Move Face**
Transforms faces and changes the shape of a body
- **Delete Face**
Removes faces on a body.
- **Replace Face**
Replaces faces and change the shape of a body
- **Split Face**
Adds edges on a face
- **Transform Body**
Transforms or scales bodies
- **Remove Body**
Deletes bodies

Insert > Modeling Feature > Fillet

What

The **Fillet** command generates rounded features on a solid or surface body and can be applied to edges or faces.



Why

The **Fillet** command is useful for:

- Generating a rounded edge part

How to Use

A. How to generate constant fillets with different radii

1. Go to **Insert > Modeling Feature > Fillet** or click the icon in the toolbar.
2. Choose the **Constant Fillet** method.



3. Select outer edges and set the radius to **2mm**.



4. To add a larger radius click the **Add Radius** button and select inner edges. Set the radius to **1mm**.

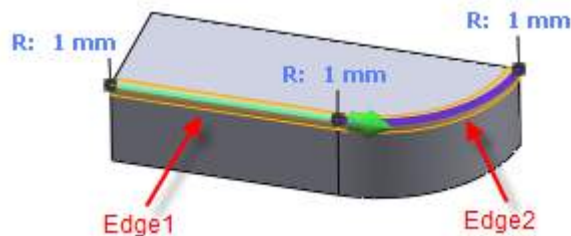


5. Click **OK** to complete the command.

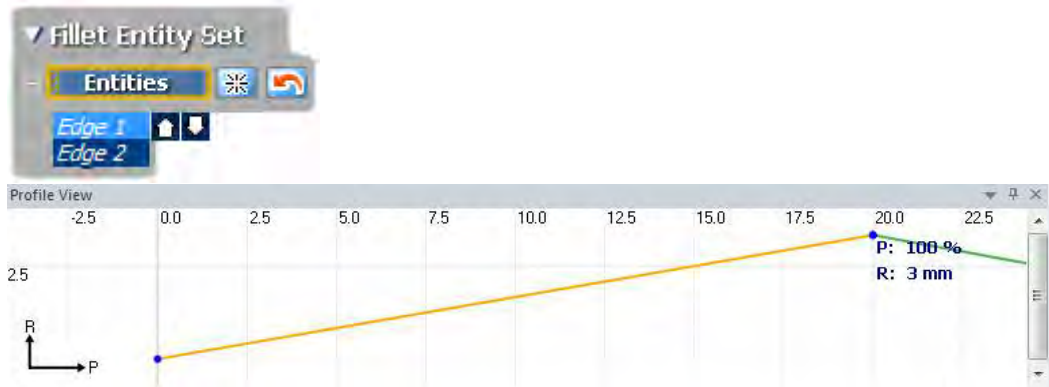


B. How to generate variable fillets

1. Go to **Insert > Modeling Feature > Fillet** or click the icon in the toolbar.
2. Choose the **Variable Fillet** method and select two edges.



3. Select **Edge 1** in the list box of **Entities** and change the end fillet radius marked as **R** to **3mm** in the **Profile View**.

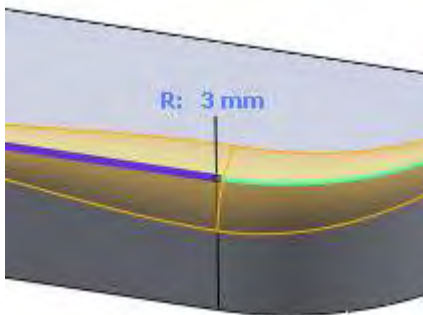


Note:

The **Up** and **Down** icons can be used to change the order of entities.



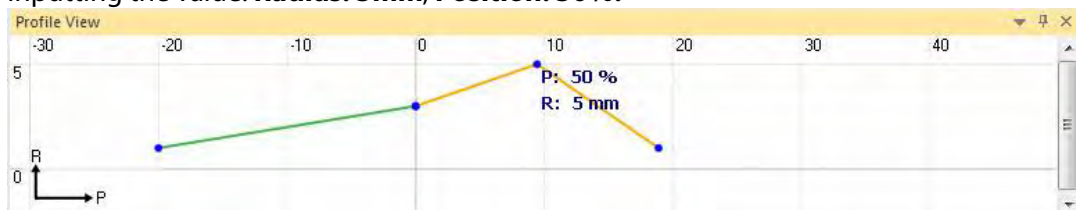
A selected edge will show a preview of the fillet.



4. Select **Edge 2** in the list box.



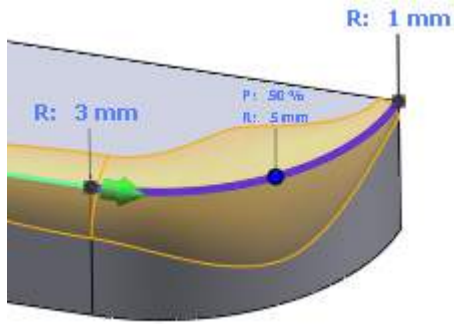
Click in the middle of the yellow line to add a control point and edit it by dragging or inputting the value: **Radius: 5mm, Position: 50%.**



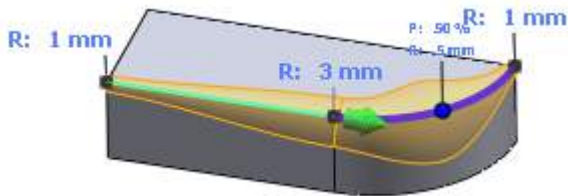
Note:

To zoom in or out of a graph in the **Profile View** or scroll the mouse wheel.

The selected edge shape will change to reflect the inputted values.



5. The preview will also change.



6. Click **OK** to complete the command.




Options in Detail

Methods

Constant Fillet – Creates constant radius fillets by selecting edges or faces and specifying the fillet radius.

If multiple radii are required click the **Add Radius** button , select more edges or faces, and specify the new radius.

To remove a fillet entity, click the **Remove Radius** button . Edges, loops or faces can be selected as **Entities**.

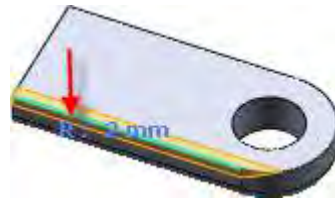
Use the **Estimate Radius From Mesh** button  to automatically extract the approximate fillet radius from a mesh. This button can only be used if meshes exist.



Tangent Propagation – Extends a selection to tangentially connected adjacent edges.

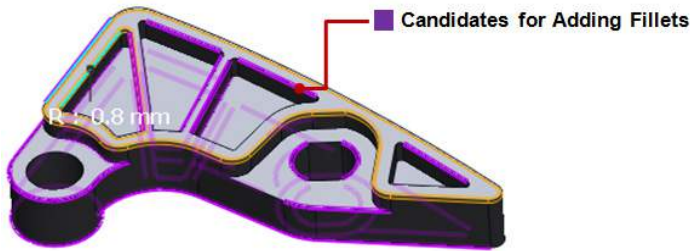


With Tangent Propagation





Without Tangent Propagation

Selection Guide – Finds candidate edges that have similar fillet radii from an input radius. The candidate edges will be highlighted on a body. When multiple fillet sets exist, only the focused set's radius will be highlighted. The range of the total number of candidate edges can be modified.

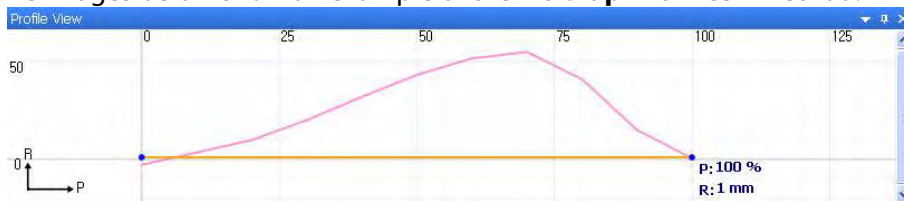


Variable Fillet - Creates variable radius fillets by specifying different radii on an edge and will initiate the **Profile View** in the console window. A control point can be added by clicking a specific location on the yellow line and the radius can be set and positioned by dragging a blue point or clicking the widget to specify a numerical value. To remove a control point, select the point and click the **Delete** key or move the point to another point and combine them.

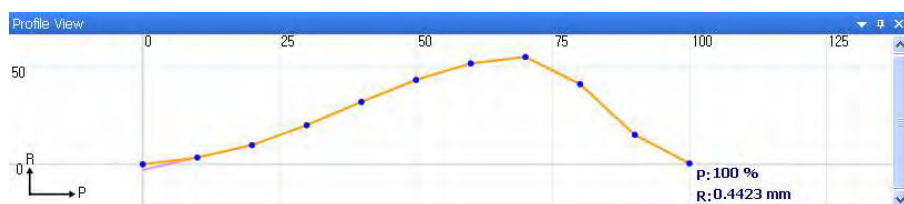


The **Estimate Radius From Mesh** button  automatically extracts variable radii from a mesh and shows the radius flow in the **Profile View**. To apply variable radii automatically, click the down arrow button  in the **Profile View** and choose the **Fit Graph To Mesh** method.

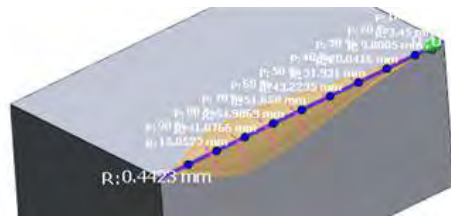
The images below show an example of the **Fit Graph To Mesh** method.



Before Fit Graph To Mesh



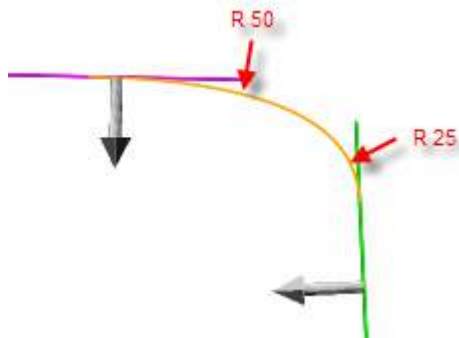
After Fit Graph To Mesh



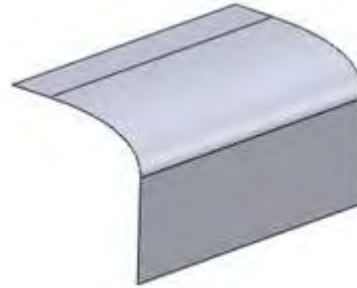
The Result on Body

Tangent Propagation – Refer to the **Constant Fillet** method.

Conic Fillet –Creates a blending area with different radii to each individual face. This option is available under the **Variable Fillet** and **Face Fillet** methods.



Conic Fillet: R 50 and R 25



The Result of Conic Fillet

Rho – Determines the shape of a blending area with a possible value between 0 and 1. The higher the value to 1, the sharper the blending shape, and the lower the value to 0, the flatter the blending shape.



Rho = 0.3

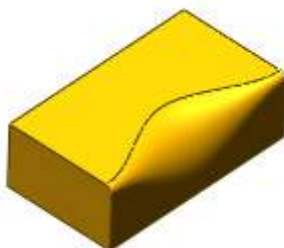


Rho = 0.9

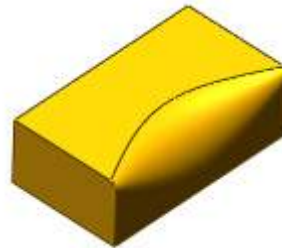
Transition Method – Selects the position of the start and end of a fillet.

Smooth Transition creates a fillet which smoothly flows from one radius to another when matching a fillet edge to an adjacent face.

Linear Transition creates a fillet which linearly flows from one radius to another without matching edge tangency with an adjacent fillet.



Smooth Transition



Linear Transition

Face Fillet – Selects faces to fillet. Non-adjacent faces can be blended with a fillet face.

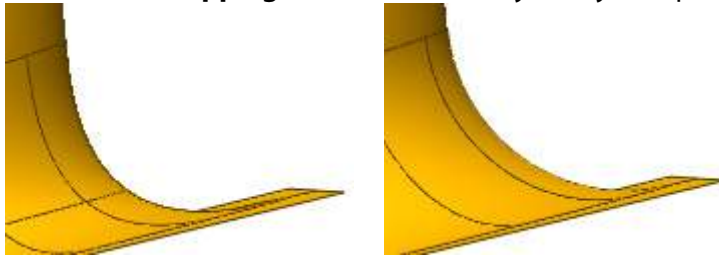




Tangent Propagation – Refer to the **Constant Fillet** method.

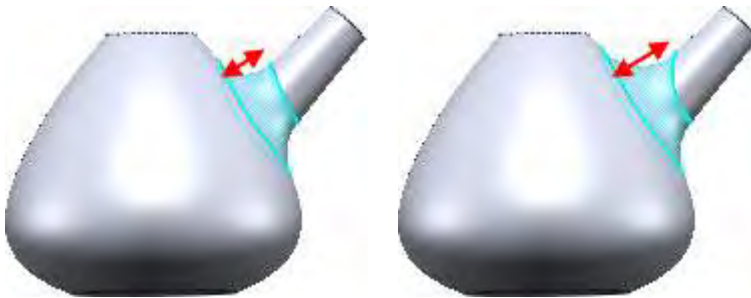
Conic Fillet – Refer to the **Variable Fillet** method.

Constant Curvature – Creates a smoother curvature fillet than a standard fillet. This method will create no jump in curvature flow at boundaries. To verify result use **Environment Mapping** under the **Accuracy Analyzer™** panel.



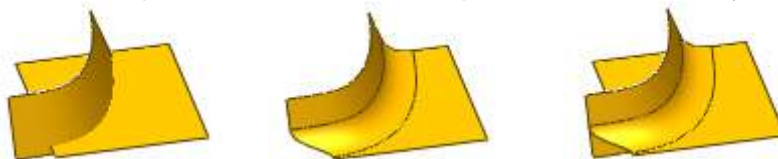
Enable Constant Curvature Disable Constant Curvature

Constant Width – Creates fillet face with constant width.



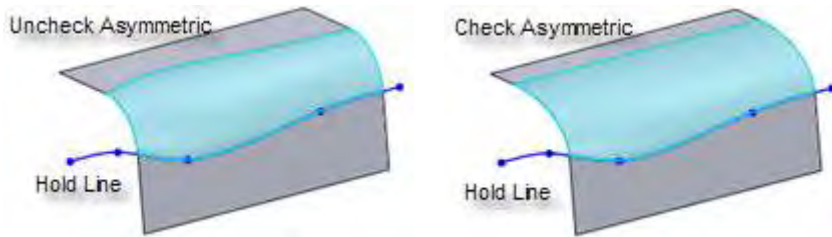
Enable Constant Width Disable Constant Width

Trim and Merge Result – Trims and merges faces into one body after filleting.



Original Faces Check Option Uncheck Option

Hold Line – Selects a body edge or curve as a boundary for where a face fillet shape overfills a hold line. The radius of a fillet is driven by the distance between the hold line and the edge to be filleted. The **Asymmetry** option create a symmetry hold line opposite to a fillet shape and is useful for creating different hold lines.



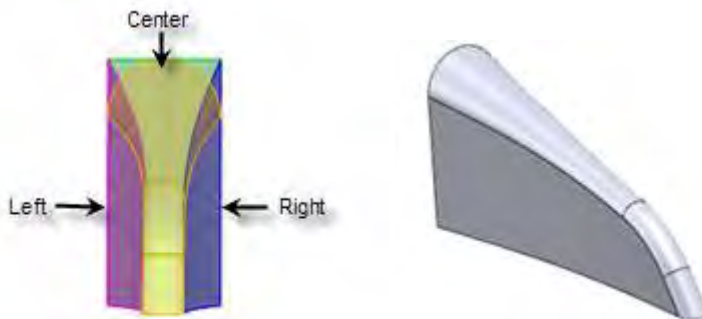
Help Point – Resolves an ambiguous selection when it may not be clear where a face blend should occur. Click the **Help Point** button and pick a vertex on the side where a face fillet should be. A fillet will be created at the location closest to the Help point.

Full Face Fillet – Creates fillets that are tangent to three adjacent faces.

Left – Selects the left face.

Center – Selects the center face.

Right – Selects the right face that is opposite to the **Left** face.



Tangent Propagation – Refer to the **Constant Fillet** method.

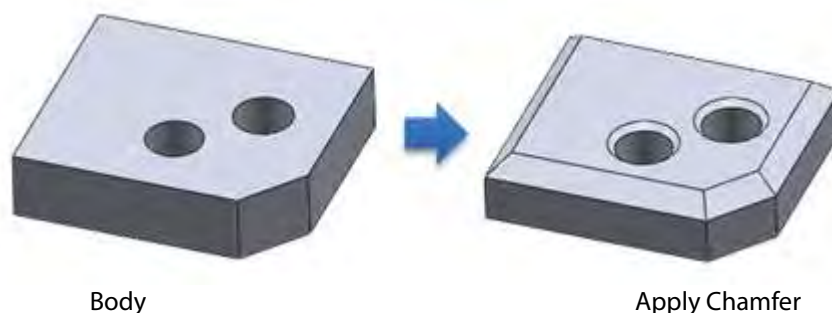
See Also

[Insert > Modeling Feature > Chamfer](#)

Insert > Modeling Feature > Chamfer

What

The **Chamfer** command generates a beveled feature on edges or faces of a solid or surface body.



Why

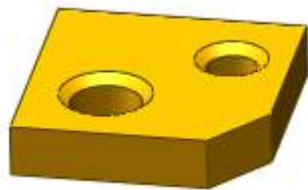
The **Chamfer** command is useful for:

- Generating a beveled part

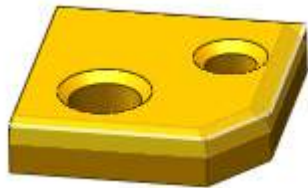
How to Use

How to apply chamfer to a body using the angle and distance conditions

1. Go to **Insert > Modeling Feature > Chamfer** or click the icon in the toolbar.
2. Choose the **Angle And Distance** method and set the **Distance** to **5 mm** and the **Angle** to **45 degrees**.



3. Select edges to chamfer. A preview or the resulting chamfer will be shown.




4. Click **OK** to complete the command.

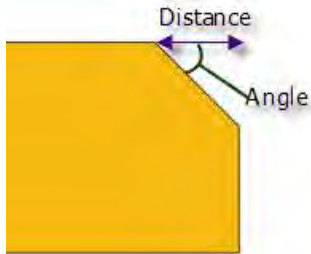


Options in Detail

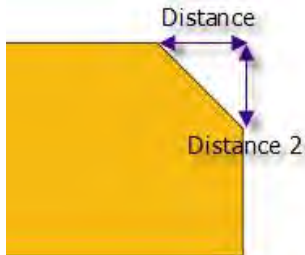
Methods

Entities – Selects edges, faces or loops.

Angle And Distance – Generates a beveled face by using **Distance** and **Angle** that are specified from edges. The **Estimate Distance From Mesh** button , automatically extracts an approximate value from a mesh but is only available if meshes already exist.



Distance And Distance – Generates a bevel face by using two **Distances** that are specified from edges.



Common Options

Flip Direction – Flips a chamfer direction.



Before Flip Direction



After Flip Direction

Tangent Propagation – Extends a selection to tangentially connected adjacent edges.

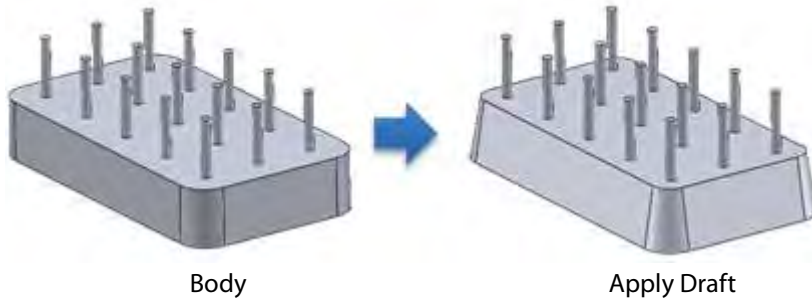
See Also

[Insert > Modeling Feature > Fillet](#)

Insert > Modeling Feature > Draft

What

The **Draft** command tapers faces on a solid or surface body using a specified angle.



Why

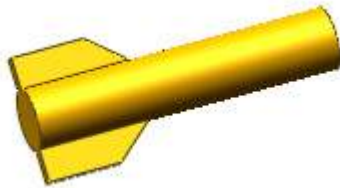
The **Draft** command is useful for:

- Tapering to ribs, bosses, walls, and other entities on a molded part
- Generating a tapered feature

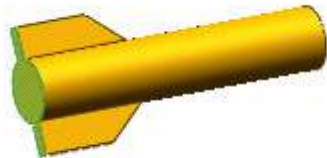
How to Use

How to apply a draft to a solid body

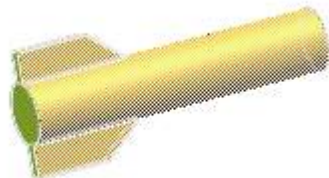
1. Go to **Insert > Modeling Feature > Draft** or click the icon in the toolbar.



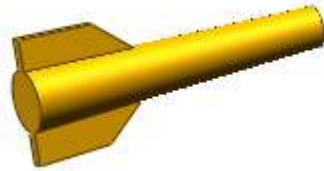
2. Select a bottom face as the **Neutral Plane**.



3. Select a side faces as the **Draft Faces**.



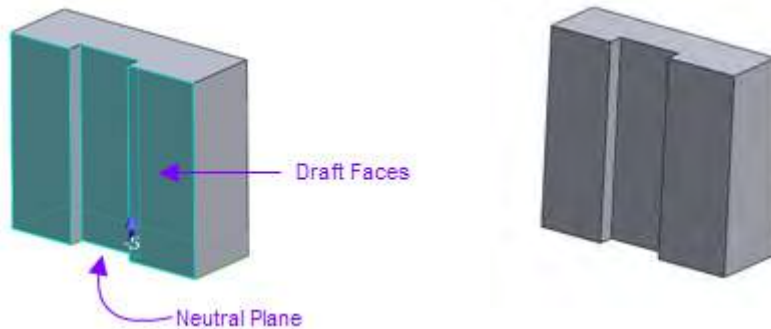
4. Set the **Angle** to **2** degrees and click the **Preview** button. To change a draft direction, click the **Flip** button. When the result in the preview are satisfactory, click **OK** to complete the command.



Options in Detail

Methods

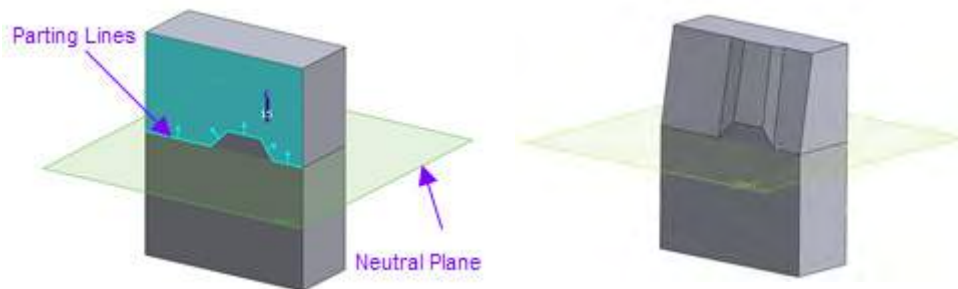
Neutral Plane Draft – Creates a draft to selected faces based on a neutral plane.



Neutral Plane – Determines the pull direction of a model by using the normal of a selected entity's such as a ref. plane, region or face.

Draft Faces – Selects faces to be drafted.

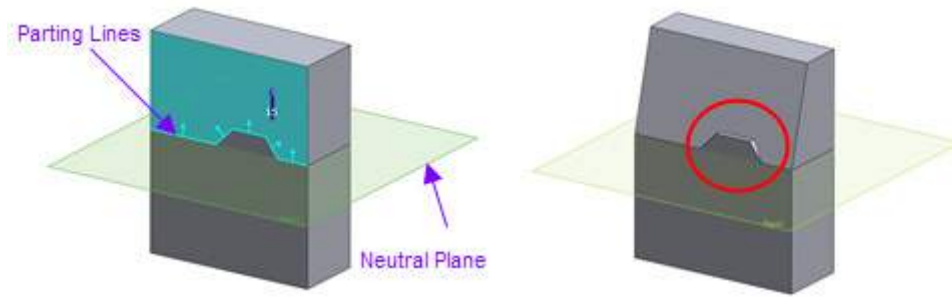
Parting Line Draft – Drafts surfaces around a parting line. The parting line can be non-planar.



Pull Direction – Determines the pull direction of a model by using a selected linear entity's direction or planar entity's normal.

Parting Lines – Select edges on a body.

Step Draft – Uses a variation of the parting line draft method to draft surfaces by creating one face rotated around the plane used as a direction of a Neutral Plane. The results are small faces, highlighted in red circle in the example below.

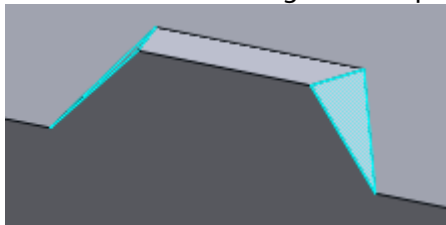


Neutral Plane – Determines the pull direction of a model by using the selected entity's normal.

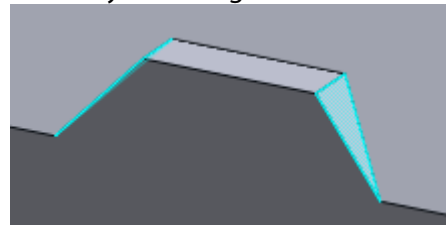
Parting Lines – Selects edges on a body.

Tapered Steps – Faces that have a normal that is not in the same direction as the pull direction of a model are generated in the same manner as tapered faces.

Perpendicular Steps – Faces that have a normal that is not the same as the pull direction of a model are generated perpendicularly to an original main face.





Tapered Steps – Highlighted faces are drafted



Perpendicular Steps – Highlighted faces are not drafted

Draft Parameters

Angle – Specifies a draft angle. To change draft direction, click the **Flip** button . The **Estimate Angle From Mesh** button  automatically extracts an angle based on a mesh.

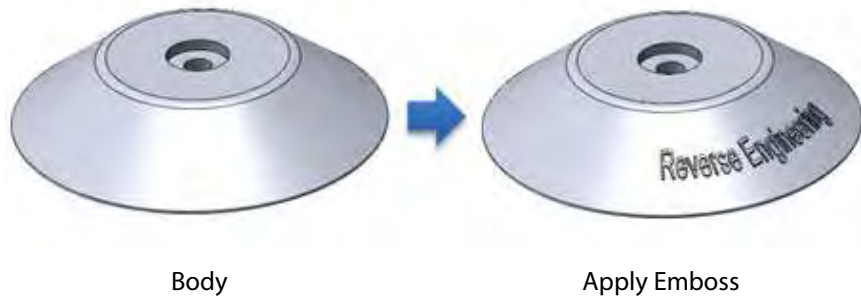
See Also

[Insert > Solid > Extrude](#)
[Insert > Surface > Extrude](#)

Insert > Modeling Feature > Emboss

What

The **Emboss** command wraps a 2D or 3D sketch onto a planar or a non-planar face and generates a convex or concave feature.



Why

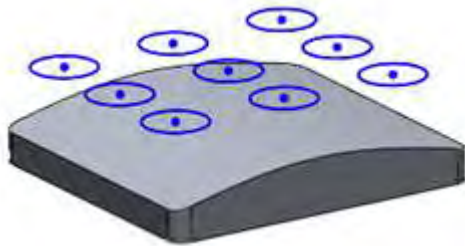
The **Emboss** command is useful for:

- Generating an extruded feature on a non-planar surface
- Generating an engraved object
- Marking a company name or a part name onto a feature

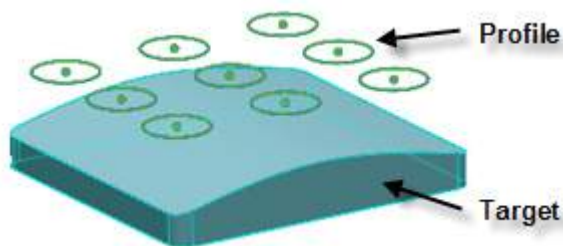
How to Use

How to apply an emboss to a body

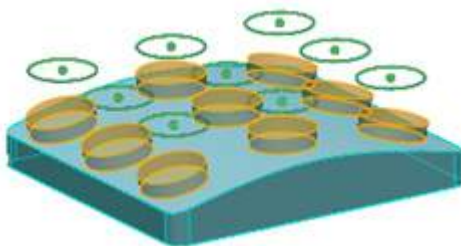
1. Go to **Insert > Modeling Feature > Emboss** or click the icon in the toolbar.



2. Choose the **Pad** option and select curves as **Profile** and a solid body as **Target**.



3. Set the **Depth** to **5 mm** and click the **Preview** button.



4. Click **Accept** to complete the command.

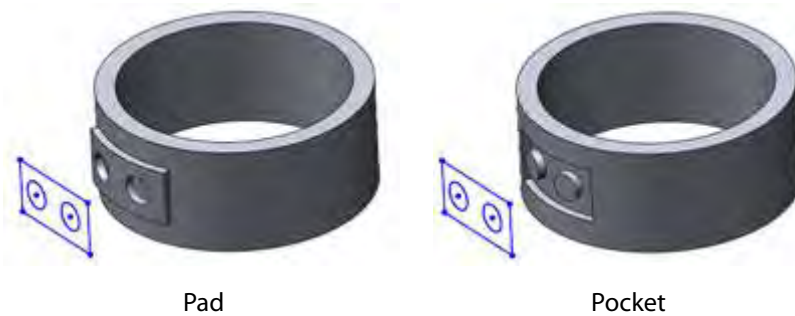


Options in Detail

Methods

Pad – Creates a convex feature.

Pocket – Creates a concave feature.



Select Entities

Profile – Selects 2D or 3D curves. Multiple loops can be selected simultaneously.

Target - Selects a face on a solid or surface body.

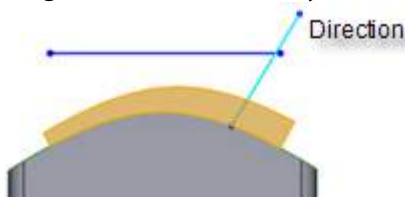
End Cap Surface

Offset – Offsets the top face of an embossed feature from a target face.



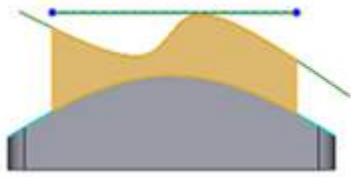
Depth – Specifies the height of an embossed feature.

Translate – Translates the top face of an embossed feature as a selected direction from a target face. A linear entity or an entity which has a normal as **Direction** can be used.



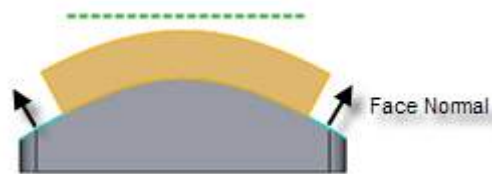
Depth – Specifies the height of an embossed feature.

Choose Other - Selects the top face of an embossed feature as an entity. A region, face, surface body or solid body can be selected as an **End Cap**.

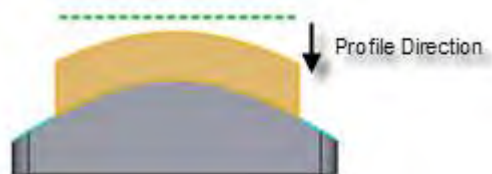


Side Wall

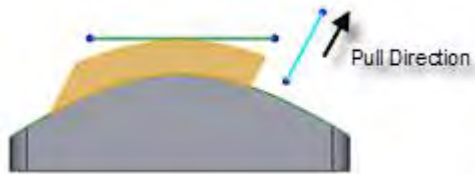
Face Normal – Sets the side face of an embossed feature in the face normal direction.



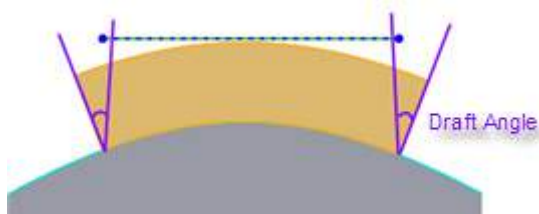
Profile Direction – Sets the side face of an embossed feature in the profile's normal direction.



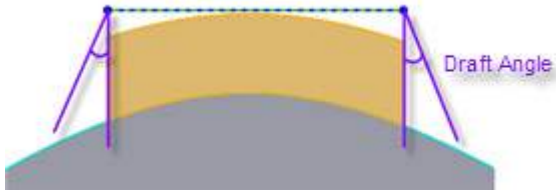
Custom Vector – Sets the side face of an embossed feature in a selected direction. A linear entity or an entity that has the same normal as the **Pull Direction** can be selected.



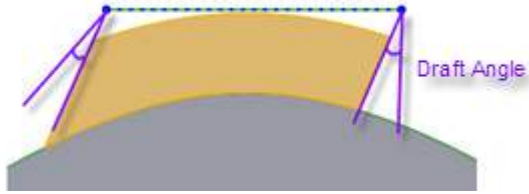
Draft – Specifies the draft angle of an embossed feature. The **Draft** option also for the **Side Wall** option.



Side Wall: Face Normal



Side Wall: Project Direction



Side Wall: Direction Vector

See Also

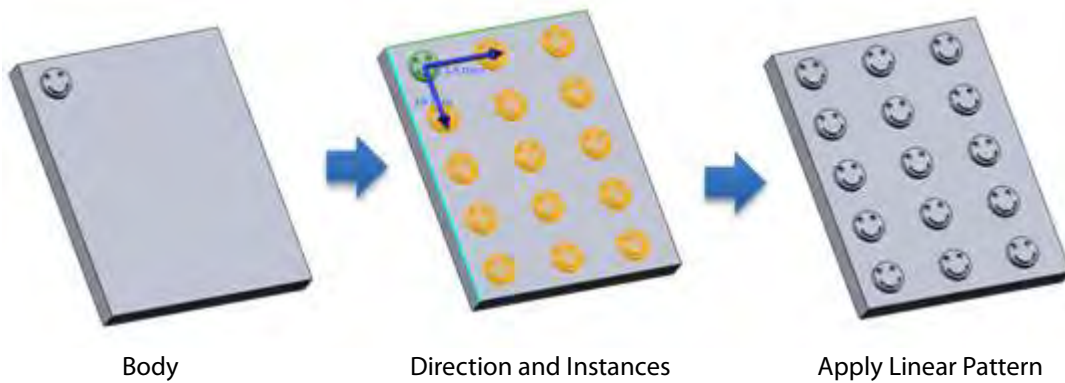
See Also

[Insert > Surface > Extrude](#)

Insert > Modeling Feature > Linear Pattern

What

The **Linear Pattern** command generates multiple features in a linear direction at one time.



Body

Direction and Instances

Apply Linear Pattern

Why

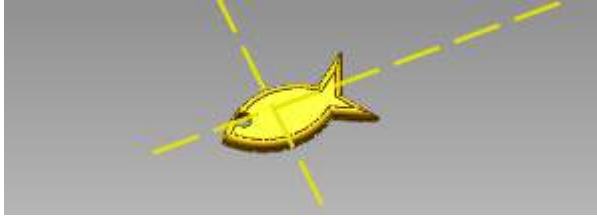
The **Linear Pattern** command is useful for:

- Generating patterned features along linear directions

How to Use

How to pattern a body along two directions

1. Prepare a body and two directional vectors.



2. Go to **Insert > Modeling Feature > Linear Pattern** or click the icon in the toolbar.

3. Select a body for **Bodies** and select a vector for **Direction1**. Set **Instance** to **3** and **Distance** to **25 mm**.

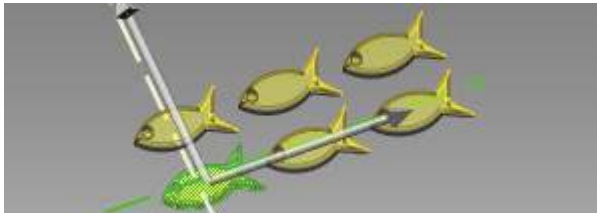


Note:

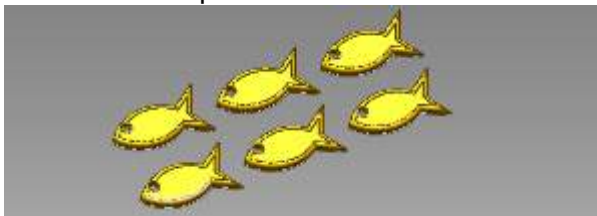
The blue distance arrow can be manipulated.



4. Select another vector as **Direction2**. Set **Instance** to **2** and **Distance** to **15 mm**.



5. Click **OK** to complete the command.



Options in Detail

Select Entities

Bodies – Selects solid or surface bodies.


Direction 1 – Selects a linear entity such as an edge or vector, or an entity with a normal such as a plane or face, for a pattern direction.

Set Instances

No. Of Instances – Sets the total number of objects for **Direction 1**.

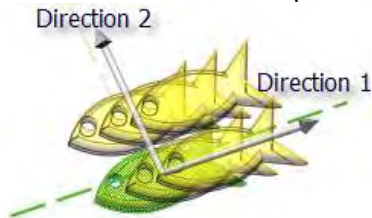


No. Of Instances: 5

Distance – Sets the spacing between pattern instances. The **Reverse Direction** button  or the on-screen arrow will reverse the direction.

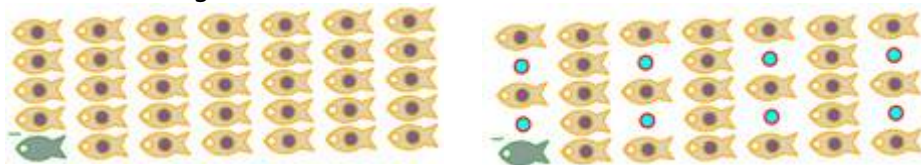


Direction 2 – Creates a pattern in a second, opposite direction from **Direction 1**.



Skip Instances

Skip Instances – Removes pattern instances by clicking on them. To restore a removed instance click it again.



See Also

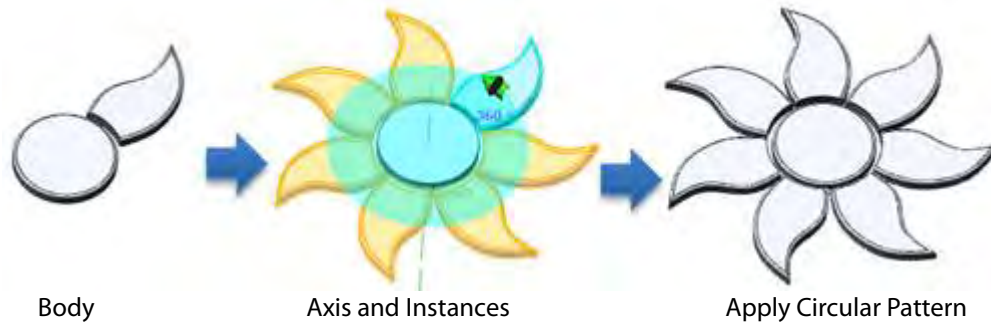
[Insert > Modeling Feature > Circular Pattern](#)

[Insert > Modeling Feature > Curve Pattern](#)

Insert > Modeling Feature > Circular Pattern

What

The **Circular Pattern** command generates multiple features around an axis at one time.



Why

The **Circular Pattern** command is useful for:

- Generating patterned features along an axis

How to Use

How to pattern a body along an axis

1. Prepare a body and a vector for a rotational axis.



2. Go to **Insert > Modeling Feature > Circular Pattern** or click the icon in the toolbar.
3. Select a body for **Bodies** and a vector for **Rotation Axis**. Set **No. of Instances** to **6** and **Total Angle** to **360**. Check **Equal Spacing**.



4. Click **OK** to complete the command.



Options in Detail

Select Entities

Bodies – Selects solid or surface bodies.

Rotation Axis – Selects a linear entity such as an edge or vector, or an entity that has an axis such as a cylinder, cone, or circular edge for a rotational axis.

Set Instances

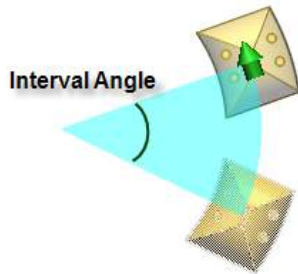
No. Of Instances – Sets the total number of objects.



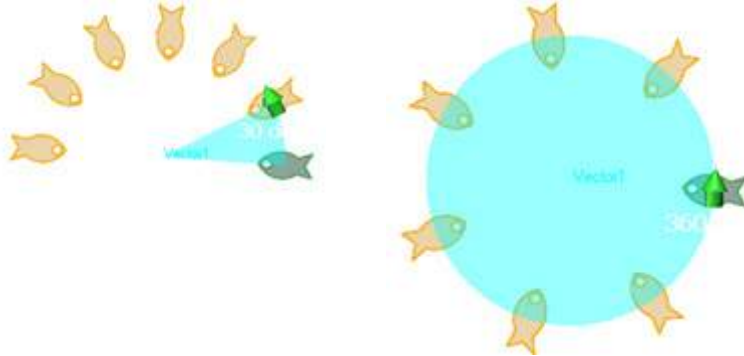
No. Of Instances: 3

No. Of Instances: 5

Interval Angle – Sets the angle between instances. To reverse the direction, click the **Flip** button or click the green arrow on the screen.



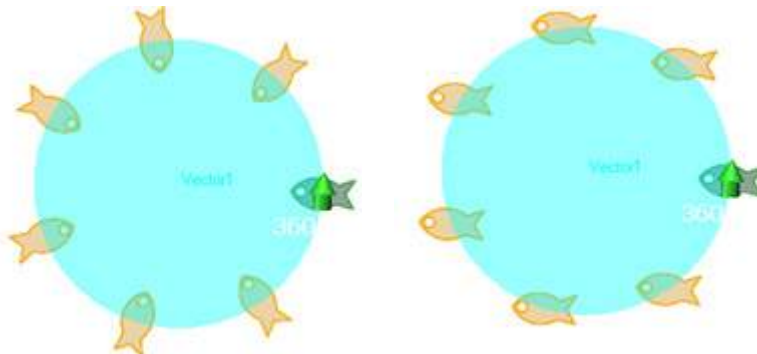
Equal Spacing – Sets equal distances between instances within a total angle.



Without Equal Spacing

With Equal Spacing

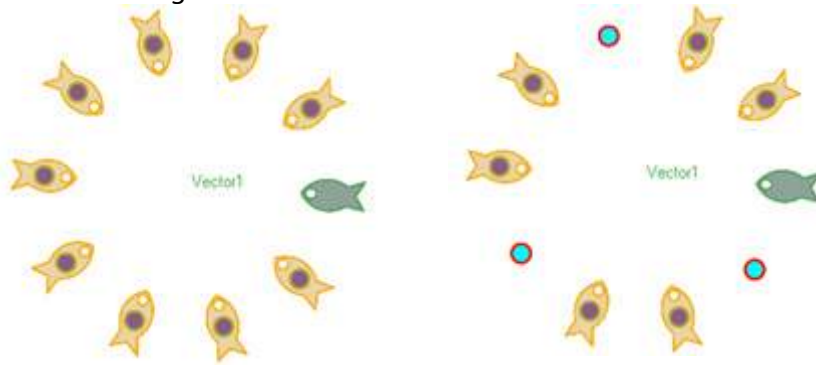
Rotate About Axis – Rotates instances around a rotation axis.



Without Equal Spacing

Without Equal Spacing

Skip Instances – Removes pattern instances by clicking on them. To restore a removed instance click it again.



See Also

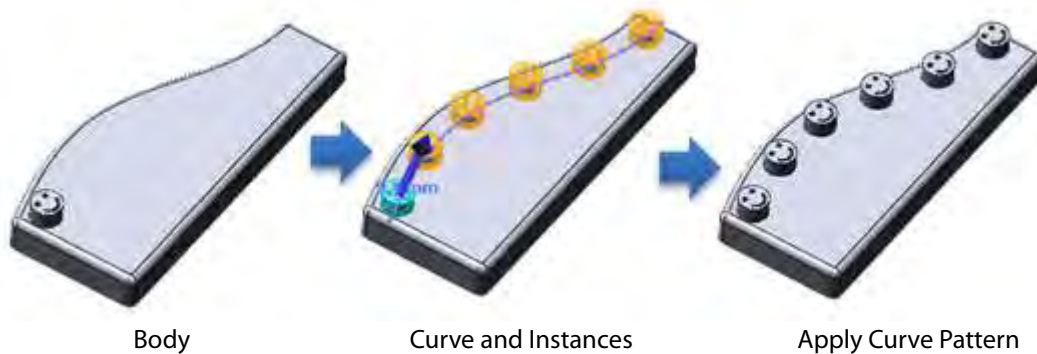
[Insert > Modeling Feature > Linear Pattern](#)

[Insert > Modeling Feature > Curve Pattern](#)

Insert > Modeling Feature > Curve Pattern

What

The **Curve Pattern** command generates multiple features around a curve at one time.



Why

The **Curve Pattern** command is useful for:

- Generating patterned features along a curve

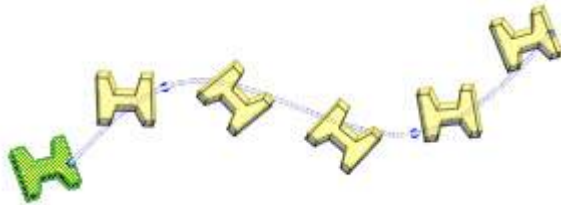
How to Use

How to pattern a body along a curve

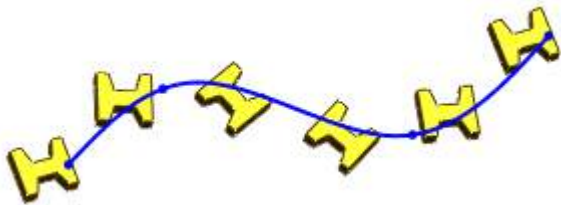
1. Prepare a body and a curve.



2. Go to **Insert > Modeling Feature > Curve Pattern** or click the icon in the toolbar.
3. Select a body for **Bodies** and a curve for **Path Curve**. Set **No. of Instances** to **6** and check **Equal Spacing**.



4. Click **OK** to complete the command.



Options in Detail


Select Entities

Bodies – Select solid or surface bodies.

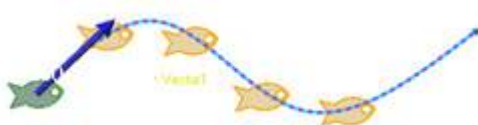
Path Curve – Selects a curve or edge for the path curve.

Set Instances

No. Of Instances – Sets the total number of objects.

Distance - Sets the spacing between pattern instances. The **Reverse Direction** button  or the on-screen arrow will reverse the direction.

Equal Spacing – Sets equal distances between instances along a selected curve.



Without Equal Spacing



With Equal Spacing

Options

Align To Seed – Aligns each instance to match the original alignment of the seed feature.

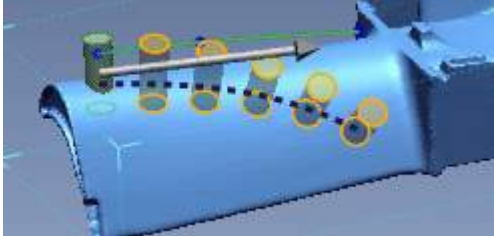


Align To Seed

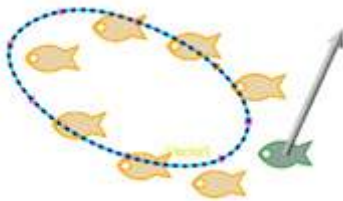


Without Align To Seed

Rotate Along Curve – Aligns the direction of instances to a selected face or mesh normal. Click the **Normal Reference** button and then select a face or a mesh.



Transform Curve – Maintains the delta X and delta Y distances from the origin of the selected curve to the seed feature for each instance. This option is available with a 2D curve.



Offset Curve – Maintains the normal distance from the origin of a selected curve to the seed feature for each instance. This option is available with a 2D curve.



Skip Instances

Skip Instance – Removes pattern instances by clicking on them. To restore a removed instance click it again.



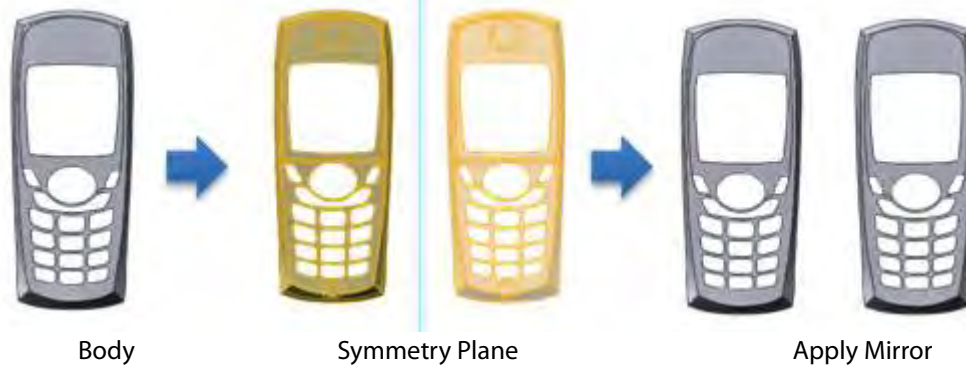
See Also

[Insert > Modeling Feature > Linear Pattern](#)
[Insert > Modeling Feature > Circular Pattern](#)

Insert > Modeling Feature > Mirror

What

The **Mirror** command generates copied features that are mirrored at a plane or a planar entity.



Why

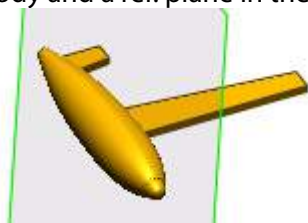
The **Mirror** command is useful for:

- Generating a symmetrical feature
- Copying a feature using a plane

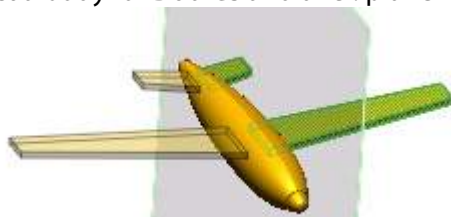
How to Use

How to generate a symmetrical body

1. Prepare a body and a ref. plane in the middle of a body.



2. Go to **Insert > Modeling Feature > Mirror** or click the icon in the toolbar.
3. Select a body for **Bodies** and a ref. plane for **Symmetry Plane**.



4. Click **OK** to complete the command.




Options in Detail

Select Entities

Bodies – Selects solid or surface bodies.

Symmetry Plane – Selects a ref. plane or a planar face as a mirroring plane.

Options

Trim And Merge – Removes overlapping areas and sews them together if a mirrored feature and original body are overlapping. Clicking the **Tangent Continuous Face** button and selecting faces will tangentially sew them. A trimmed area can be flipped by using the **Reverse Clipping Direction** button .



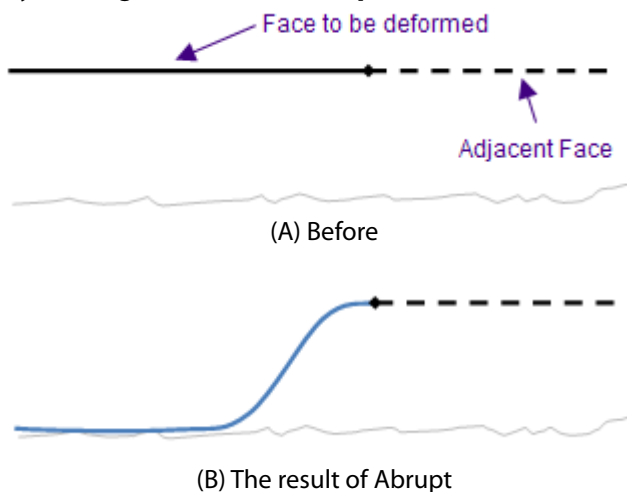
Without Trim And Merge

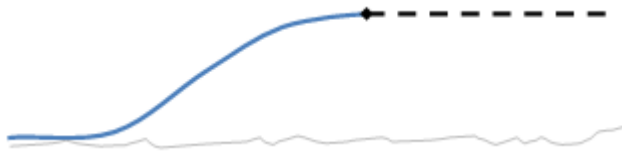


With Trim And Merge

Advanced Continuity Options

Shape Transition To Adjacent Faces – Centralizes the deformation of a face around an edge by moving the slider to **Abrupt**.





(C) The result of Gradual

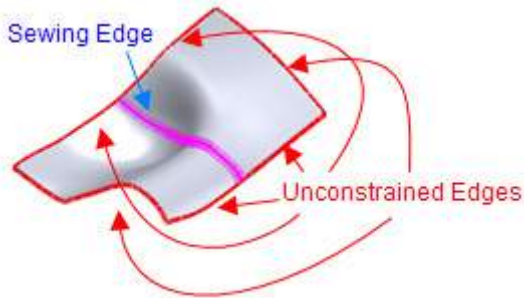
Boundary Accuracy – Determines accuracy when sewing edges. Moving the slider to **Tight**, will create more points on an edge and will attempt to tightly sew them. The **Preview** button shows the maximum deviation between edges.



Loose

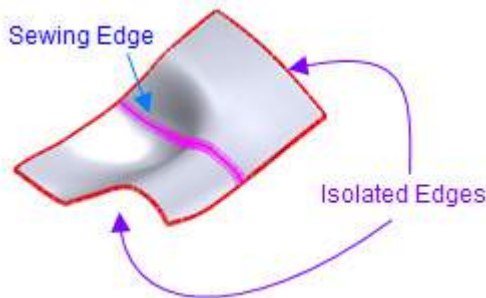
Tight

Unconstrained Edge – Determines the deformation method for non-sewing edges that are unconstrained.



Do Not Preserve – Does not preserve unconstrained edges which may deform them.

Preserve Isolated Edges Only – Preserves isolated edges that are not adjacent to sewing edges.



Preserve All – All edges will not be deformed.

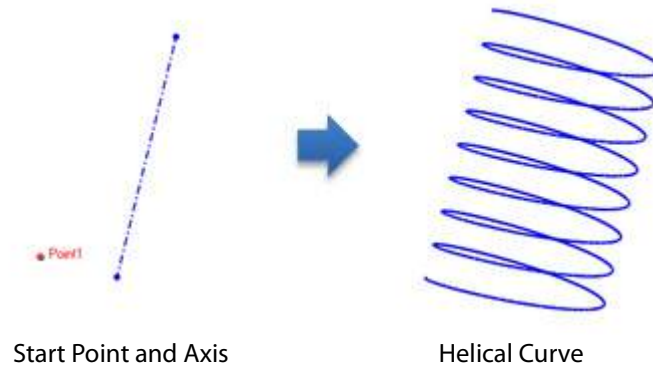
See Also

[Tools > Sketch Tools > Mirror](#)

Insert > Modeling Feature > Helix

What

A **Helix** is a three-dimensional curve that has a tangent line at any point that is at a constant angle with an axis. The **Helix** command easily generates a helical curve.



Why

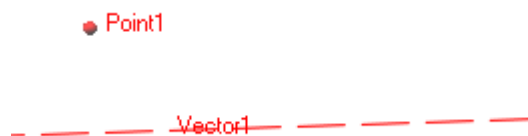
The **Helical** command is useful for:

- Sweeping or lofting a path
- Generating screw shapes
- Generate coil spring shapes

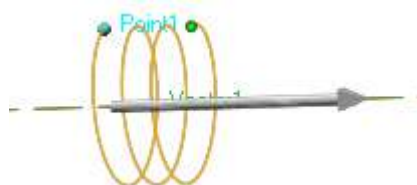
How to Use

How to generate a helical curve

1. Prepare a point as a starting position and a vector as an axis.



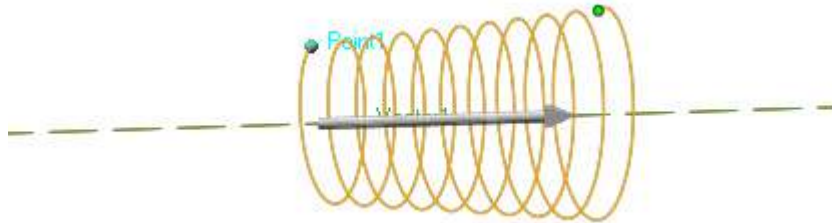
2. Go to **Insert > Modeling Feature > Helix** or click the icon in the toolbar.
3. Choose the **Constant** method. Select a vector as an **Axis**.
4. Select a ref. point as the **Start**.



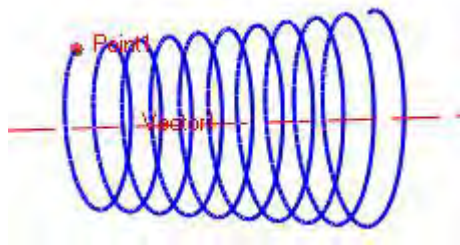
- Click the end point on the graph in the **Profile View** and set the **Height, Radius, Pitch** and **Revolution**. The blue point can be dragged or specified by a numerical value in each field.



Whenever a value changes the resulting curve will be displayed on the screen.



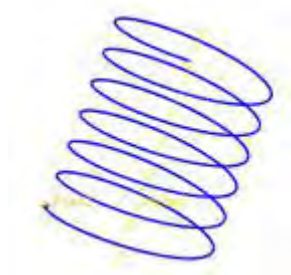
- Click **OK** to complete the command.



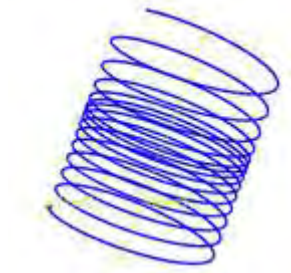
Options in Detail

Methods

Constant – Creates a constant pitch throughout a helix.



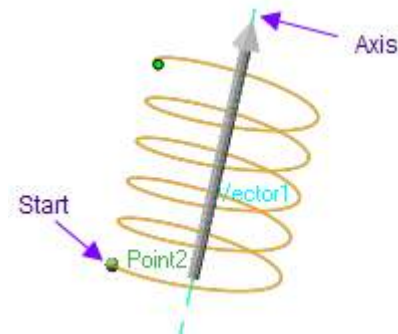
Variable – Creates a variable pitch based on specified parameters.




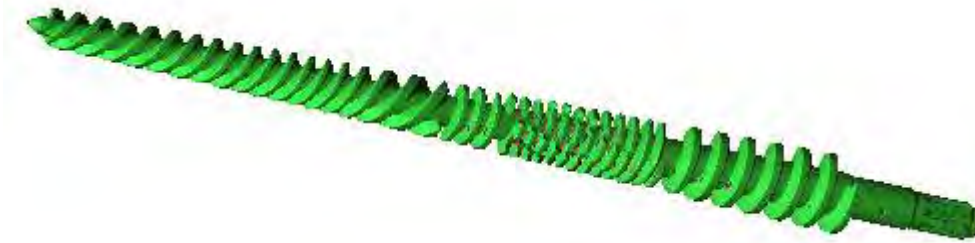
Select Entities

Axis – Selects a linear entity for the axis of a helix.

Start – Selects a point entity for the starting position of a helix.



Plot Mesh Section As Guide – Uses a section profile to extract the design intent of helix parameters. Clicking the  button will display the section profile of a mesh in the **Profile View**. This option is available if a mesh exists already.



(A) Mesh



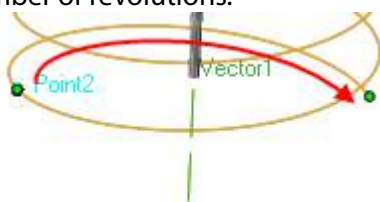
(B) Extracted Mesh Section

Options

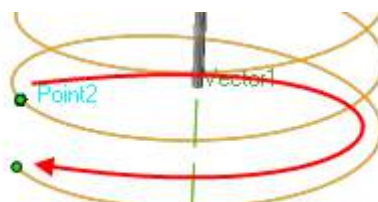
Counter Clockwise – Sets a helical direction as counterclockwise.

Clockwise – Sets a helical direction as clockwise.

Extend To Opposite Direction – Extends a helix curve to the opposite direction by a specified number of revolutions.



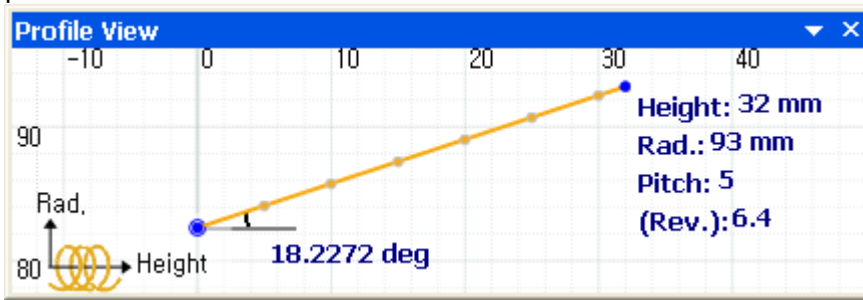
Revolution: 0.5



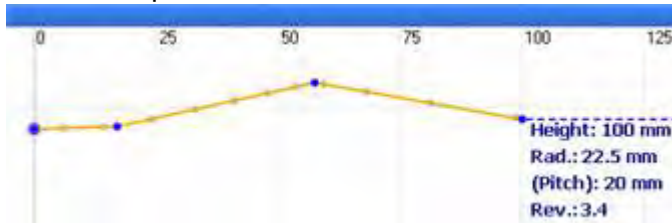
Revolution: 1

Controlling Profile View

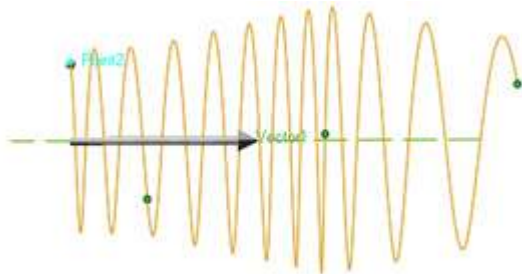
Used to define helix parameters in the **Profile View**. Clicking a node point will display editable parameters.



- The horizontal axis represents **Height** and the vertical axis represents **Radius**.
- The left blue point represents the starting position and the right blue point represents the ending position.
- A node point can be dragged or directly specified on the widget by clicking it.
- To remove an added node, move the node onto another point and overlap it or by pressing the **Delete** key.
- The **Variable** option allows for the addition of more node points by clicking on the graph.



(A) Variable Options Parameters



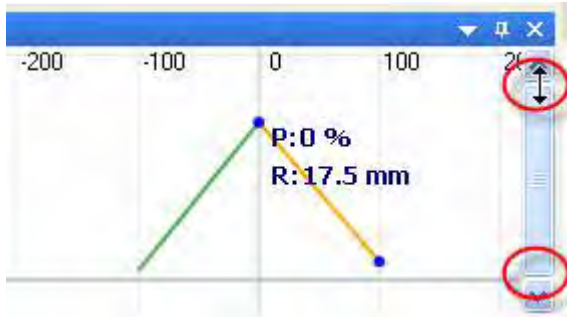
(B) The Result of Helical Curve

Viewing in Profile View

Zoom – Use the mouse wheel.

Pan – Use the left mouse button or the scroll bar in the Profile View window.

Scale (Y Direction) – Use Shift + left mouse button or move the top or bottom edge of the scroll bar.



Note:

The  button at the right top corner of the **Profile View**, displays additional functions.

- Zoom Fit
- Reset

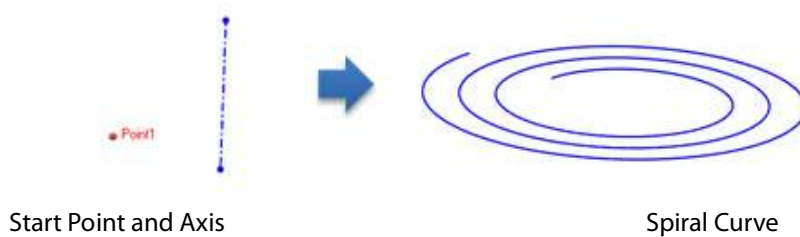
See Also

[Insert > Modeling Feature > Spiral](#)

Insert > Modeling Feature > Spiral

What

A **Spiral** is a two-dimensional curve which emanates from a central point, getting continuously farther away in a circular pattern. The **Spiral** command generates a spiral curve.



Why

The **Spiral** command is useful for:

- Sweeping or lofting a path

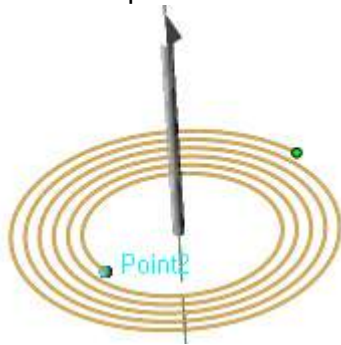
How to Use

How to generate a spiral curve

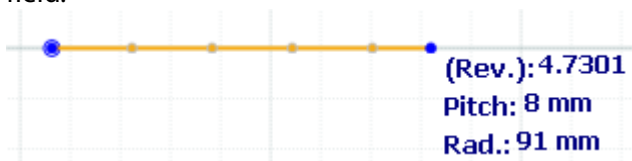
1. Prepare a point as a starting position and a vector as an axis.



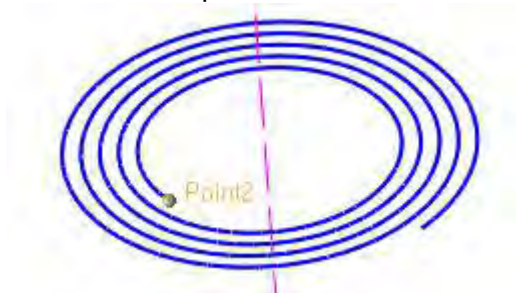
2. Go to **Insert > Modeling Feature > Spiral** or click on the icon in the toolbar.
3. Select a vector for **Axis**.
4. Select a ref. point for **Start**.



5. Click the end point on the graph in the **Profile View** and set the **Height, Radius, Pitch** and **Revolution**. The blue point can be dragged or specified by a numerical value in each field.



6. Click **OK** to complete the command.



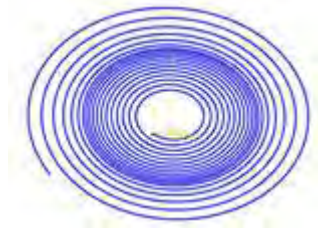
Options in Detail

Methods

Constant – Creates a constant pitch throughout a spiral.



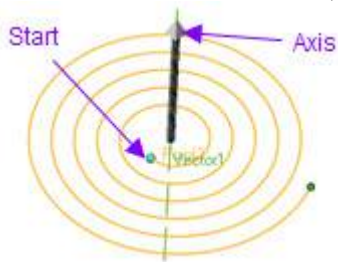
Variable – Creates a variable pitch based on specified parameters.




Select Entities

Axis – Selects a linear entity for the axis of a spiral.

Start – Selects a point entity for the starting position of a spiral.



Plot Mesh Section As Guide – Uses a section profile to extract the design intent of helix parameters. Clicking the  button will display the section profile of a mesh in the **Profile View**. This option is available if a mesh exists already.



(a) Mesh



(B) Extracted Mesh Section

Options

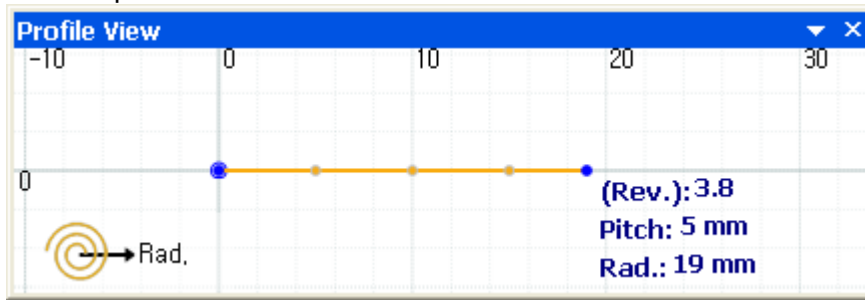
Counter Clockwise – Sets a spiral direction as counterclockwise.

Clockwise – Sets a spiral direction as clockwise.

Extend To Opposite Direction – Extends a spiral curve to the opposite direction by a specified number of revolutions.

Controlling Profile View

Used to define spiral parameters in the **Profile View**. Clicking a node point will display editable parameters.



- The horizontal axis represents the **Radius**.
- A node point can be dragged or directly specified on the widget by clicking it.
- The **Variable** option allows for the addition of more node points by clicking on the graph.
- To remove an added node, move the node onto another point and overlap it or by pressing the **Delete** key.
- Move the starting position by dragging the node.

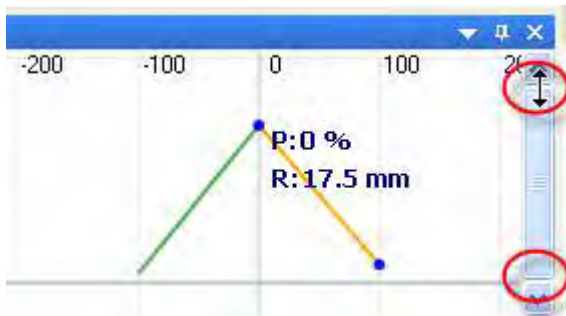


Viewing in Profile View


Zoom – Use the mouse wheel.

Pan – Use the left mouse button or the scroll bar in the Profile View window.

Scale (Y Direction) – Use Shift + left mouse button or move the top or bottom edge of the scroll bar.



Note:

The  button at the right top corner of the **Profile View**, displays additional functions.

Zoom Fit
Reset

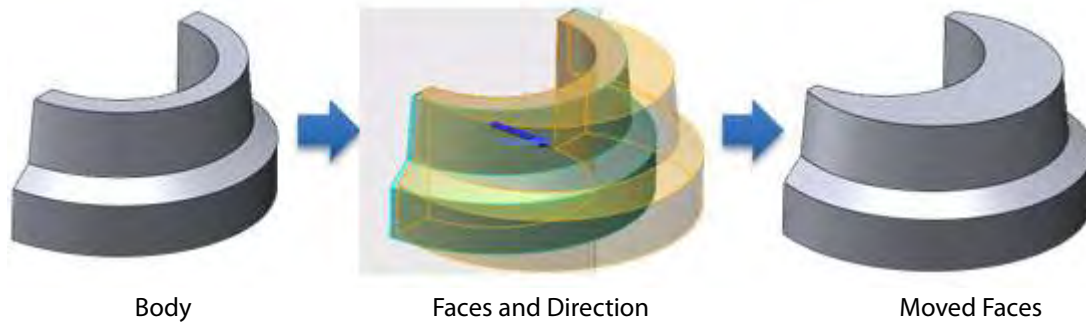
See Also

[Tools > 3D Sketch Entities > Spline](#)

Insert > Modeling Feature > Move Face

What

The **Move Face** command transforms a face and reconstructs the accompanying solid or surface body.



Why

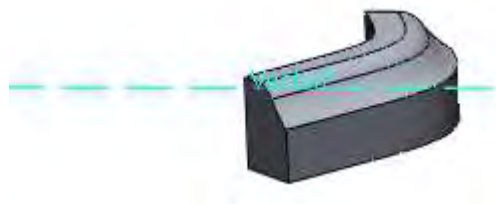
The **Move Face** command is useful for:

- Moving and deforming a body

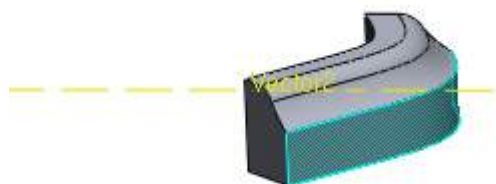
How to Use

How to transform a body face

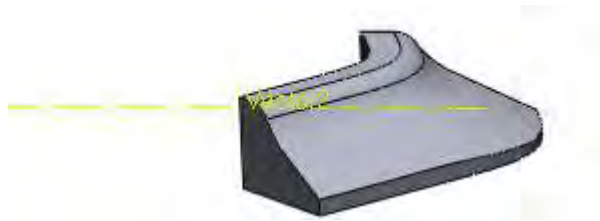
1. Prepare a body and vector as a direction.



2. Go to **Insert > Modeling Feature > Move Face** or click the icon in the toolbar.
3. Choose the **Translate** option and select a body face for **Faces** and a vector for **Direction**. Set the distance to **10 mm**.



4. Click **OK** to complete the command.



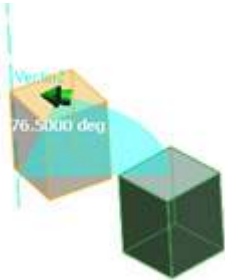
Options in Detail

Methods

Translate – Moves selected faces linearly.



Rotate – Moves selected faces by rotating them.



Select Entities

Faces – Selects faces on a solid or surface body.

Direction – Selects a linear entity or an entity with a normal to define a moving direction. Specify the **Distance** to define the amount of movement.

Axis – Selects a linear entity or an entity with a normal to define a rotation axis. Specify the **Angle** to define the amount of revolution.

See Also

[Insert > Modeling Feature > Delete Face](#)

[Insert > Modeling Feature > Replace Face](#)

[Insert > Modeling Feature > Split Face](#)

Insert > Modeling Feature > Delete Face

What

The **Delete Face** command removes faces on a solid or surface body.



Why

The **Delete Face** command is useful for:

- Reconstructing features

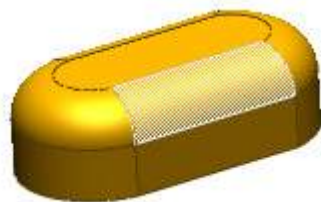
How to Use

How to remove a body face

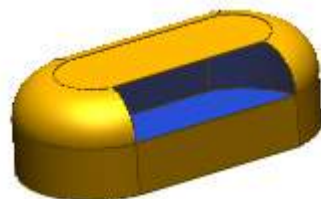
1. Prepare a solid or surface body.



2. Go to **Insert > Modeling Feature > Delete Face** or click the icon in the toolbar.
3. Choose the **Delete** option and select a face to delete.



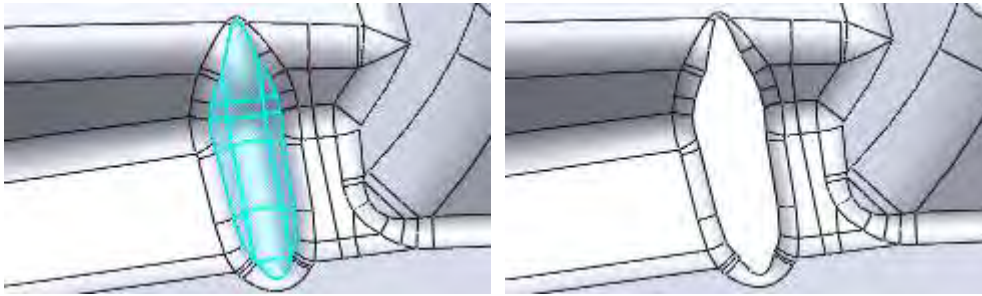
4. Click **OK** to complete the command.



Options in Detail

Methods

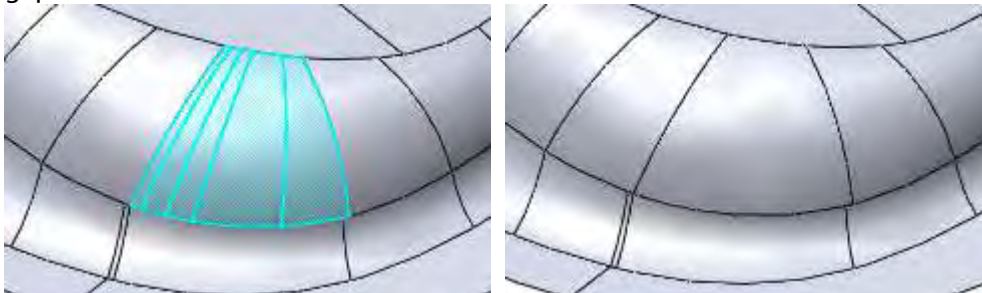
Delete – Removes selected faces.



Delete And Heal – Removes selected faces and automatically patches and trims the body.



Delete And Fill – Removes selected faces and replaces them with a single face to close any gaps.



Select Entities

Faces – Selects faces on a solid or surface body to remove.

See Also

[Insert > Modeling Feature > Move Face](#)

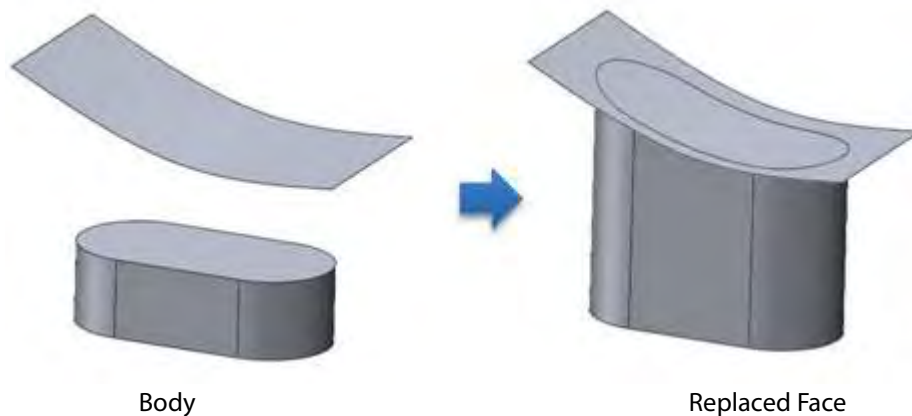
[Insert > Modeling Feature > Replace Face](#)

[Insert > Modeling Feature > Split Face](#)

Insert > Modeling Feature > Replace Face

What

The **Replace Face** command removes a selected face, extends neighboring faces, and replaces the original face with a target face.



Why

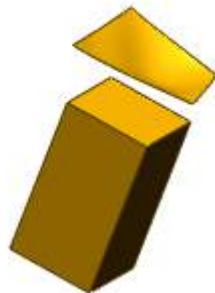
The **Replace Face** command is useful for:

- Reconstructing a body face
- Extending a feature to a surface
- Copying another face and applying it to a feature

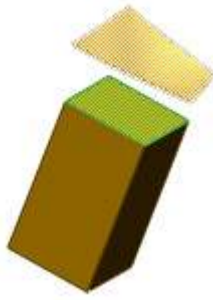
How to Use

How to replace a body face

1. Prepare a solid body and a surface body.



2. Go to **Insert > Modeling Feature > Replace Face** or click the icon in the toolbar.
3. Select a top face for **Target Faces**. Select a freeform surface for **Tool Entity**.



4. Click **OK** to complete the command.



Options in Detail

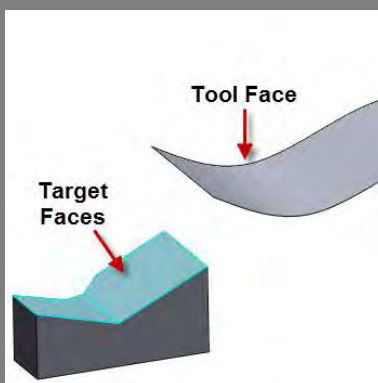
Select Entities

Target Faces – Selects faces to replace.

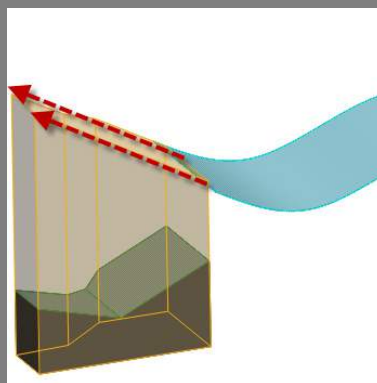
Tool Entity – Selects a region or surface body to replace a target face.

Note:

When a selected target face does not cover a tool face, the application will automatically extend the tool face tangent to a position that can touch the extruded target faces and replace the target faces.



Selecting Target Faces and Tool Face



Replacing Target Faces

See Also

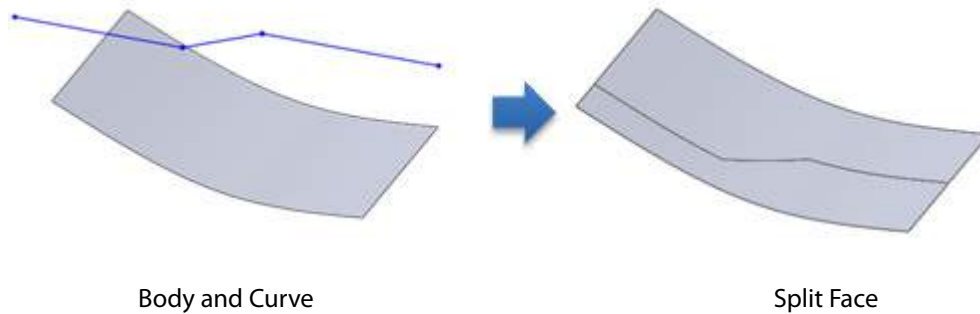
[Insert > Modeling Feature > Move Face](#)

[Insert > Modeling Feature > Delete Face](#)

Insert > Modeling Feature > Split Face

What

The **Split Face** command splits a target face by using the projection, silhouette and intersection methods. After splitting a face, a target entity will have several faces but will still be a single entity.



Why

The **Split Face** command is useful for:

- Adding edge faces
- Adding parting lines

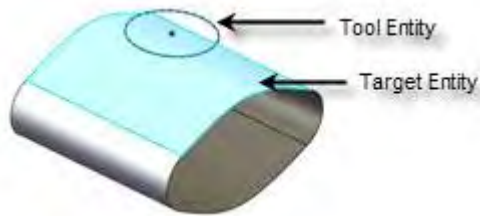
How to Use

How to split a body face by using a curve

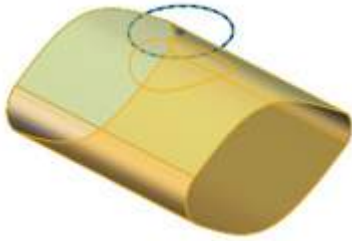
1. Prepare a curve and a body.



2. Go to **Insert > Modeling Feature > Split Face** or click the icon in the toolbar.
3. Choose the **Projection** method. Select a curve for **Tool Entities** and a face for **Target Entities**.



4. Click **Preview**. If the face cannot be split then change the projection direction by using the **Flip Direction** button.



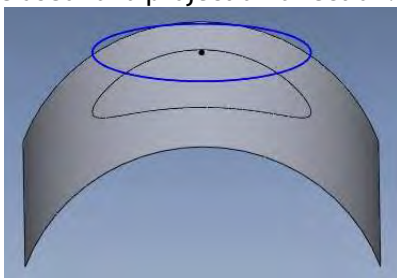
5. Click **OK** to complete the command.



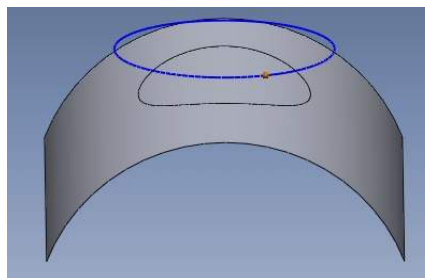
Options in Detail

Select Entities


Projection – Determines the projection method. When tool entities are 2D curves, a sketch normal will be used for a projection direction. If tool entities are 3D curves, a face normal will be used for a projection direction.



Split with 2D Sketch



Split with 3D sketch

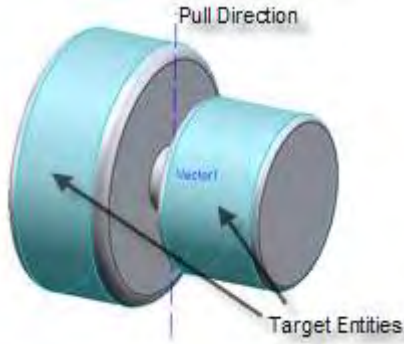
Single Direction – Projects a split line in one direction. To reverse the direction, click the **Flip Direction** button .

Bidirection – Projects a split line in both directions.

Silhouette – Creates a silhouette in a pull direction and splits target entities with the silhouette lines.

Pull Direction - Defines a silhouette direction.

Target Entities – Selects faces on a solid or surface body.

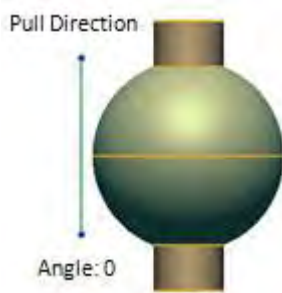


Pull Direction & Target Entities

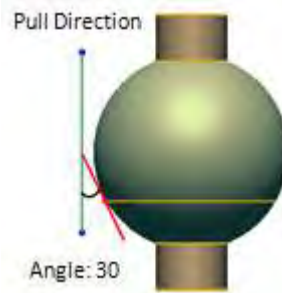


Split Result

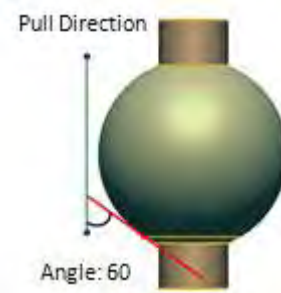
Draft Angle – Sets the angle between pull direction and tangential direction on a point of a face.



Draft Angle: 0



Draft Angle: 30



Draft Angle: 60

Intersection – Divides target entities by intersecting them with other faces.



Pull Direction & Target Entities



The Split Result

See Also

[Insert > Modeling Feature > Move Face](#)

[Insert > Modeling Feature > Delete Face](#)

[Insert > Modeling Feature > Replace Face](#)

Insert > Modeling Feature > Transform Body

What

The **Transform Body** command moves, rotates or scales a solid or surface body. It can also be used to align a body to another body or mesh by using datum pairs.



Why

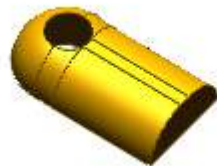
The **Transform Body** command is useful for:

- Copying and moving a body
- Repositioning a body
- Aligning a body to another body or a mesh

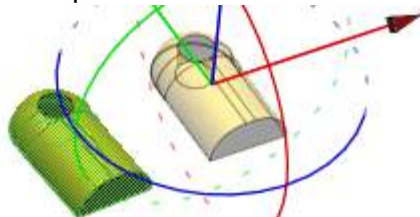
How to Use

How to move and rotate a body

1. Prepare a solid or surface body.



2. Go to **Insert > Modeling Feature > Transform Body** or click the icon in the toolbar.
3. Select a body and check the **Duplicate** box to copy it. Click the **Next Stage** button.
4. Select the **Rotate And Translate** method and input a **Rotate** and **Translate** value or drag the manipulator on the screen.



5. Click **OK** to complete the command.



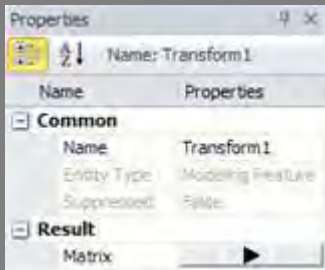
Note:

After the transformation of a body, the transformation matrix can be exported. To export the transformation matrix follow the steps below:

1. Select a Transform feature in the **Feature** tree.



2. See the **Properties** panel.



3. Click the  button.

4. Use the **Export** button and save the matrix. The file will be saved as Transform Matrix File (.trm).

1	0	0	20
0	0.9848	-0.1736	10
0	0.1736	0.9848	10
0	0	0	1

Export

Options in Detail

Select Entities

Bodies – Selects solid or surface bodies.

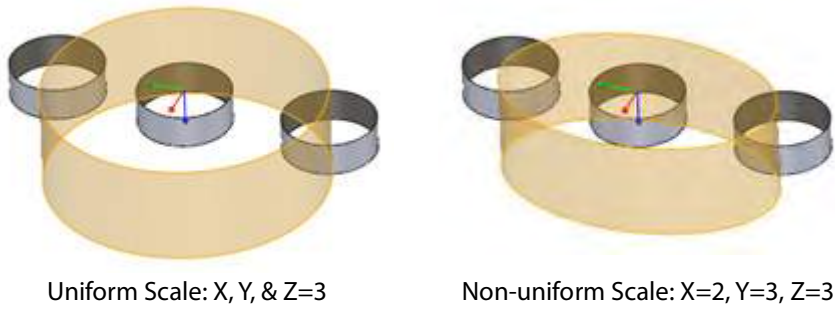
Duplicate – Copies selected bodies.

Methods

Rotate And Translate – Applies rotation and translation to a body in the X, Y and Z direction and axis. You can interactively use the manipulator on the screen or input specific values.

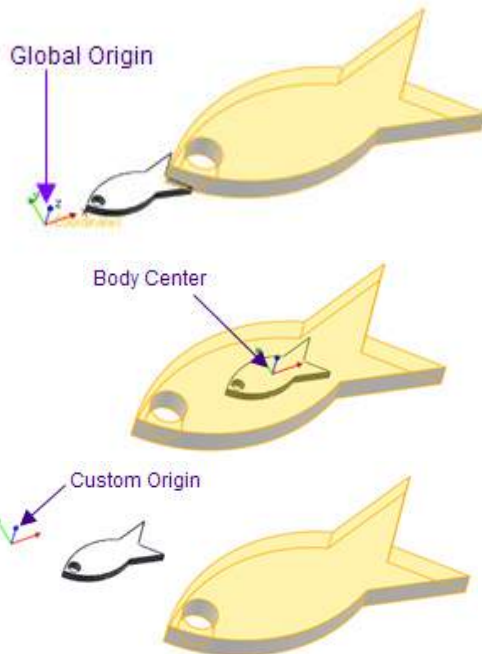
Scale - Scales a solid or surface body along its center, a coordinate system, or a custom position.

Uniform Scale – Scales a body based on the **X, Y** and **Z** directions concurrently. If this option is not used, scale can be applied individually in the **X, Y** and **Z** directions.



Unit Converter – Converts scale values by using unit conversion. Changing unit types in **From** and **To**, will automatically apply the changes to the scale value. Changing the scale value directly will set the **Unit Converter** as default.


Scale Center – Selects a position of a scale origin. **Global Origin**, or the global coordinate, **Body Center**, or the center of a body, and **Custom**, or the custom origin, can be chosen.



By Matrix – Allows for the direct input of values into a transformation matrix. After using the **Rotation And Translate** and **Scale** options the transformation matrix will be calculated and filled. If a value in the matrix is modified, the corresponding **Rotate, Translate, Scale** values will reflect those changes.

2	0	0	300
0	1	0	0
0	0	1	0
0	0	0	1



A transformation matrix file can be imported by using the **Matrix File Open** button .

If **Set** is selected the imported matrix is applied as it is.

If **Multiply** is selected the imported matrix will be multiplied from the current position.

If **Inverse Multiply** is selected the imported matrix will be inverted and multiplied from the current position.

If **Transpose** is selected the imported matrix will be transposed from the current position.

Datum Alignment – Aligns a body to another body or a mesh by using datum pairs. Select a face that can be a datum such as a cylinder, sphere, plane, or edge on the original body and then select a pair of datum on another body.



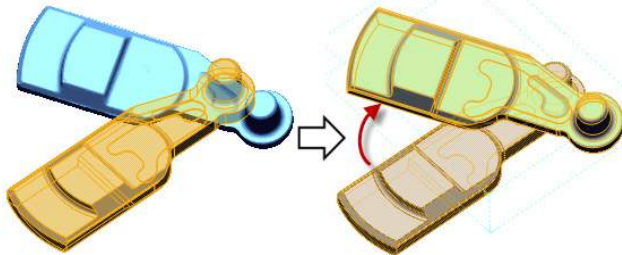
Datum Pairs – Type(Moving – Target, Flip) – Defines datum pairs in Moving and Target order.

Note:

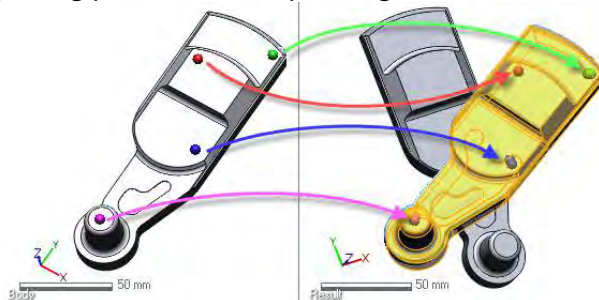
A moving datum can be flipped by using the Flip button  when datum pairs are defined.

Align To Scan – Aligns a body to a desired location in a scan data.

By Auto Guess – Aligns a body to a desired location in a scan data by automatically recognizing the geometry features of target scan entities.



By Picked Pair Points – Aligns a body to a desired location in scan data by manually picking points on corresponding features.



Sampling Ratio – Samples data points in a target scan by a specified value. When the ratio is 100%, all selected data will be used. When the ratio is 50%, half of the selected data will be used.

Global Coordinate – Sets constraints by using the global coordinate system

Lock Translation – Locks the translation of a transform in each coordinate axis.

Lock Rotation – Locks the rotation of a transform in each coordinate axis.

User Defined Coordinate – Sets constraints by using a user-defined coordinate system.

Lock Translation – Refer to the **Global Coordinate** option.

Lock Rotation – Refer to the **Global Coordinate** option.

See Also

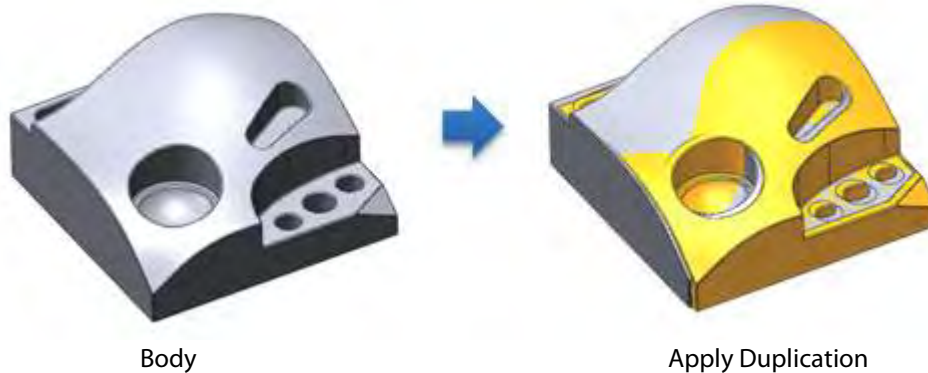
[Insert > Modeling Feature > Duplicate Body](#)

[Insert > Modeling Feature > Remove Body](#)

Insert > Modeling Feature > Duplicate Body

What

The **Duplicate** command copies a solid or a surface body.



Why

The **Transform Body** command is useful for:

- Copying a body

How to Use

How to copy a body

1. Prepare a body.

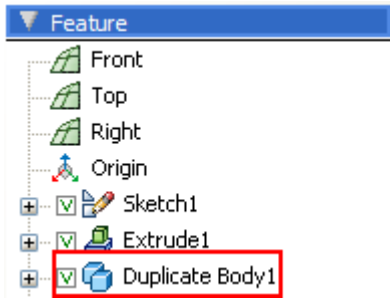


2. Go to **Insert > Modeling Feature > Duplicate Body** or click the icon in the toolbar.

3. Select a body to duplicate.



4. Click **OK** to complete the command. A new entity named **Duplicated Body 1** will be created in the **Feature Tree**.



Options in Detail

Bodies – Selects bodies to copy

See Also

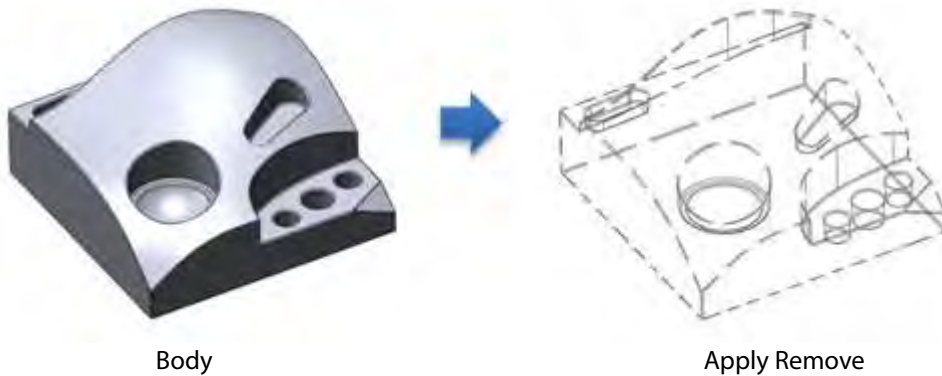
[Insert > Modeling Feature > Transform Body](#)

[Insert > Modeling Feature > Remove Body](#)

Insert > Modeling Feature > Remove Body

What

The **Remove Body** command deletes a solid or a surface body.



Body

Apply Remove

Why

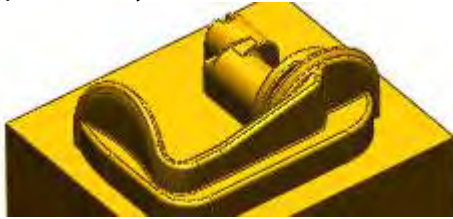
The **Remove Body** command is useful for:

- Deleting a body

How to Use

How to delete a body

1. Prepare a body.

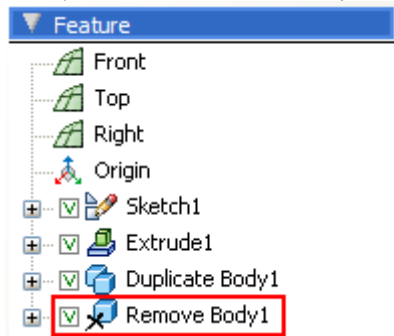


2. Go to **Insert > Modeling Feature > Remove Body** or click the icon in the toolbar.

3. Select a body to remove.



4. Click **OK** to complete the command. The selected body will be removed and the new entity named **Remove Body 1** will be created in the **Feature Tree**.



Options in Detail

Bodies – Selects bodies to remove.

See Also

[Insert > Modeling Feature > Transform Body](#)

[Insert > Modeling Feature > Duplicate Body](#)

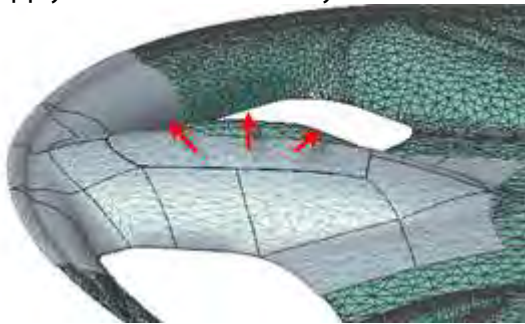
7.4.10. Insert > CAD Correct

The **CAD Correct** menu features various commands used to enable the editing and reforming of the geometries on a generated body or imported body based on deformed scan data. During the manufacturing process, a manufactured object can be different from its original CAD data because of design optimization after CAD modeling.

Insert > CAD Correct > Refit

What

The **Refit** command applies a fitting algorithm to body faces to match them with a deformed mesh. To apply this command a body and mesh must be aligned.



Refit Face

Why

The **Remove Body** command is useful for:

- Correcting original CAD data with a final product

How to Use

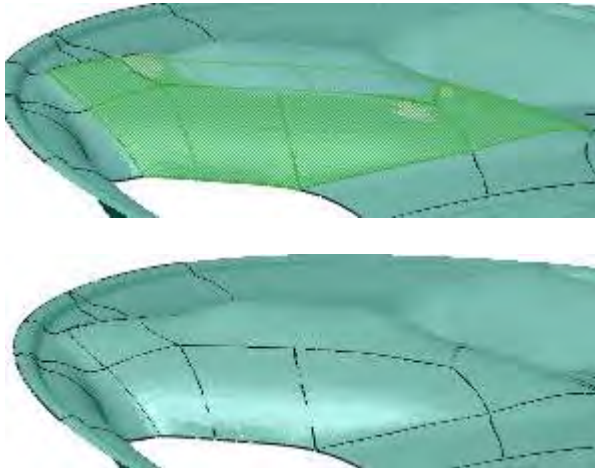
How to refit a body to a design-changed mesh

1. Prepare a mesh and a body. They should be aligned by using the **Tools > Align > Quick / Best Fit / Datum** commands.



2. Go to **Insert > CAD Correct > Refit** or click the icon in the toolbar.

3. Select a mesh for a reference **Mesh** and select faces on a body for **Faces** which will be target faces.



4. Click **OK** to complete the command.



Before Refit

After Refit

Options in Detail

Select Entities

Mesh – Selects a reference mesh.

Faces – Selects faces on a solid or a surface body.

Group Faces By Stiffness – Individually sets the weight factor for fitting to each face. The **Loosely Fit Faces**, **Tightly Fit Faces** and **Free Faces** buttons can be used to define the tightness for faces.

Loosely Fit Faces – Selects faces that will be loosely fit to a mesh. Faces around target faces for refitting can be chosen and looseness can be controlled by using the slider.

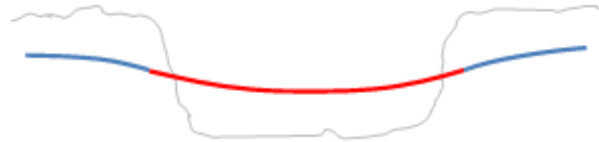
Tightly Fit Faces - Select faces that will be tightly fit to a mesh. Faces for refitting can be chosen and the tightness can be controlled by using the slider.

The sliders in the **Loosely Fit Faces** and **Tightly Fit Faces** options interact together. If one of the slider is moved, the other slider will change automatically.

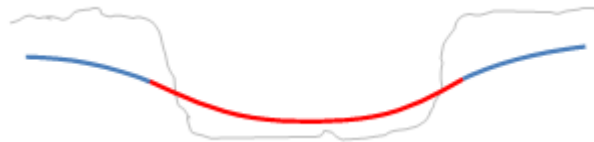
The following images illustrate influence of combinations of weight values. Moving the slider that is next to **Tightly Fit Faces** to **Tight**, and the slider that is next to **Loosely Fit Faces** to **Loose** will increase the deviation of loose faces correlatively when deviation decreases between tight faces and a reference mesh.



(A) Before Refitting



Red - Tightly Fit Faces, Blue - Loosely Fit Faces
(B) Minimum Weight of Tight Faces

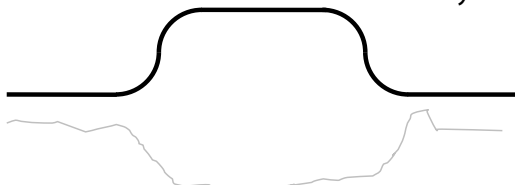


(C) Middle Weight of Tight Faces



(D) Maximum Weight of Tight Faces

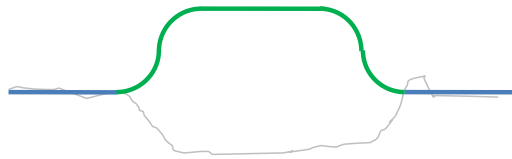
Free Faces – Selects faces that are not constrained by a reference mesh. The resulting shape of the free faces will be influenced by deformation from loose and tight faces only.



(A) Before refit



(B) Result from tight face (blue)



(C) Result from free face (green)

Fitting Options

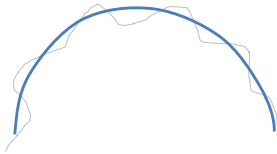
Type Of Control Vertices – Selects the type of control vertices.

The **Global** option will create an overall shape from a resulting body that will be better in smoothness.

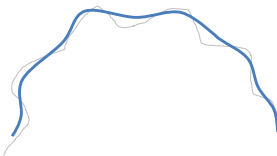
The **Local** option will create a resulting shape that will be more influenced by the local features on a reference mesh.



(A) Before refit



(B) After refitting with the Global option



(C) After refitting with the Local option

Geometry Capture Accuracy – Defines geometry capture accuracy. When the slider is moved towards **Loose**, less poly-faces will be used to create a shape. When the slider is moved towards **Tight**, more poly-faces will be used to create a shape.

Smoothness – Determines the smoothness level of a fitting surface.



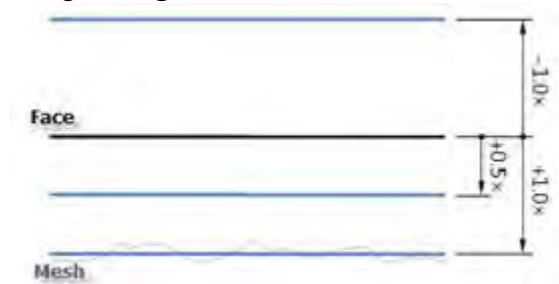
Smoothness: Less



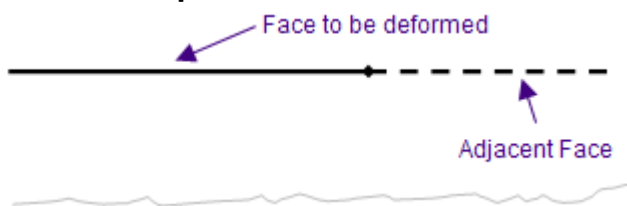
Smoothness: More

More Options

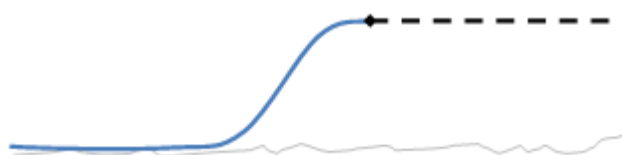
Compensation Factor – Controls the amount of deformation scaling by a specified value. A fitting face will offset from a mesh as a scale value. The default value is 1.0 and if the value is negative, the deforming direction will be flipped. Positive or negative values can be toggled by using the **Sign** button.



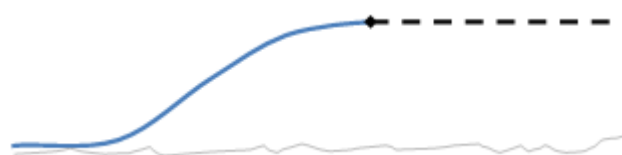
Shape Transition To Adjacent Faces – Centralizes a modification around edges by moving the slider towards **Abrupt**.



(A) Before Refitting



(B) Result of Abrupt



(C) Result of Gradual

Preserve Boundary Edge – Prevents boundary edge faces from being deformed.

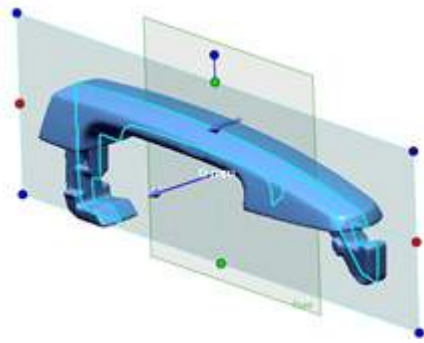
See Also

[Tools > Mesh Tools > Refit](#)

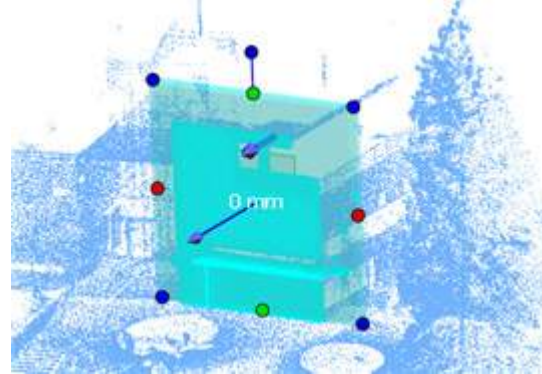
7.4.11. Insert > Mesh Sketch



The **Mesh Sketch** mode extracts sectional polylines or silhouette polylines based on a mesh or a point cloud as well as the creation of 2D geometries such as lines, circles, arcs, rectangles based on extracted polylines. Constraints on sketch entities can be applied to generate a fully parametric model. The **Mesh Sketch** mode extracts correct and accurate design intent from scan data.



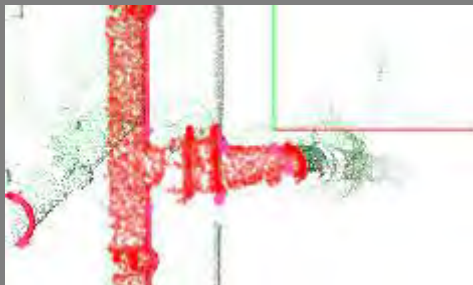
Mesh Sketch to Mesh



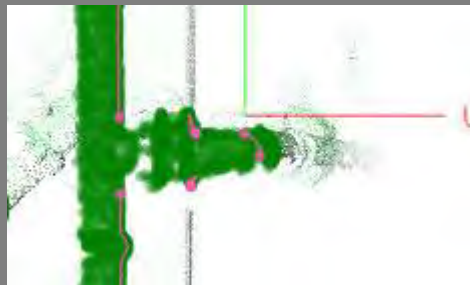
Mesh Sketch to Point Cloud

Note:

The color and point size of section points that are extracted from point clouds in the **Mesh Sketch** mode can be set by going to **File > Preferences**. Go to **Section Points** under **Colors** under **Sketch** in the **Display** tab or **Section Point Size** under **Size (pixel)** under **Sketch** in the **Display** tab.



Red, Size 3

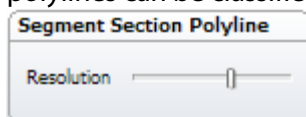


Green, Size 6

Entering the **Mesh Sketch** mode will display tool palettes that apply functions globally. The values in the tool palette will be preserved if the mode is exited and entered again.

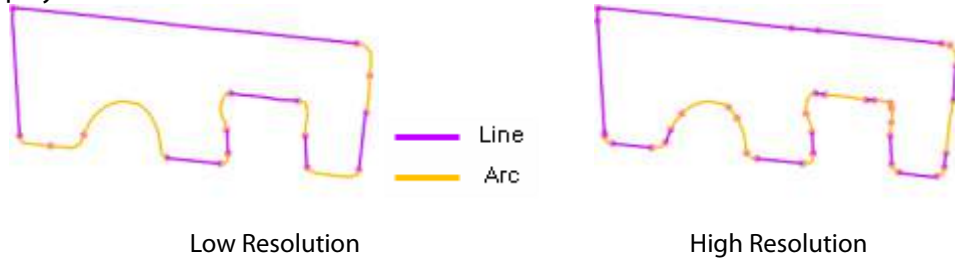
Segment Section Polyline

Section polylines are a group of small lines extracted from a section of scan data. The extracted polylines can be classified as lines or arcs.



Resolution – Determines the resolution of polylines. Moving the slider to the left will create fewer segments, and lines and arcs will have a loose tolerance. Moving the slider to the right will create more segments, and lines and arcs will have a tight tolerance. Moving the slider will

display classified lines and arcs in real-time.



Entity Fitting

During the creation of 2D geometries by fitting polylines, polylines can be filtered. Set a value and click the **OK** button, and the application will apply the condition to all of the entities in the Mesh Sketch mode.



Sampling – Executes a fitting by using sampled points according to a specified value. When section polylines are messy and large they can be sampled to accelerate calculation speeds.

Remove – Remove noise on section polylines.

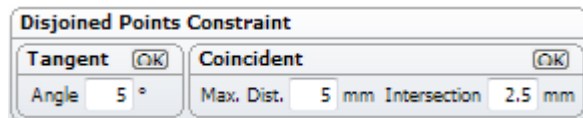
The **N*Sigma** option requires an **N** value and will execute a fitting by excluding fitting data that has an error value larger than **N** times of the standard deviation.

The **Ratio** option requires a ratio value and will execute a fitting by sorting fitting data only with the data that falls into the ratio.

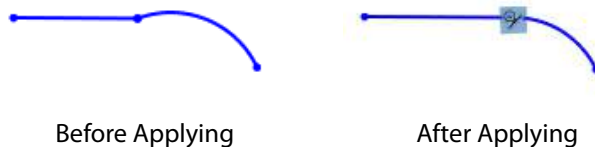
The **Absolute Distance** option requires an error distance value and executes a fitting by excluding fitting data outside of this distance.

Disjoined Points Constraint

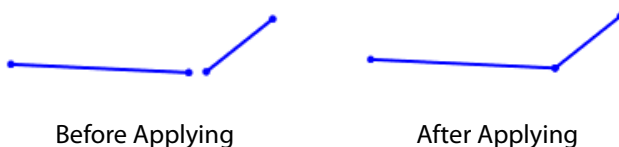
Automatically connects disjoined sketch entities such as lines and arcs globally, or only selected entities.



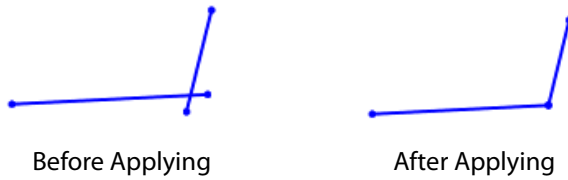
Angle – Connects curves and arcs with a tangent constraint that have a tangent angle between an arc and an arc, or an arc and a line that is less than a specified value.



Max. Dist. – Connects curves and arcs with a maximum distance less than a given distance value.



Intersection – Connects curves and removes the lesser distance after an intersection.



How to Use

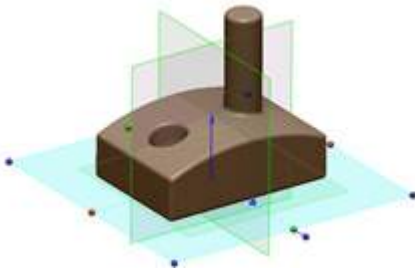
A. How to get sectional polylines by offsetting a plane

1. Prepare a mesh and a ref. plane.

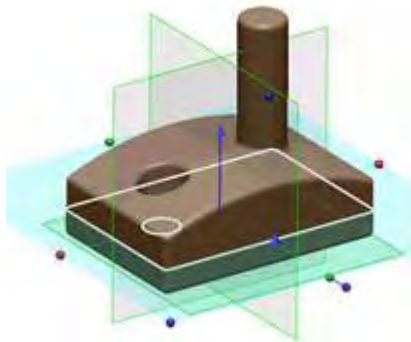


2. Click the **Mesh Sketch** mode icon in the toolbar, or go to **Insert > Mesh Sketch**.

3. Select a ref. plane for a base plane.



4. Drag the blue arrow to offset the plane and make an intersection with the mesh. The offset value can also be input next to the **Offset Distance From Base Plane** option.



5. Click **OK** to complete the command. Hide the mesh to clearly see the section polylines.

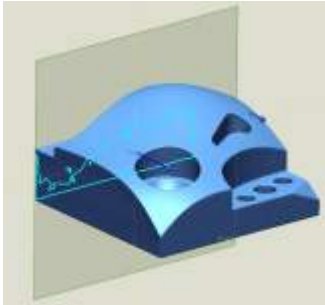


B. How to get silhouette polylines from a mesh

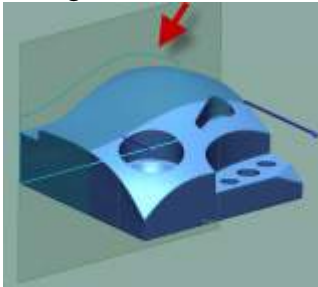
1. Prepare a mesh and a ref. plane.



2. Click the **Mesh Sketch** mode icon on the toolbar, or go to **Insert > Mesh Sketch**.
3. Select a ref. plane for a base plane.



4. Change the **Silhouette Range** value to cover the whole mesh.

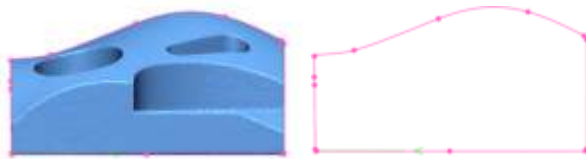


Note:

The silhouette range can be changed by dragging the blue arrow on the screen.



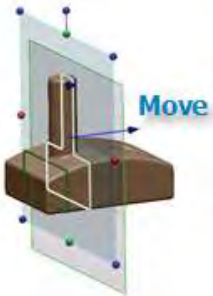
5. Click **OK** to complete the command.



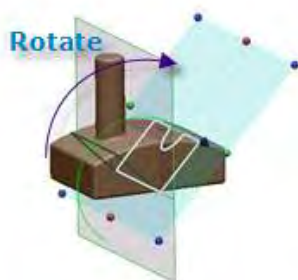
Options in Detail

Methods

Planar Method – Extracts section polylines at an intersection with a plane defined by the offset method. The intersecting position of a base plane can be selected by dragging the long blue arrow manipulator in the Model View or by directly inputting the offset value in the **Offset Distance From Base Plane** option. A planar entity can be selected for a **Base Plane**.



Rotational Method – Extracts section polylines at an intersection with a plane defined by the rotation method. The position of a base plane can be determined by rotating the plane along an axis. Select a planar entity for a **Base Plane** and a linear entity for a **Central Axis**.



Add Section Polyline

Section Polyline – Creates multiple section polylines by repositioning a base plane and clicking the **Add** button or removes a section polyline by selecting the section polyline in the list box and clicking the **Delete Section** button. To remove all section polylines, click the **Reset** button.

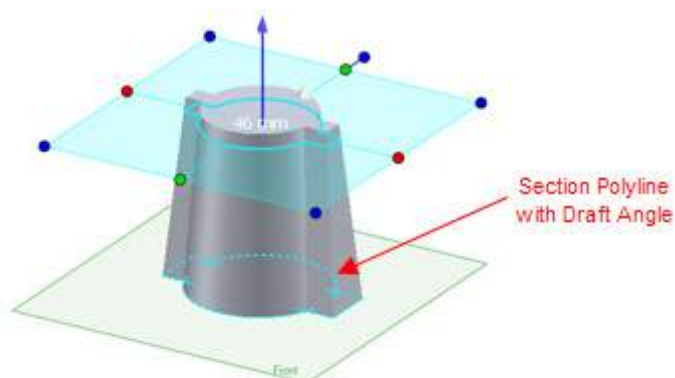
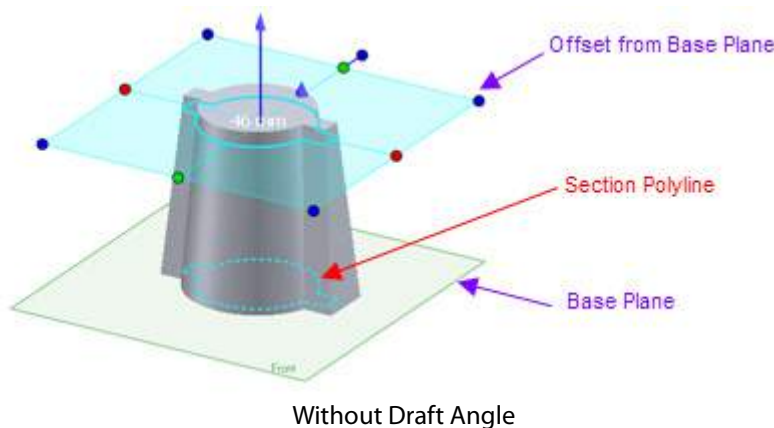
Offset Distance From Base Plane – Offsets from a base plane by inputting a distance from the base plane or by dragging the blue arrow in the Model View.

Silhouette Range – Extracts a silhouette from a mesh or point cloud as polylines by manually inputting a value or dragging the small blue arrow in the Model View. The value is the thickness used to find a silhouette range and can be resized and rotated by dragging the red, green and blue control points on the plane.



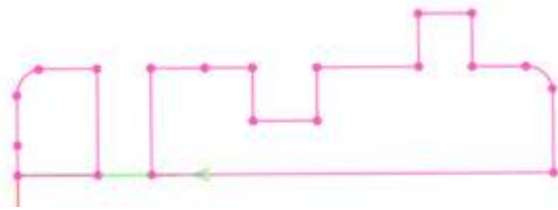
Options

Draft Angle - Sets a draft angle when extracting a section polyline from a drafted mesh feature. If a draft angle is set, an un-drafted section polyline from the offset distance from the base plane can be extracted.

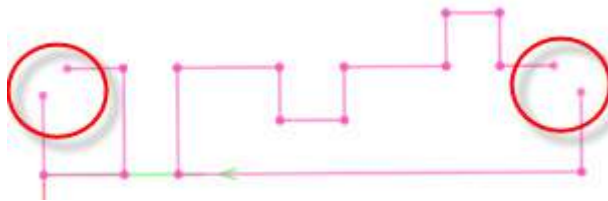


With Draft Angle

Exclude Round Corner - Remove round polylines less than a specified value. This option can be used to create a sketch without filleting.

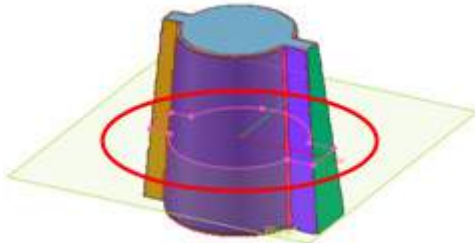


Disable Option

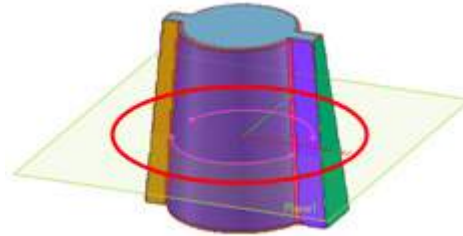


Enable Option

Limit To Regions - Selects regions to extract section polylines from. This option is only available if a mesh has regions.

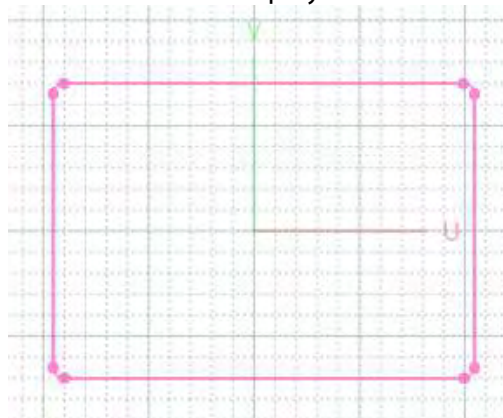
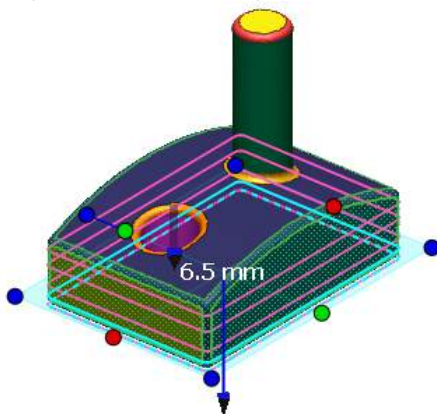


Without Region Limitation



With Region Limitation (front & back regions)

Avg. Section - Averages sections to make a master section polyline.



Note:

This option will be automatically enabled when entering Mesh Sketch Setup command through the Insert Sketch command. For more information, see [User Interface > Quick Modeling](#).

See Also

[Insert > Sketch](#)

7.4.12. Insert > Sketch

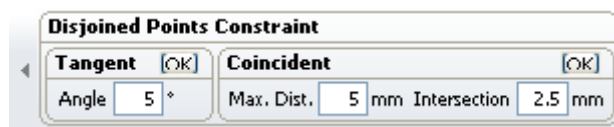


The **Sketch** mode creates 2D geometries such as lines, arcs, and splines, and can edit created 2D geometries.

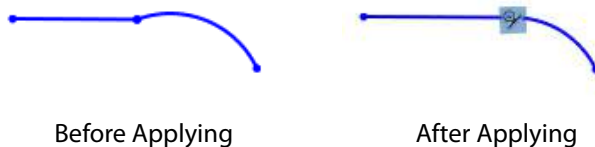
Entering the **Sketch** mode will display a tool palette that globally applies functions to all entities or selected entities. The values in the tool palette will be preserved if the mode is exited and entered again.

Disjoined Points Constraint

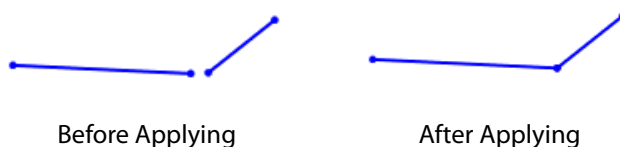
Automatically connects disjoined sketch entities such as lines and arcs globally or only on selected entities.



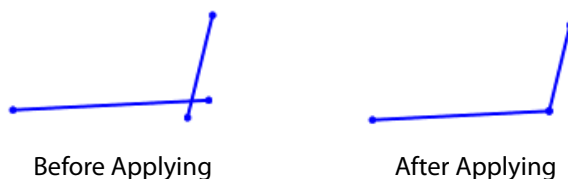
Angle – Connects curves and arcs with a tangent constraint that have a tangent angle between an arc and an arc, or an arc and a line that is less than a specified value.



Max. Dist. – Connects curves and arcs with a maximum distance less than a given distance value.

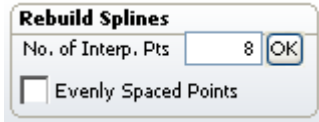


Intersection – Connects curves and removes the lesser distance after an intersection.



Rebuild Splines

Reorganizes a spline curve by changing the number of node points.



No. of Interp. Pts – Sets the total number of interpolation points by selecting splines, specifying a total number of nodes, and clicking the **OK** button.



Interpolation Points: 4



Interpolation Points: 8

Evenly Spaced Points – Spaces dense interpolation points evenly.



Before Evenly Spaced Points



After Evenly Spaced Points

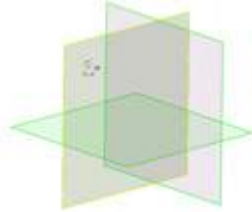
How to Use

How to create a sketch without a mesh

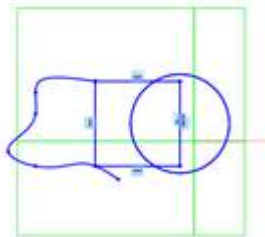
1. Click the **Sketch Mode** button on the toolbar, or go to **Insert > Sketch**.
2. A dialog tree will be displayed to define a sketch plane.



3. Select a planar entity such as a ref. plane or a face on a solid or surface body.



4. In the **Sketch** mode, create lines, arcs and splines.



5. Edit the sketch by using the **Trim** and **Fillet** commands.



6. Click the **Apply** button at right bottom corner or click the **Sketch** mode button again to apply all changes and exit the **Sketch** mode.

A solid or surface body can be generated from a sketch by using the **Extrude**, **Revolve**, **Loft**, and **Sweep** commands.

See Also

[Insert > Mesh Sketch](#)

7.4.13. Insert > 3D Mesh Sketch



The **3D Mesh Sketch** creates splines on a mesh using various commands. Created splines can be edited with tools such as the **Trim**, **Offset**, and **Project** commands.

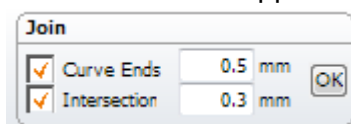
Entering the **3D Mesh Sketch** mode will display a tool palette that globally applies functions to entities in the mode. The values in the tool palette will be preserved if the mode is exited and entered again.

Note:

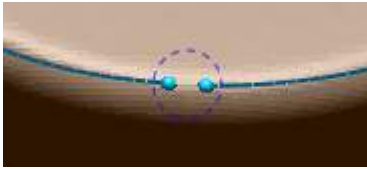
The difference between **3D Mesh Sketch** and **3D Sketch** modes is that curves generated in **3D Mesh Sketch** mode always lay on a mesh but curves in **3D Sketch** freely exist in space.

Join

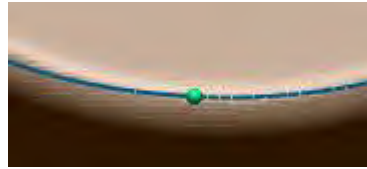
Automatically joins disconnected splines and create intersection points. Set a value and click the **OK** button and the application will apply the value to all splines or selected splines.



Curve Ends – Joins curve ends that have a distance smaller than a given value.

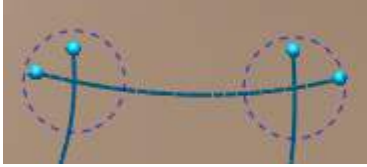


Before Applying

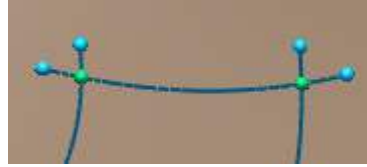


After Applying

Intersection – Creates intersection points between splines.



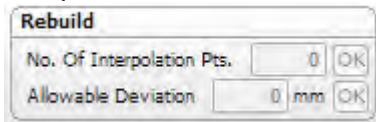
Before Applying



After Applying

Rebuild

Regenerates splines by controlling node points and deviations between splines and a mesh. This tool palette is available after selecting splines.



No. Of Interpolation Pts. – Sets the total number of interpolation points by selecting splines, specifying a total number of nodes, and clicking the **OK** button. To reduce the deviation between spline and mesh data, increase the number of node points.



Number: 4

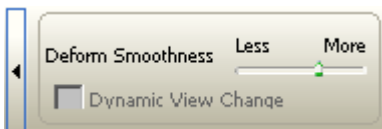


Number: 7

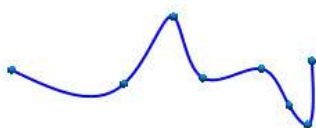
Allowable Deviation – Regenerates curves by a specified deviation. Node points will be regenerated and rearranged automatically to satisfy the deviation.

Deform Smoothness

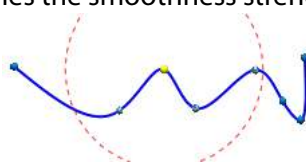
Defines the smoothness and viewing direction when creating a spline.



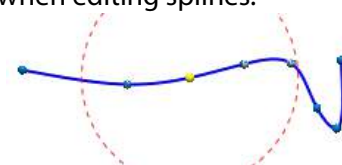
Deform Smoothness – Determines the smoothness strength when editing splines.



Original Curve



Smoothness: Min



Smoothness: Max

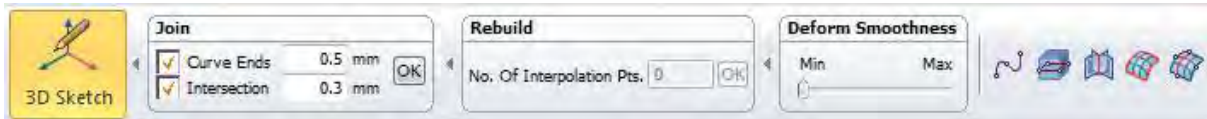
Dynamic View Change – Automatically centralizes the view whenever a point is picked on a

mesh. This option is available after generating a spline.

See Also

[Insert > 3D Sketch](#)

7.4.14. Insert > 3D Sketch



The **3D Sketch** mode creates **3D** geometries such as splines, sections and boundaries on a mesh. Created curves can be edited by using the **Trim**, **Offset**, and **Project** commands.

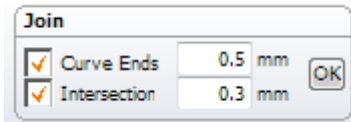
Entering the **3D Sketch** mode will display a tool palette that globally applies functions to all entities in the mode. The values in the tool palette will be preserved if the mode is exited and entered again.

Note:

The difference between **3D Mesh Sketch** and **3D Sketch** modes is that curves generated in **3D Mesh Sketch** mode always lay on a mesh but curves in **3D Sketch** freely exist in space.

Join

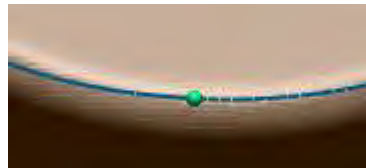
Automatically joins disconnected splines and create intersection points. Set a value and click the **OK** button and the application will apply the value to all splines or selected splines.



Curve Ends – Joins curve ends that have a distance smaller than a given value.

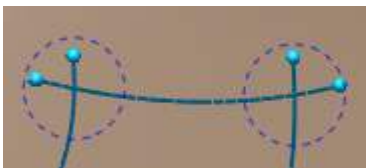


Before Applying

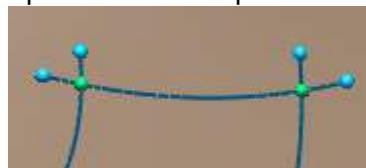


After Applying

Intersection – Creates intersection points between splines.



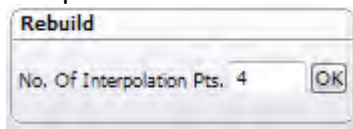
Before Applying



After Applying

Rebuild

Regenerates splines by controlling node points and deviations between splines and a mesh. This tool palette is available after selecting splines.



No. Of Interpolation Pts. – Sets the total number of interpolation points. Select splines and specify the total number of nodes and click the **OK** button.

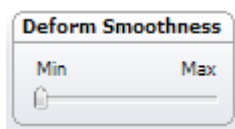


Number: 4



Number: 7

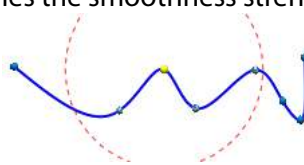
Deform Smoothness



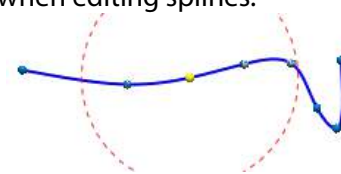
Deform Smoothness – Determines the smoothness strength when editing splines.



Original Curve



Smoothness: Min



Smoothness: Max

See Also

[Insert > 3D Mesh Sketch](#)

7.4.15. Insert > Scanner Direct Control

The **Scanner Direct Control** menu enables the use of various scanning devices directly within the application.

A preferred scanner device can be set in the **Initial Setup Dialog** that will be displayed when the application is initially started or in the **Device** list in the **Hardware** table at **File > Preferences**.

Insert > Scanner Direct Control > LiveScan(TM)

What can LiveScan™ do?

LiveScan™ is designed to support the collection of scan data or probing data using various devices to use with the Geomagic Design X application. The following are supported in LiveScan™.

Calibration (Optional as device)

- Hardware calibration can be performed.
- The bundle software from a device must be installed.

Laser and Probe Setting (Optional as device)

- Laser and probe parameters can be set.
- The bundle software from a device must be installed.

Move Device

- Maintains a coordinate system after moving a device.

Align

- Aligns scanning or probing coordinate to CAD's coordinate.

Scanning

- Acquires scanning data.

Probing

- Acquires probing data.

Generate Entity

- Instantly generates from acquired scanning or probing data.
- Ref. Geometry (Point, Vector, Plane, Polyline)
- Point Cloud
- Mesh
- Surface Body
- Solid Body
- 2D Sketch
- 3D Sketch

Supported Devices in LiveScan™

The following table shows supported devices in the 32-bit and 64-bit versions of the application.

Manufacturer	Measurement Device	32bit	64bit	Web Site
Creaform	HandyPROBE	✓	✓	www.creaform3d.com
FARO	FaroArm/ScanArm	✓	✓	www.faro.com
Kreon*	Zephyr/Aquilion/Solano	✓	✓	www.kreon3d.com
LDI	SLP Series	✓	✓	www.laserdesign.com
Leica	T-Scan TS50	✓	✓	metrology.leica-geosystems.com
NDI	ScanTRAK	✓	✓	www.ndigital.com
Nikon Metrology**	Handheld	✓	✓	www.nikonmetrology.com
Perceptron	ScanWorks	✓	✓	www.perceptron.com
Romer	Absolute	✓	✓	www.romer.com
	Infinite/Stinger	✓	✓	
	Sigma/Omega	✓		

* Kreon users should install "Rapidform plug-in installer" which is distributed by Kreon.

** The hardware that is supported by the Handheld API is the following:

- Optical CMM (K-Series system) with the laser scanners: MMD, MMC, MMDx
- MCA II 7-axis articulated arm with the scanners: MMD, MMC, MMDx
- MCA II 6-axis articulated arm without scanner
- MCA 7-axis articulated arm with the laser scanners: MMD, MMC, MMDx, LC60D
- MCA 6-axis articulated arm with the scanners: MMD, MMC
- Romer Cimcore Infinite 1.0 7-axis articulated arm with the laser scanners: MMD, MMC, MMDx
- Romer Cimcore Infinite 2.0 7-axis articulated arm with the laser scanners: MMD, MMC, MMDx
- Romer Cimcore Infinite 1.0 6-axis articulated arm with the laser scanners: MMD, MMC
- Romer Cimcore Infinite 2.0 6-axis articulated arm without laser scanner
- Faro Fusion, Titanium, Platinum 7-axis articulated arm with the laser scanners: MMD, MMC,

MMDx


- Faro Fusion, Titanium, Platinum 6-axis articulated arm with the laser scanners: MMD, MMC

How can LiveScan™ control a scanner?

Geomagic Design X's LiveScan™ technology contains three function controls. The functions of the buttons will vary between different type and manufacturers of scanners.

Button on LiveScan™	Function
A	Scan - Resumes scanning and pauses scanning. Probe – Picks a point.
B	Scan - Stops scanning. Probe - Creates a scan entity.
C	Scan - Not Available Probe – Cancels the scanning.

To perform the A, B and C functions of LiveScan™ as described above, use the buttons on each scanner device specified in the following table.

Manufacturer	Measurement Device	Button
Creaform	Handy PROBE	<p>There are Three buttons on the hardware.</p> <p>Probing</p> <p>Function A: Click the middle button.</p> <p>Function B: Click the buttons on the left side.</p> <p>Function C: Press the buttons on the right side.</p>
FARO	FaroArm ScanArm	<p>Scanning</p> <p>Function A: Click the green button.</p> <p>Function B: Click the red button.</p> <p>Probing</p> <p>Function A: Click the green button.</p> <p>Function B: Click the red button.</p> <p>Function C: Press the red button for a few seconds.</p> 
Kreon	Zephyr Aquilion Solano	The function control varies according to the arm type. For a Cimcore arm, the function control is same with that of a Perceptron arm.

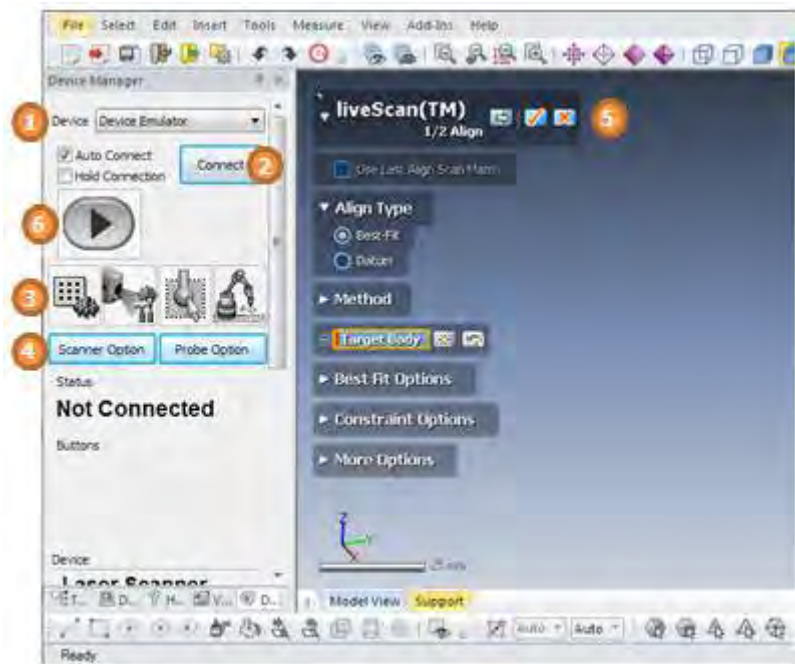
		<p>Scanning Function A: Click the middle button. Function B: Click the buttons on both sides.</p> <p>Probing Function A: Click the middle button. Function B: Click the buttons on sides. Function C: Press the buttons on the sides for a few seconds.</p>
LDI	SLP Series	<p>The function control varies according to the arm type. For a FARO arm, the function control is as follows:</p> <p>Scanning Function A: Click the green button. Function B: Click the red button.</p> <p>Probing Function A: Click the green button. Function B: Click the red button. Function C: Press the red button for a few seconds.</p>
Nikon Metrology	Handheld	<p>Scanning Function A: Click the middle button. Function B: Click the buttons on both sides.</p> <p>Probing Function A: Click one of the four buttons on the inside. Function B: Click the button on the outside.</p> 
Perceptron	ScanWorks	<p>For a Cimcore arm, the function control is as follows:</p> <p>Scanning Function A: Click the middle button Function B: Click the buttons on both sides.</p> <p>Probing Function A: Click the middle button. Function B: Click the buttons on both sides. Function C: Press the buttons on both sides for a few</p>

		seconds.
Romer	Absolute	There are three buttons on the hardware. Probing Function A: Click the middle button. Function B: Click the buttons on both sides. Function C: Press the buttons on both sides for a few seconds.
	Infinite/Stinger	
	Sigma/Omega	
Steinbichler	T-Scan 3/CS	Scanning Function A: Hold the button to resume and release the button to pause. Probing Function A: Click the button on the application screen. Function B: Click the button on the application screen.

LiveScan™ workflow

Before starting LiveScan™, a hardware device must be installed with its drivers or software and be ready to communicate with the PC.

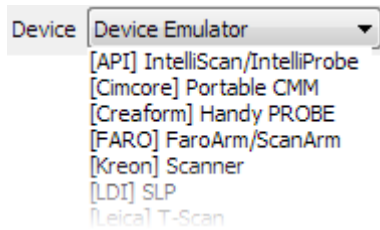
- 1 Select a device supported in the application.
- 2 Connects the device. (If connected, this button changes to "Disconnect")
- 3 (Optional procedure as a device) Hardware parameters settings for Calibration, Laser Setting, Probe Selection.
- 4 (Optional procedure) Software parameters settings for scanning and probing.
- 5 Choose "Align" or "Scan" stage.
- 6 Start scanning or probing. (If started, this button changes to "Stop")






How to Use

A. Aligning the scan coordinate to CAD data using Best-Fit method.

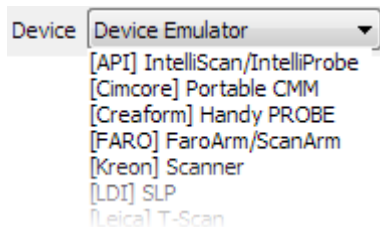
1. Import CAD data.
2. Go to **Insert > Scanner Direct Control > LiveScan(TM)** or click the icon in the toolbar.
3. Choose a device.




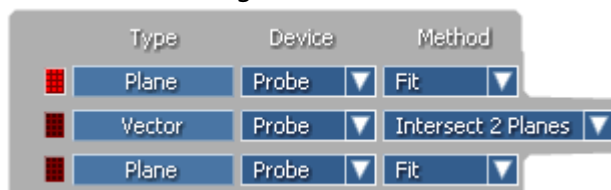
4. Click the **Connect** button .
5. Set the alignment method to **Best-Fit** in the LiveScan™ dialog tree.
6. Click the **Target Body** button and choose a CAD data.
7. Click the **Start** button  to start scanning.
8. Scan a real part by pushing the “A” button on the device. It must cover the entire area.
9. After scanning, push the “B” button. The Best-Fit alignment calculation will be performed.
10. To acquire additional scan data or probe data, move to the **Next Stage** .

B. Aligning the scan coordinate to CAD data using Datum method.


1. Import CAD data.
2. Go to **Insert > Scanner Direct Control > LiveScan(TM)** or click the icon in the toolbar.
3. Choose a device.



4. Click the **Connect** button .
5. Set the alignment method to **Datum** in the LiveScan™ dialog tree.
6. Select datums on the CAD data such as plane, cylinder, cone and sphere. Selected datum types will be listed. The device type (probe or scanner) and a method for each datum can be configured.

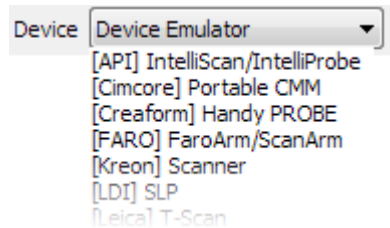





7. Click the **Start** button  to start scanning.

9. Probe a real feature by pushing the “A” button on the device.
10. After acquiring probing points for a datum, push the “B” button.
11. Complete the remaining datums.
12. After completing the last datum the Datum alignment calculation will be performed.
13. To acquire additional scan data or probe data, move to the **Next Stage** .

C. Scanning a real part.

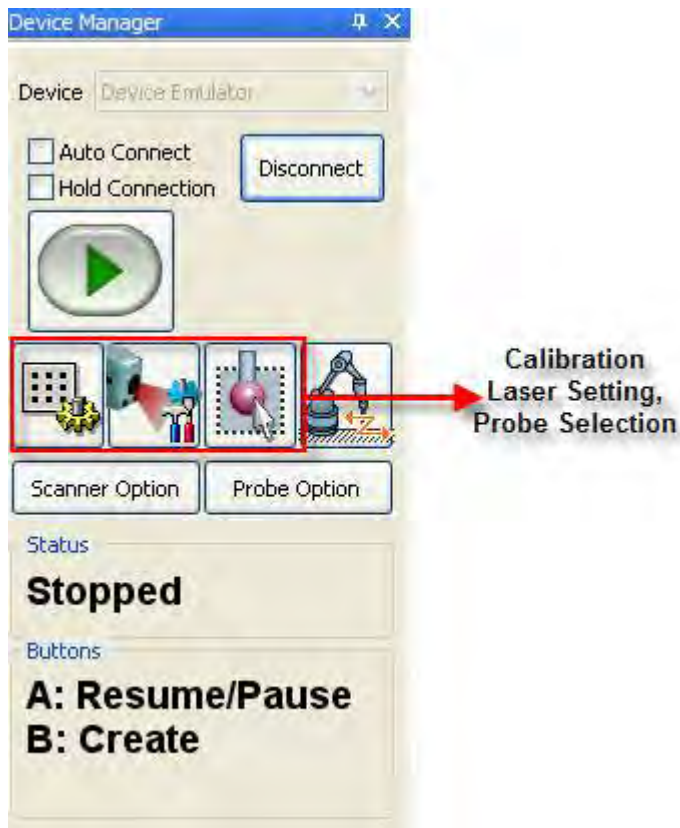
1. Import CAD data.
2. Go to **Insert > Scanner Direct Control > LiveScan(TM)** or click the icon in the toolbar.
3. Choose a device.



4. Click the **Connect** button .
5. Move to the **Next Stage**  in the LiveScan™ dialog tree.
6. Choose a device type from a Laser Scanner or Touch Probe.
7. Choose a **Feature Type** which will be generated in the application from acquired scan or probe data.
8. Click the **Start** button  to start scanning or probing.
9. Click **OK** to complete the LiveScan™ command.

Options in Detail of Device Manager

1. Device Manager



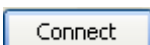
- Auto Connect
- Hold Connection

When the **Auto Connect** option is enabled, the last connected device will be remembered and will connect automatically when starting LiveScan™.

When the **Hold Connection** option is used, the application will not disconnect a device even if the LiveScan™ command is closed.



Used for a scanning device calibration.



Connects a selected device.



Used for a scanning device settings.



Used to reposition a device to scan or probe a part.



Used for the probe settings of a device.

Move Device

Used to when scanning a large object that requires multiple scans due to the single scan size limitations of a scanning device.



How to use the Move Device command

If Device Type is a Scanner:

1. Click **Next Step** (2nd process) and then scan the part.
2. Click **Next Step** (3rd process) and then move the device.
3. Click **Next Step** (4th process) and then re-scan the area from the 2nd step.
4. Click **Next Step** (5th process) and the application calculates the moved position of the device.

If Device Type is a Probe:

1. Click **Add** and create 3 or more Ref. Points (The creation method can be defined in Method)
2. Click **Next Step** (2nd process) and then prove the part regarding to the Ref. Points.
3. Click **Next Step** (3rd process) and then move the device.
4. Click **Next Step** (4th process) and then re-prove the part from 2nd step.
5. Click **Next Step** (5th process) and then the application calculates the moved position of the device.

Options in Detail of Scanner Options Dialog

The **General** tab lists general settings when scanning.



Display



Zoom Fit – Displays an entire model during scanning. The view will be refreshed when the scan line is out of the screen.



Dynamic Panning – Aligns a view by following a device's front direction.

Note:

The view will be fixed if both Zoom Fit and Dynamic Panning are not selected.

View Direction

Fixed Top View – Fixes the top view direction by using hardware information. The **Set** button defines the Top view direction.

Smart View – Intelligently recognizes a scanning direction within a system-defined view range and dynamically adapts the view direction to a proper viewpoint.

Free View – The view direction will be automatically set according to a scanning direction.

Auto Restart

Auto Restart – Performs post processing and readys to scan without pressing the **A** button after stopping a scan by pressing the **B** button.

Auto Save

Auto Save - Automatically saves raw scan data in a specified location. The folder is automatically named by date and saved in the .lsf format.

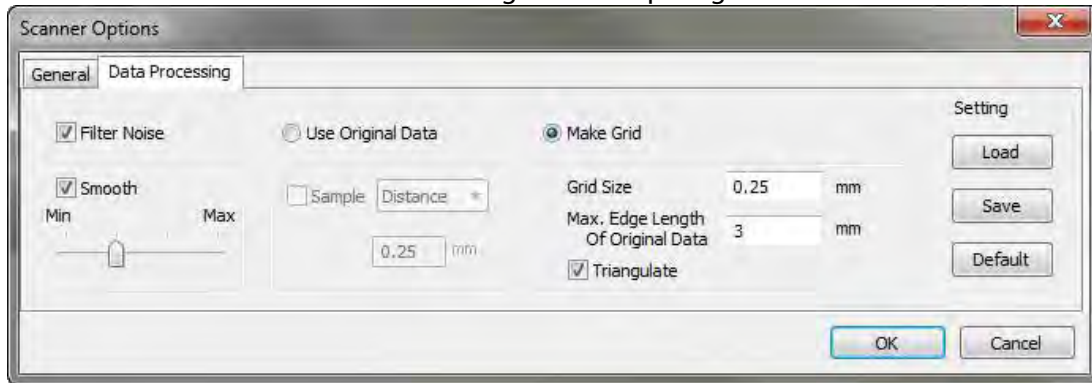
For example: xxx/20070704/ScanRawData1_1627_1.lsf

Sound and Button

Enable Sound While Scanning – Enables sound while scanning.

Hotkey for A/B Buttons – Defines keyboard hotkeys for the A and B button. Click a box and enter a desired key.

The **Data Process** tab lists detailed settings when acquiring scan data.



Filter Scan Data

Filter Noise – Reduces the noise introduced during scanning.

Smoothness of Scan Data

Smooth – After post processing from acquired scan data, smoothing is applied.

Scan Data Post Processing

Use Original Data – For post processing of scan data, uses original scan data from scan device.

Sample - Skips several points when scanning data large data that it is beyond memory limits.

Distance – Specifies a length that the software uses to find points.

Ratio - Specifies the sampling ratio to load points.

Make Grid – Generates a grid type uniform scan data.

Grid Size – Defines a grid resolution. High resolution of a grid will make a dense scan data.

Max. Edge Length Of Original Data – limits the maximum length of the polygonal edges of the original data.

Triangulate – Creates a mesh from the scan data.

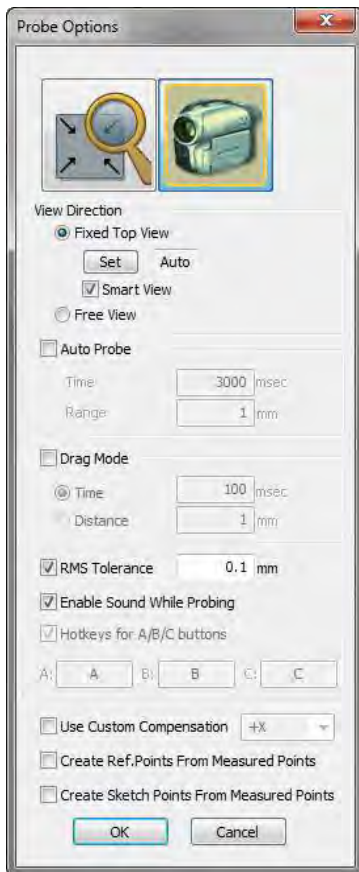
Setting

Load - Imports saved settings for the **Data Process** tab.

Save - Saves settings in the **Data Process** tab as an .ini file.

Default – Resets all settings as default values.

Options in Detail of Probe Options Dialog



View Direction

Fixed Top View – Fixes the top view direction by using hardware information. The **Set** button defines the Top view direction.

Smart View – Intelligently recognizes a probing direction within a system-defined view range and dynamically adapts the view direction to a proper viewpoint.

Free View – Freely sets the view direction according to a probing direction.

Auto Probe

Auto Probe – Acquires a probe position automatically without using the **A** button when the probe tip is held still near a target position. Pressing the **A** button changes the acquisition status to **Pause** and **Resume**. In the **Pause** state, no probe position is acquired, even if the probe is held at a position. In the **Resume** state, the auto probing scheme will be used. The current status at the top right in Model View.

Time - Defines the hold time when the probe tip stays near the target position.

Range - Defines the allowable area of acquiring position from the target position.

Drag Mode

Drag Mode – Allows to capture multiple points while pushing the **A** button.

Time – Every point as defined time will be acquired.

Distance – Every point as defined distance will be acquired.

Tolerance and Sound

RMS Tolerance – Sets an RMS tolerance value and will display any value outside of the tolerance during probing as red and will trigger a beeping alert from the PC in use.

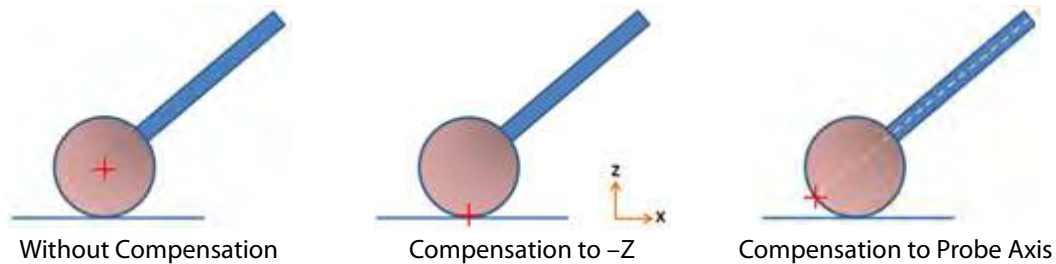
Enable Sound While Probing – Triggers a beeping alert from the PC in use when generating a probing point.

Hotkey

Hotkey for A/B/C Buttons – Defines keyboard hotkeys for the A, B and C button. Click the box and enter the desired key.

Compensation and Generate Entity

Use Custom Compensation - Compensates a probe radius by the direction of the X, Y, and Z global axes or the probe axis and is useful when a scanning part is aligned to a CAD in the application. Compensation is applied to a surface normal direction when the **Surface** option is chosen.

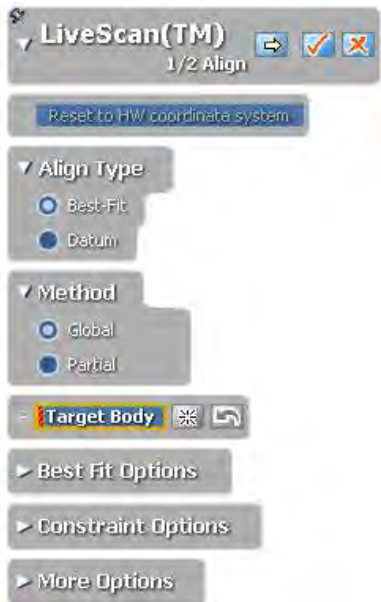


Create Ref. Points From Measured Points – Registers probed points as ref. points.

Create Sketch Points From Measured Points – Creates points in the **Sketch** mode from probed points.

Options in Detail of LiveScan(TM) Dialog Tree

1/2 Align Stage



Reset to HW Coordinate System – Resets the coordinate system to the HW coordinate system.

Note:

When running the LiveScan™ command, the scanning coordinate system aligns by default to the previously used coordinate system by using the last used transform matrix.

Align Type

Best Fit – Alignment method that is applied when scanning is finished.

Global – Alignment method that is applied to a selected whole body.

Partial – Alignment method that is applied only to selected faces on a body.

Max. Distance – Sets the maximum distance between alignment entities.

Sampling Ratio – Samples data points by a specified value. When the ratio is 100%, all selected data will be used. When the ratio is 50%, half of the selected data will be used.

Max. Iteration Count – Determines a value that will limit the number of times a registration iterates.

Max. Average Deviation – Determines a value as deviation criteria used to judge if a registration result is satisfactory. The application will attempt to make the average deviation of a final model less than this value within the Max. Iteration Count.

Offset – Applies the Best Fit method by offsetting at a specified distance.

Anchor Type – Defines the reference position for constraints of translation or rotation. Selects the type of anchor used in the command. If the **Scan Data Center** option is selected, the center of the bounding box for selected scan data is shown and used as the anchor point. If the **Model Center** option is selected, the center of the target model will be used as an anchor point. If **Local Coordinate** is selected, you can define custom coordinate as the anchor point. If the **Global Origin** option is selected, the origin point in

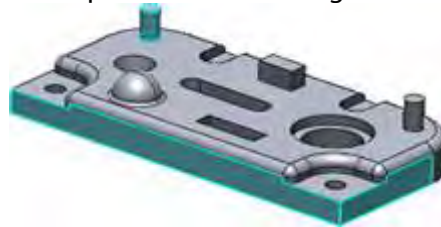
the global coordinate will be used as the anchor point.

Lock Translation/Rotation – Limits movement and rotation in each direction while scan data is aligned.

Ignore Outlier – Ignores data far from the average during calculation.

Datum – Used for alignment by selecting geometry such as a plane, cylinder, sphere, or vertex on a body, allowing the selection of a device and a method to acquire pair geometry from a real-part. After acquiring all geometries, the application performs **Datum** alignment.

Type	Device	Method
Vector	Probe	Intersect 2 Planes
Plane	Probe	Fit
Plane	Probe	Fit



2/2 Scan Stage



Device

Laser Scanner – Acquires data using a scanner.

Touch Probe – Acquires data using a probe.

Feature Type (Laser Scanner)

Mesh / Point Cloud – Creates meshes or point clouds after scanning.

Ref. Geometries – Creates a ref. plane after scanning.

Surface Body – Creates a surface body by averaging scanned points. You can create a plane, cylinder, cone, sphere and torus after scanning.

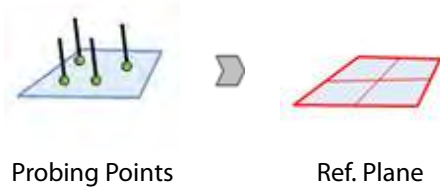
Solid Body – Creates a solid body by averaging scanned points. You can create a cylinder, cone, sphere and torus after scanning.

Feature Type (Touch Probe)

Mesh / Point Cloud – Creates meshes or point clouds after probing.

Ref. Geometries – Creates ref. geometries after probing.

Ref. Planes – Creates a ref. plane by fitting probing points.

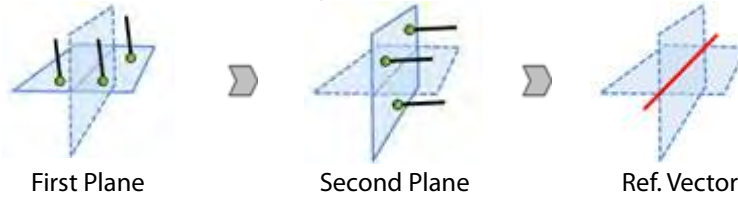


Probing Points

Ref. Plane

Ref. Vector – Creates a ref. vector using one of the following methods:

Intersect 2 Planes – Creates two planes by probing, and then generates a ref. vector at the intersecting line of the two planes.



First Plane

Second Plane

Ref. Vector

Fit Without Compensation – Creates a ref. vector by using use probe centers to average probing points without compensation.



Probing Points

Ref. Vector

Projection And Fit Without Compensation – Creates a projected ref. vector from probing points without compensation.



Base Plane

Probing Points

Ref. Vector

From Fitting Cylinder – Creates an axial ref. vector of a cylinder that was created by fitting probing points. At least seven points are required to create a cylinder.



Probing Points

Ref. Vector

From Fitting Cone – Creates an axial ref. vector of a cone that was created by fitting probing points. At least eight points are required to create a cylinder.

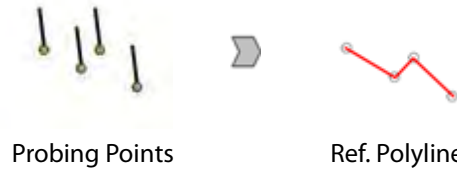


Probing Points

Ref. Vector

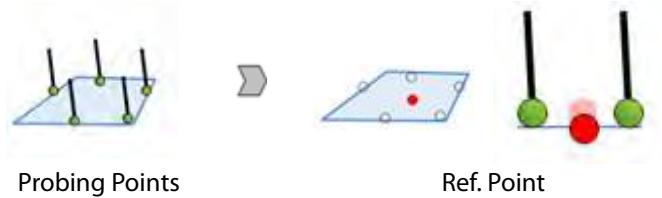
Ref. Polyline – Creates a ref. polyline from probing points without

compensation.

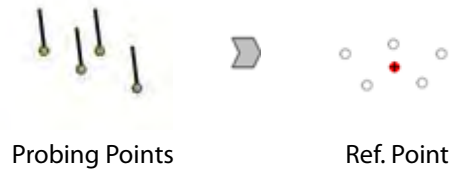


Ref. Points – Creates a ref. point using one of the following methods:

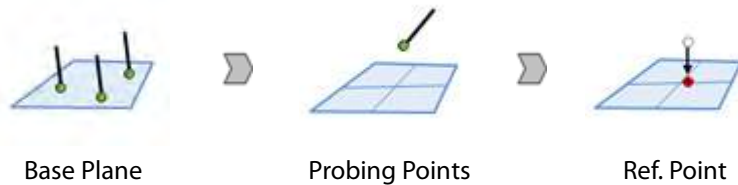
Average – Internally creates a plane by fitting probing points and compensates a plane. The averaged probing point will be projected onto the compensated plane.



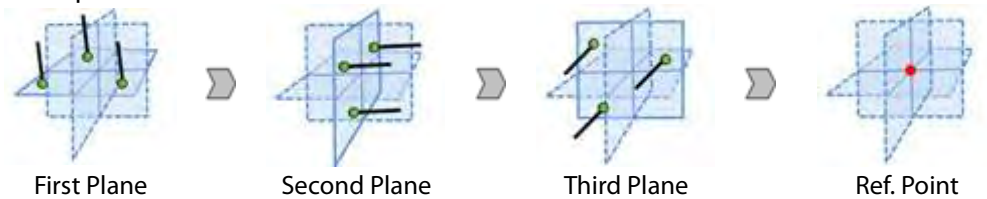
Average Without Compensation – Creates a ref. point by averaging probed points without compensation.



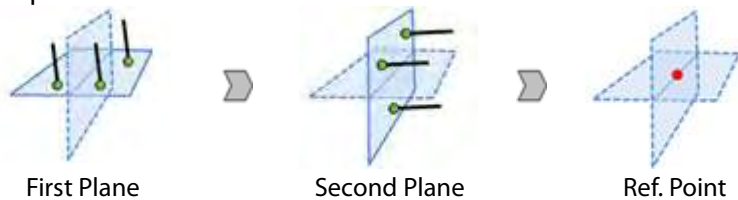
Projection And Average Without Compensation - Creates a projected ref. point without compensation.



Intersect 3 Planes – Creates a ref. point at the intersecting position of three planes.

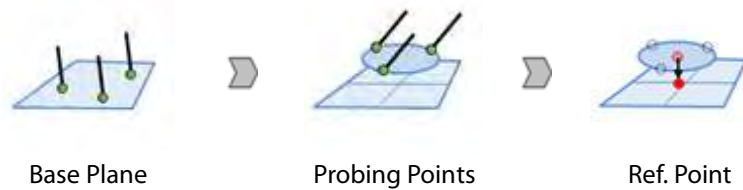


Intersect 2 Planes – Creates a center point at the intersecting line between two planes.



From Fitting Circle – Creates a projected ref. point from the center position

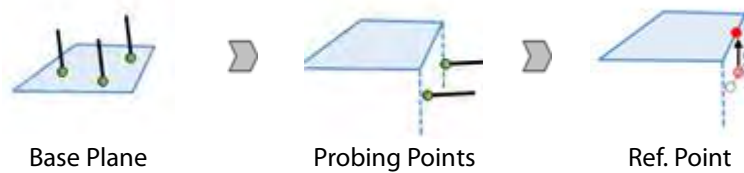
of a fitted circle.



From Fitting Sphere – Creates a ref. point at the center of a fitting sphere.



From An Edge Of A Plane – Creates an edge point on a ref. plane.



Intersection Points On A Plane – Creates a ref. point at the intersecting position between a plane and a line generated by two points.



Instant Point Without Compensation – Creates ref. points by a single acquisition of the A button.



Surface Body – Creates a surface body by averaging probed points after probing. A plane, cylinder, cone, sphere, or torus can be created.

Solid Body – Creates a solid body by averaging probed points after probing. A cylinder, cone, sphere or torus can be created.

Sketches – Creates sketch entities after probing. A line, rectangle, circle, point, or slot can be created.

3D Sketches – Creates a spline after probing.

Set Name Prefix – Defines the prefix of an entity name.

Use Device Buttons To Change LiveScan™ Option – Allows changes in options in the LiveScan™ dialog tree by using scanner buttons.

Insert > Scanner Direct Control > Geomagic Capture

The Geomagic Capture 3D scanner is a structured-light scanning system that uses a blue LED light system. When using the Geomagic Capture command the scanner should be installed in advance.


Supported devices

Device	32-bit	64-bit
Geomagic Capture	Yes	Yes

Note:

Geomagic Capture is not supported on Microsoft Windows XP.

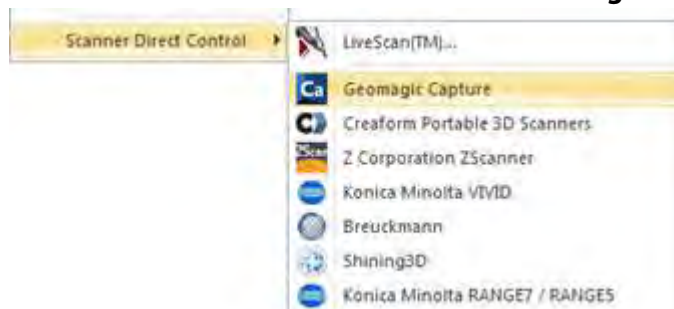
Scanner Specifications

	Scan Speed		0.3 seconds per scan
	Field of View (diagonal) / Near End – Far End		172 – 260 mm
	Field of View (X-Y) / Near End – Far End		124 x 120 mm – 192 x 175 mm
	Clearance Distance		300 mm
	Depth of Field		180 mm
	Resolution	Average Points	985,000 per scan
		Average Polygons	1.97 million per scan
		Point to Point Distance	0.162 mm (At center of measurement volume)
	Accuracy / Near End – Far End		60 microns – 118 microns (µm)
	Calibration		Pre-calibrated

How to Use

How to Create a Scan Using Geomagic Capture

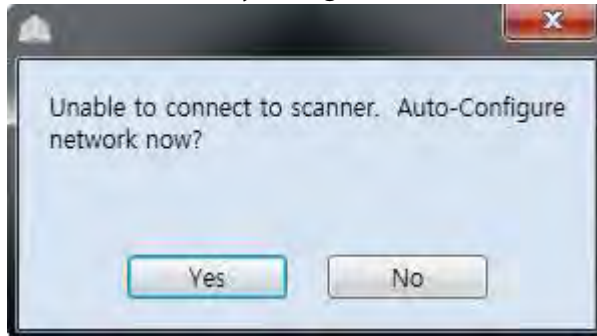
1. Connect a Geomagic Capture 3D scanning device to the PC.
2. Go to **Insert > Scanner Direct Control > Geomagic Capture**.



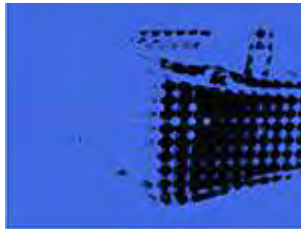
3. The command will initiate. Click the **Connect** button to connect to the Geomagic Capture

3D scanning device.

4. If it's the first time using the scanner it must be configured to the network so a dialog box asking to Auto-Configure the network will be displayed. Click **Yes** to auto-configure or click **No** to manually configure.



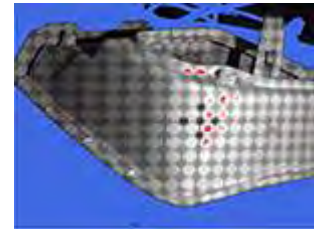
5. Adjust the Exposure setting to get an ideal scan. A scan that shows a lot of red is over-exposed, and a scan that shows a lot of blue and black is under-exposed.



Under-Exposed



Over-Exposed



Optimal

Note:

Scan quality can vary by the following factors:

- **Scanner calibration**

The more calibration scans you take, the better the accuracy of the scanner.

- **Environmental conditions**

Light and vibration can cause inaccuracies. Ensure the system is stable and the exposure is set to the conditions.

- **Stability of the scanner and the object**

If either the scanner or the object move during the scan, the data will be inaccurate. Ensure both are stable and secure.

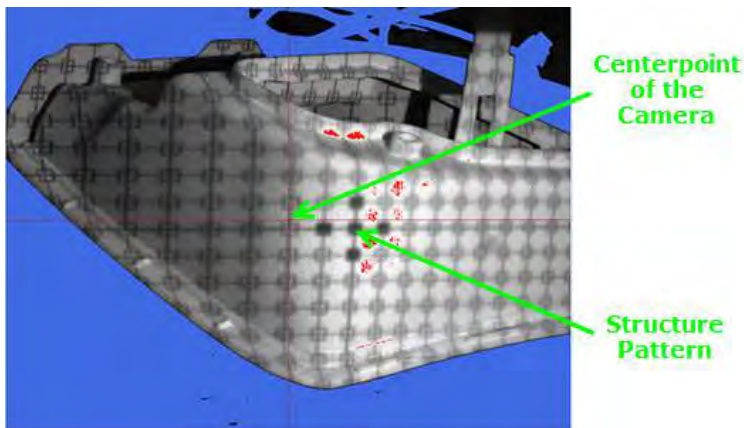
- **Shape and size of the object**

If the part you want to scan has limited features, you must use markers or spheres to align the data.

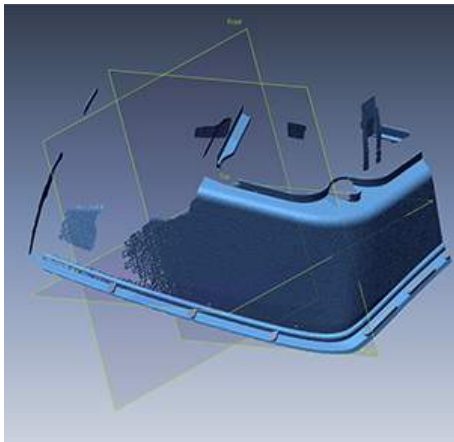
6. The red crosshairs show the centerpoint of the camera and the gray cross-section on the object shows the structure pattern. Align the structure pattern to the centerpoint of the camera.

Note:

An exact alignment is not necessary. As long as the structure pattern and the centerpoint of the camera are close to each other a satisfactory scan can be created.



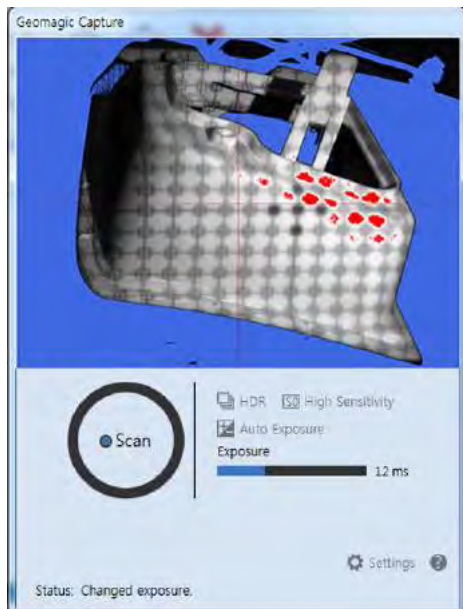
7. Then click the **Scan** button. A scan will be taken and imported in the application.



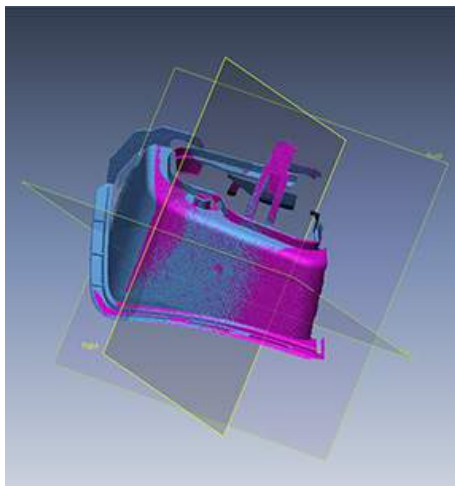
8. Reposition the model and click the **Scan** button again to take another scan.

Note:

If a scan was made using the **Best Fit** alignment method, make sure at least 30% of the original scan area is in the next scan. If the **Target** alignment method was used, make sure that at least three targets from the first scan area are visible in the next scan.



9. The second scan will be imported into the application and shown in a magenta color.



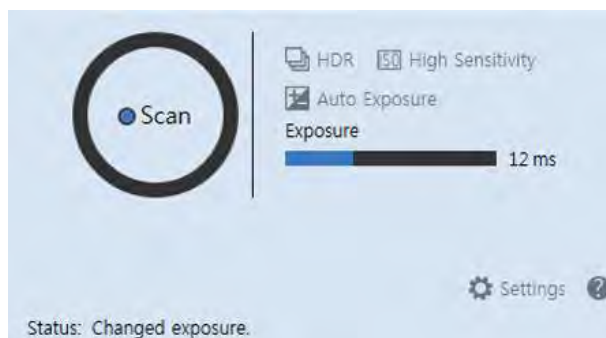
Note:

During a scanning process when a new scan is made the previous scan will change from magenta to blue and the new scan will be shown in magenta.

10. Continue scanning until the project is completed and click the **Accept** button. The **Mesh Buildup Wizard** will initialize.

Options in Detail

Options



Scan – Takes a scan shot and sends it to the Geomagic application.

HDR – Scans multiple times at varying exposure levels in order to capture objects with large contrast variances.

High Sensitivity – Special scanning mode that allows for the capture of noisy surfaces that would otherwise be difficult to capture.

Auto Exposure – Automatically controls the exposure setting of the 3D scanner.

Exposure – Changes the level of light that is sent and received by the scanner.

Settings – Goes to the settings menu.

Status – Shows the current working status.

Help – Shows the Help document.

Settings



Auto Align – Automatically aligns scans using either the Best Fit or Target methods.

Best Fit – Aligns a current scan shot to the previous scan shot. All other scan shots will be ignored.

Target – Uses targets on a real part to align.

Note:

When using the **Auto Align** method, refer to the tips below:

- Prevent the scan floor from being captured in a scan
- Apply white scanning spray or powder evenly and make sure the object is adequately covered
- Dim the lights
- Use proper sized target markers

Resolution – Changes the resolution of the scanner.

High – Constructs a high-density mesh (1/1 capture ratio).

Medium – Constructs a medium-density mesh (1/4 capture ratio).

Low – Constructs a low-density mesh (1/9 capture ratio).

Scanner Setup

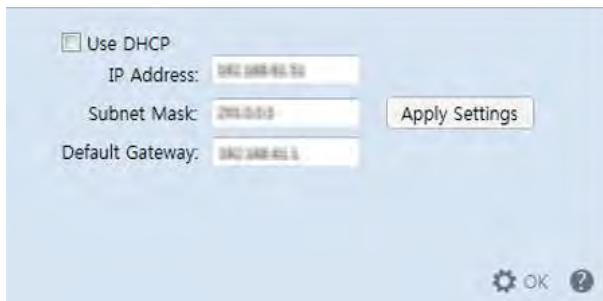
Auto Connect – Connects the scanning device by automatically configuring a network.

Manual Setup – Connects the scanning device by manually setting a network configuration.

Firmware Version

Update Firmware – Checks for new firmware and updates the current firmware if a newer version exists.

Scanner Setup



Use DHCP – Uses the DHCP IP protocol instead of a static IP address.

IP Address – Used to manually set a static IP address.

Subnet Mask / Default Gateway – Used to manually set a subnet mask and a default gateway.

Apply Settings – Applies changes made to the Use DHCP radio box as well as changes made to the IP Address, Subnet Mask, and Default Gateway input fields.

Insert > Scanner Direct Control > Creaform Portable 3D Scanners

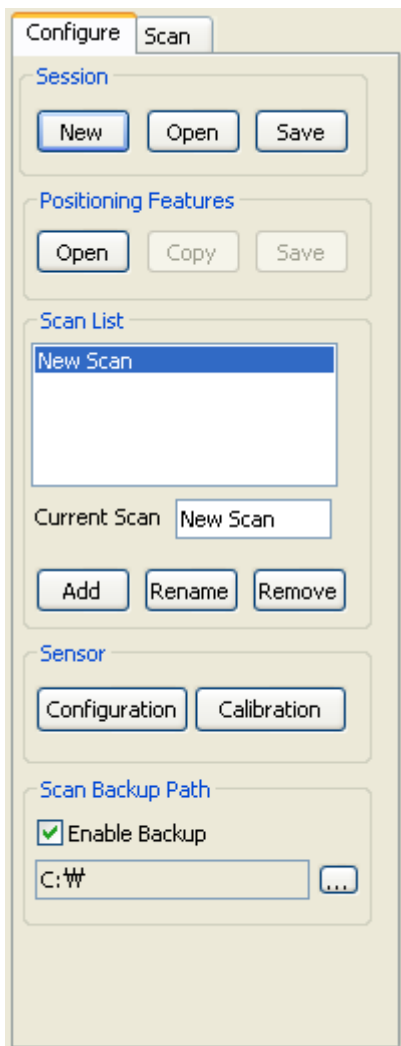
VXscan or VXelements should be installed in advance to directly control the Creaform Portable 3D Scanners.

Supported devices

Device	32-bit	64-bit
Portable 3D Scanners	Yes	Yes

Options in Detail

1. Configure



Session

New – Starts a new session.

Open – Opens an existing session file (.CSF).

Save – Saves the current session (.CSF).

Positioning Features

Open – Opens a positioning target file (.TXT).

Copy - Copies positioning targets to a new scan.

Save - Saves positioning targets in a file (.TXT)

Scan List - Shows existing scans of the current session and highlights the current scan.

Add – Adds a scan to the current session.

Rename Scan - Changes the name of the current scan in the **Current Scan** box.

Remove Scan - Removes the current scan.

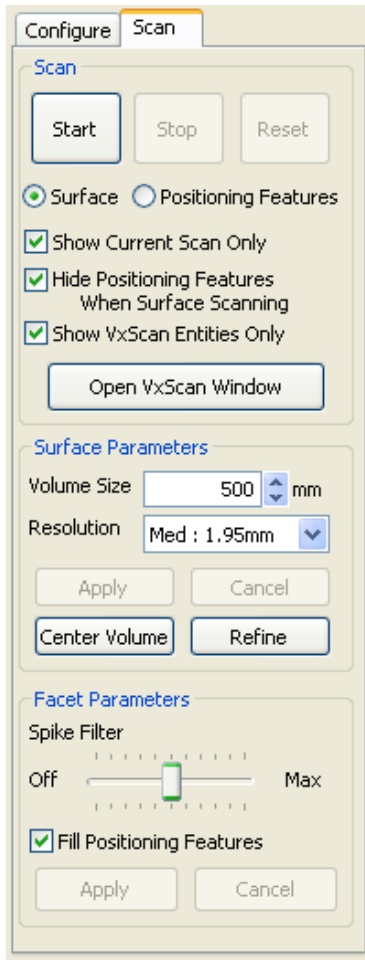
Sensor

Configuration - Shows the Sensor Configuration dialog box for Portable 3D Scanners.

Calibration – Shows the Sensor Calibration dialog box for Portable 3D Scanners.

Scan Backup Path – Sets the backup folder for scan data.

2. Scan



Scan

Start – Starts scanning.

Stop – Stops scanning.

Reset – Clears the scan data for the current scan.

Surface – Gathers surface data and positioning features during scanning.

Positioning Features – Gathers only positioning features during scanning.

Show Current Scan Only – Shows only the scan data of a selected scan.

Hide Positioning Features When Surface Scanning – Hides positioning features during scanning.

Show VxScan Entity Only – Shows only meshes and ref. point entities related to scan data.

Open VxScan Window – Opens the VxScan window and enables users to access all functions that VxScan provides.

Surface Parameter - Sets scanning volume size or resolution.

Volume Size – Adjust surface reconstruction settings and sets the scanning volume. Changing the volume size directly affects the voxel size of each resolution level.

Resolution – Select High, Medium or Low resolution.

Apply - Confirms settings.

Cancel – Restores a previous setting.

Center Volume - Centers the volume on an object being scanned.

Refine - Performs an operation on a surface that decreases the impact of possible positioning errors that may occur during scanning.

Facet Parameter

Spike Filter – Determines the level of spike filtering.

Fill Positioning Features – Fills holes created by the positioning features on an object.

Apply – Confirms settings.

Cancel – Restores a previous setting.

Insert > Scanner Direct Control > Z Corporation ZScanner 700

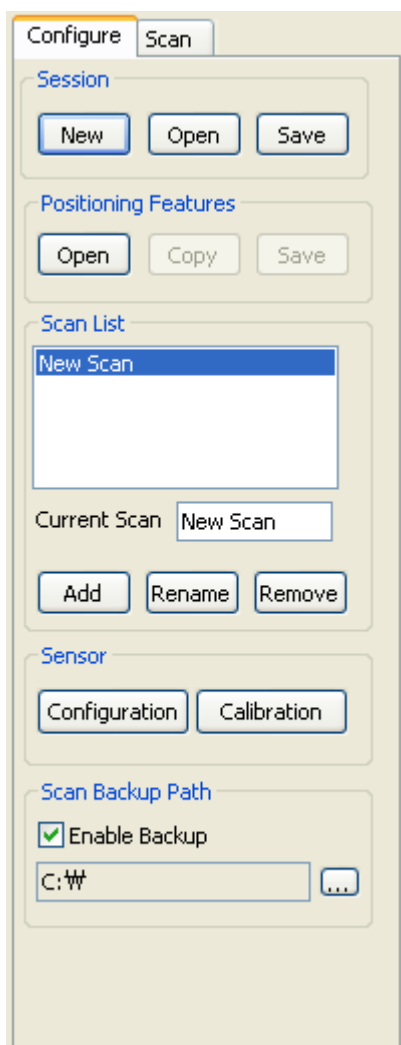
ZScan should be installed in advance to directly control a ZScanner 700 scanner.

Supported devices

Device	32-bit	64-bit
ZScan	Yes	Yes

Options in Detail

1. Configure



Session

New – Starts a new session.

Open – Opens an existing session file (.CSF).

Save – Saves the current session (.CSF).

Positioning Features

Open – Opens a positioning target file (.TXT).

Copy - Copies positioning targets to a new scan.

Save - Saves positioning targets in a file (.TXT).

Scan List - Shows existing scans of the current session and highlights the current scan.

Add – Adds a scan to a current session.

Rename Scan - Changes the name of the current scan in the **Current Scan** box.

Remove Scan - Removes the current scan.

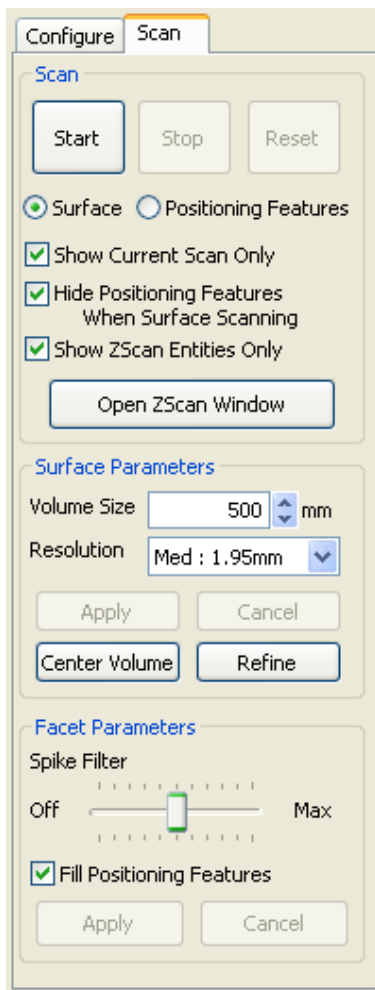
Sensor

Configuration - Shows the Sensor Configuration dialog box for ZScan.

Calibration – Shows the Sensor Calibration dialog box for ZScan.

Scan Backup Path – Set the backup folder for scan data.

2. Scan



Scan

Start – Starts scanning.

Stop – Stops scanning.

Reset – Clears the scan data for the current scan.

Surface – Gathers both surface data and positioning features during scanning.

Positioning Features – Gathers only positioning features during scanning.

Show Current Scan Only – Shows only the scan data of a selected scan.

Hide Positioning Features When Surface Scanning – Hides positioning features during scanning.

Show ZScan Entity Only – Shows only meshes and ref. point entities related to scan data.

Open ZScan Window – Opens the ZScan window and allows users to access all functions that ZScan provides.

Surface Parameter - Sets scanning volume size or resolution.

Volume Size – Adjust surface reconstruction settings and sets the scanning volume. Changing the volume size directly affects the voxel size of each resolution level.

Resolution – Select High, Medium or Low resolution.

Apply - Confirms settings.

Cancel – Restores a previous setting.

Center Volume - Centers the volume on an object being scanned.

Refine - Performs an operation on a surface that decreases the impact of possible positioning errors that may occur during scanning.

Facet Parameter

Spike Filter – Determines the level of spike filtering.

Fill Positioning Features – Fills holes created by the positioning features on an object.

Apply – Confirms settings.

Cancel – Restores a previous setting.

Insert > Scanner Direct Control > Konica Minolta VIVID

Supported devices

Device	32-bit	64-bit
VIVID 910	Yes	Yes
VIVID 9i	Yes	Yes

Options in Detail

1. General



Scan

Name – Used to input the name of a scan shot and its suffix.

Auto Align – Automatically aligns captured scan shots.

Rotary Axis is used for a rotary table device, and **Geometry** performs auto alignment with an application algorithm. The **With Only Last Scan** option performs alignment between the last scan shot and a previous one.

Save Raw Data – Saves raw scan data to a desired directory.

Preview – Shows a preview image with a color map. Red represents near distances from a scanner and blue represents far distance. Anything displayed in black will not be seen by a scanner, because it is out of range. When a preview is satisfactory, click **Accept**.

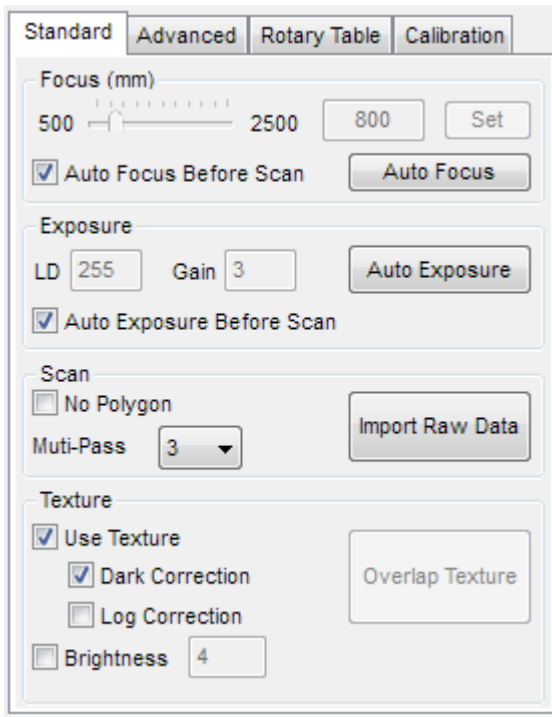
Scan – Starts scanning.

Auto Scan – Starts auto scanning using a rotary table. Parameters can be set in the **Rotary Table** tab in the **Auto Scan** option.

Digitizer Parameter – Applies parameters from the **Standard** and **Advanced** tab to a scanner device when the **Set** button is clicked. The **Get** button retrieves the parameters from a scanner device to the **Standard** and **Advanced** tab.

Mode – Select the **Standard**, **Extend** or **User** mode provided by a scanner device.

2. Standard Tab



Focus – Sets a focusing distance. Clicking **Auto Focus**, will set a scanner to find the best focusing distance automatically. Checking the **Auto Focus Before Scan** option will allow a scanner to **Auto Focus** whenever it scans.

Exposure – Sets the exposure of scanner device.

LD - LD (Laser Diode) Power.

Gain - CCD Gain.

Auto Exposure Before Scan – Sets exposure values automatically before starting a scan.

Auto Exposure – Finds the best exposure for scanning. Checking the **Auto Exposure Before Scan** option allows a scanner to use **Auto Exposure** whenever it scans.

Scan

No Polygon – Gathers only point clouds.

Multi-Pass – Determines the number scans.

Import Raw Data – Imports .cdk and .cdm file formats.

Texture

Use Texture – Gathers textures. To correct a texture color, use the **Dark Correct** or **Log Correction** options.

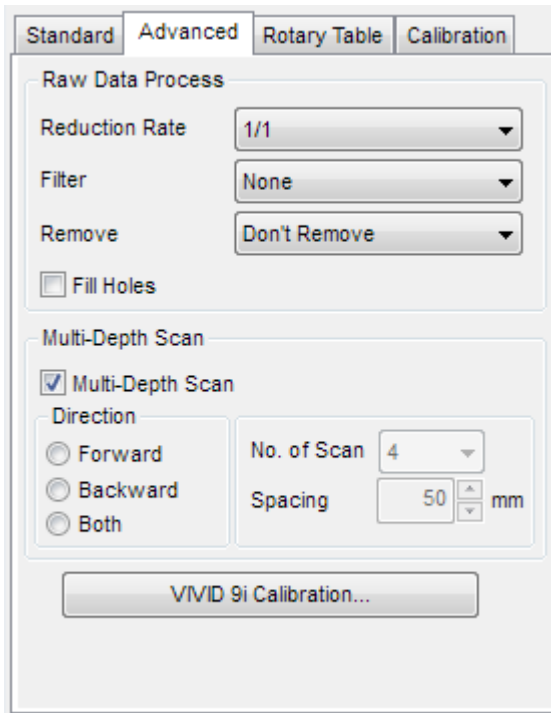
Dark Correction - Applies dark correction to color data. Use for striation-type noise in images.

Log Correction - Applies logarithmic correction to color data. Used to brighten neutral colors.

Brightness – Corrects texture brightness.

Overlap Texture - Uses the texture of a current model and overlaps it onto previously scanned data. A non-textured model can overlap another texture in different illumination conditions.

3. Advanced Tab



Raw Data Process

Reduce Ratio – Decides the sampling ratio used to import scan data.

Filter - Decides the filtering method for poly-vertices.

The **None** method will not filter any poly-vertices.

The **Noise Filter** method will filter noisy poly-vertices.

The **High Quality** method will filter unreliable data but is only effective for importing scan data obtained using a Vivid910.

The **Noise Filter & High Quality** method uses both the **Noise Filter** and **High Quality** methods and is also only effective for importing scan data obtained using a Vivid910.

Remove – Chooses a method to remove unnecessary parts while importing scan data.

The **Don't Remove** method will not remove any parts

The **Bound** method removes boundary points.

The **Bound & xx Degree** method removes boundary points and polygons within a corresponding angle range between 5, 10, 15, and 20 Degrees to a view vector.

Fill Holes – Fills small holes in a mesh.

Multi-Depth Scan

Multi-Depth Scan – Executes multiple scans while changing distances.

For example, setting **Spacing** to **20** and **Scan Num** to **3** when the distance is **600**, will scan at the 580, 600, and 620 positions and create three meshes in the working window with the names "Default - 000deg: 01", "Default - 000deg: 02" and "Default - 000deg: 03".

Direction – Selects the direction for multi scanning.

Forward – Starts scanning in the forward direction from a current focus length.

Backward – Starts scanning in the backward direction from a current focus length.

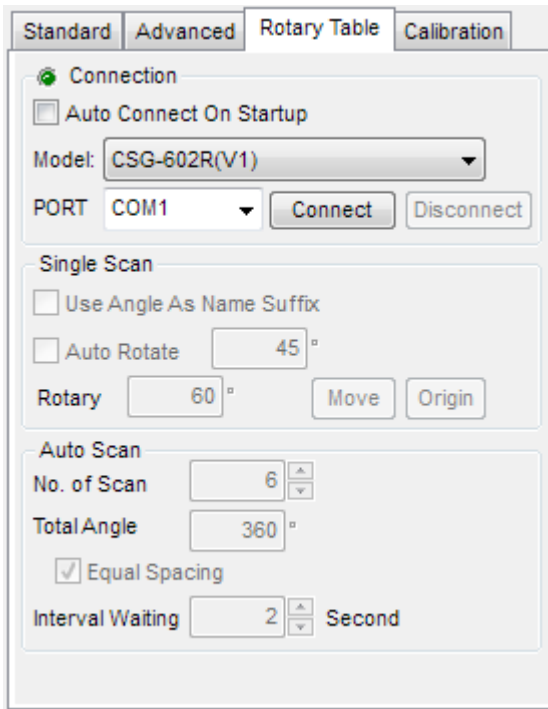
Both – Starts scanning in the forward and backward direction from a current focus length.

No. of Scan – Defines how many scan shots will be captured.

Spacing – Defines distance increments for focus length.

VIVID 9i Calibration – Calibrates a VIVID 9i scanner which requires a calibration jig.

4. Rotary Table Tab



Connection

Auto Connect On Startup – Automatically communicates with a rotary table upon startup.

Model – Selects a rotary table which you want to connect.

PORT – Selects the port used to communicate with a rotary table.

Connect – Starts communication with a rotary table.

Disconnect – Stops communication with a rotary table.

Single Scan

Use Angle As Name Suffix – Uses a specified angle for the file name suffix.

Auto Rotate – Automatically turns a rotary table by a specified angle after scanning.

Rotary – Manually turns a rotary table.
The **Move** button starts to turn a rotary table.
The **Origin** button moves a rotary table to its origin position.

Auto Scan – Sets the parameters used for an automated scan.

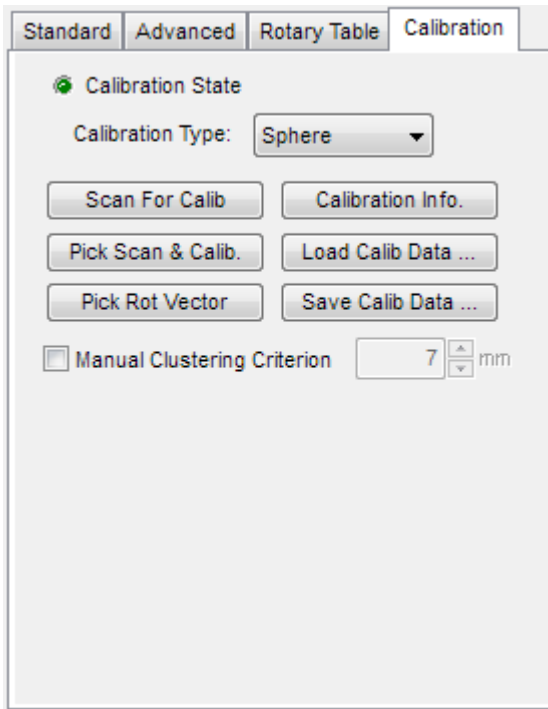
No. of Scan – Sets the number of rotations.

Total Angle – Sets the entire angle of scanning.

Equal Spacing – Applies scanning with an equal angle decided by the **No. of Scan** and **Total Angle** options.

Interval Waiting – Sets the value for scanning after a rotary table has been turned. A time can be set to prevent a model from shaking by the sudden stop of a rotary table.

5. Calibration Tab



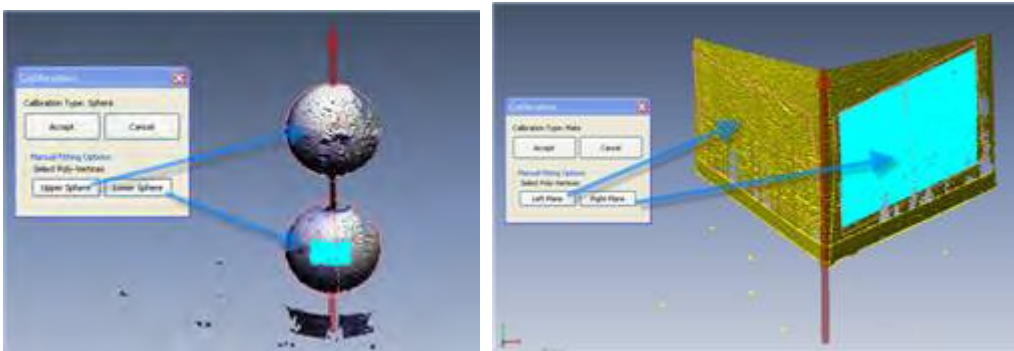
The **Calibration** tab contains parameters for calibration of a rotary table.

Calibration State shows the calibration status or if a stored calibration has been successfully loaded. If it is not lit a calibration is necessary.

Calibration Types chooses the method of calibration. The default is **Plate** for plates provided by Konica Minolta. When a calibration ball provided by 3D Systems is used the **Sphere** type needs to be used.

Scan For Calib uses a jig used for calibration that can be scanned. The axis vector of the rotary table can automatically be found by processing the scan data of a part in the application.

If the results of an axis vector are not satisfactory, it can be manually re-assign by selecting scan areas using the **Upper Sphere**, **Lower Sphere**, **Left Plate**, and **Right Plate** buttons. The new axis vector will be recalculated from the manual selection of areas.



When scan data contains noise, the application may fail to calibrate and the scan data may need to be edited by making a reference vector in the application. After editing scan data, the **Pick Rot Vector** option can be used to set a rotation vector. Click the **Pick Scan & Calib.** Option and select a scan data to perform calibration.

Calibration Info. shows the current calibration information.

Load Calib Data will load calibration information previously stored in a file.

Save Calib. Data will save the current calibration information to a file.

Manual Clustering Criterion can be used to classify clusters for calibration purposes and if it is not checked, the value will be automatically calculated.

Insert > Scanner Direct Control > Breuckmann

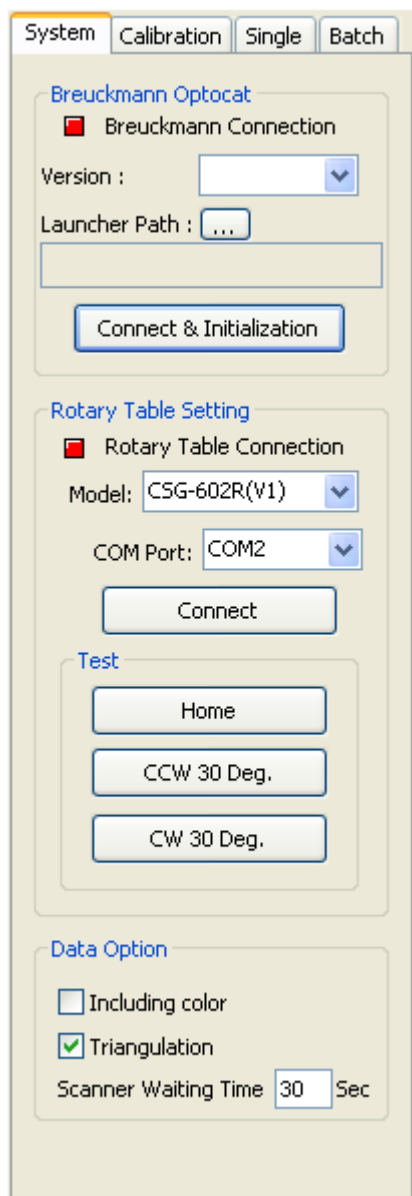
To execute the **Breuckmann** command, previously, **Optocat** should be installed on your machine.

Supported devices

Device	32-bit	64-bit
Breuckmann	Yes	Yes

Options in Detail

1. System



Breuckmann Optocat

Version – Chooses an installed version of Optocat.

Launcher Path – Chooses the file path of the Optocat .exe file.

Connect & Initialization – Connects a scanner and defines the initial setup preferences supported by Optocat. The color of 'Breuckmann Connection' will change from red to green when successful.

Rotary Table Setting

Model – Chooses a type of a rotary table.

COM Port – Chooses a port for the application to communicate with a rotary table. If a rotary table is connected by USB, this option will be disabled.

Connect – Connects a selected rotary table. The color of 'Breuckmann Connection' will change from red to green when successful.

Test – Tests to verify that a rotary table is working correctly.

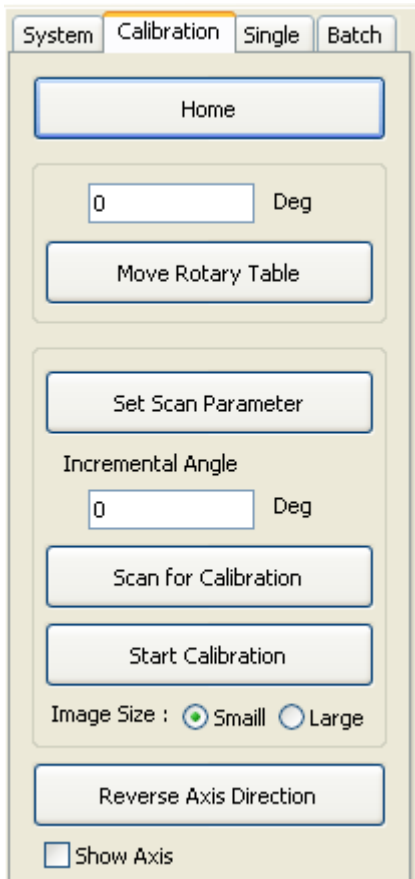
Data Option

Including Color – Imports color information.

Triangulation – Starts triangulation when scan data is acquired.

Scanner Waiting Time – Sets the time between scanning intervals.

2. Calibration



Home – Rotates a rotary table to a default position.

Move Rotary Table – Rotates a rotary table by a specific angle.

Set Scan Parameter – Opens the option dialog box of Optocat.

Incremental Angle– Specifies an angle between scans.

Scan for Calibration – Calibrates by scanning three times by a specified angle.

Start Calibration – Opens the Calibration Window dialog box and calculates the rotational axis of a rotary table by picking markers or spheres on three images and clicking the **Start Calibration** button.

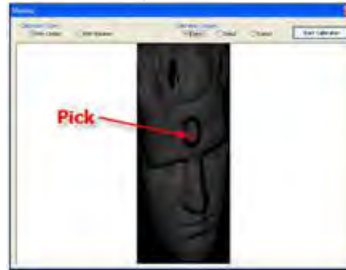


Image Size – Selects an image size in the **Calibration Window** dialog box.

Reverse Axis Direction – Flips a rotation axis.

Show Axis – Displays an axis.

3. Single



Home - Rotates a rotary table to a default position.

Move Rotary Table – Rotates a rotary table by a specific angle.

Set Scan Parameter - Opens the option dialog box of Optocat.

Scan – Starts scanning.

Auto Align – Automatically aligns scan shots.

Geometry – Aligns scan data using a feature-to-feature recognition alignment algorithm.

Rotary Axis – Aligns scan data using the axis of a rotary table that can be found in the Calibration tab.

With Only Last Scan – Performs alignment with the last scan.

4. Batch



Auto Filling – Displays the **Auto Filling** dialog box where the **Start / End Angle** and **Count** options can be utilized. Angles will be displayed in the **Scan List** box.

Clear – Removes all angles in the Scan List box.

Set Scan Parameter - Displays the option dialog box of Optocat appears.

Batch Scan – Starts scanning.

Align Geometry – Aligns scan data using a feature-to-feature recognition alignment algorithm.

Rotary Axis – Aligns scan data using the axis of a rotary table that can be found in the **Calibration** tab.

Insert > Scanner Direct Control > KODEN OCM-A (32 bit only)

The default setting of **KODEN OCM-A** devices are included in OCM-A_Info.ini.file. The file is located in ~<Product Installation Path>\bin\KODEN OCM-A.

Supported devices

Device	32-bit	64-bit
KODEN OCM-A	Yes	No

Options in Detail

Connect – To communicate with **KODEN OCM-A** devices, click this button. When connected to the device, the button will change to **Disconnect**.

Auto – Automatically applies the Zoom fit function while scanning is in progress.

ZoomFit – Displays and refreshes an entire model during scanning.

Insert > Scanner Direct Control > Roland LPX (32 bit only)

Supported devices

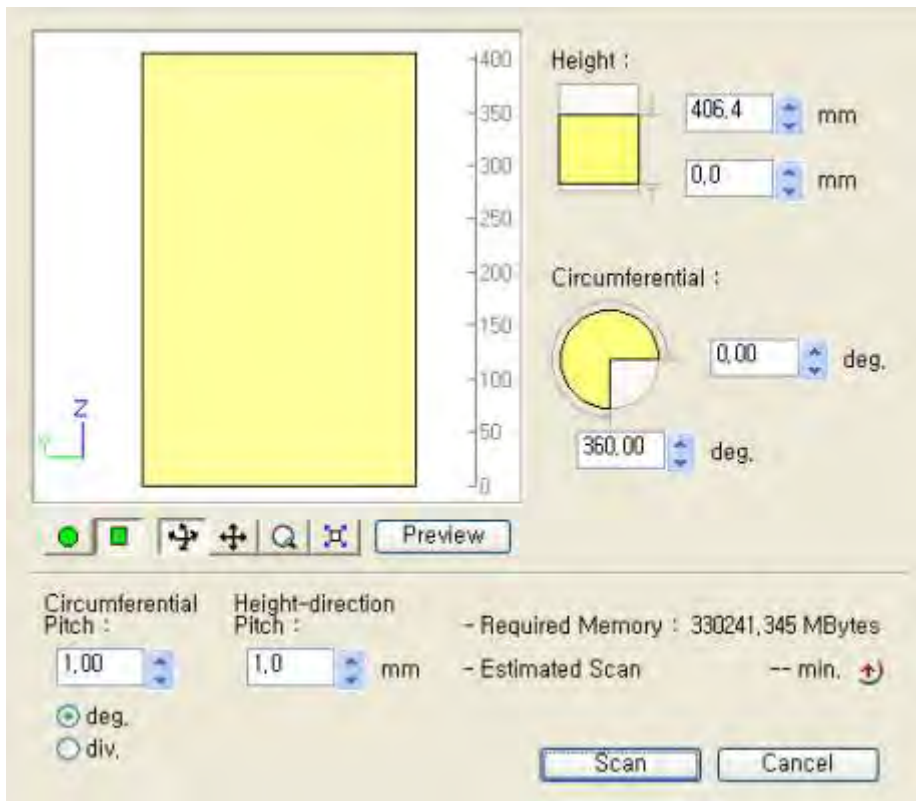
Device	32-bit	64-bit
LPX Series	Yes	No

Options in Detail

Connect – Communicate with a **Roland LPX** scanner.

Port – Selects a port.

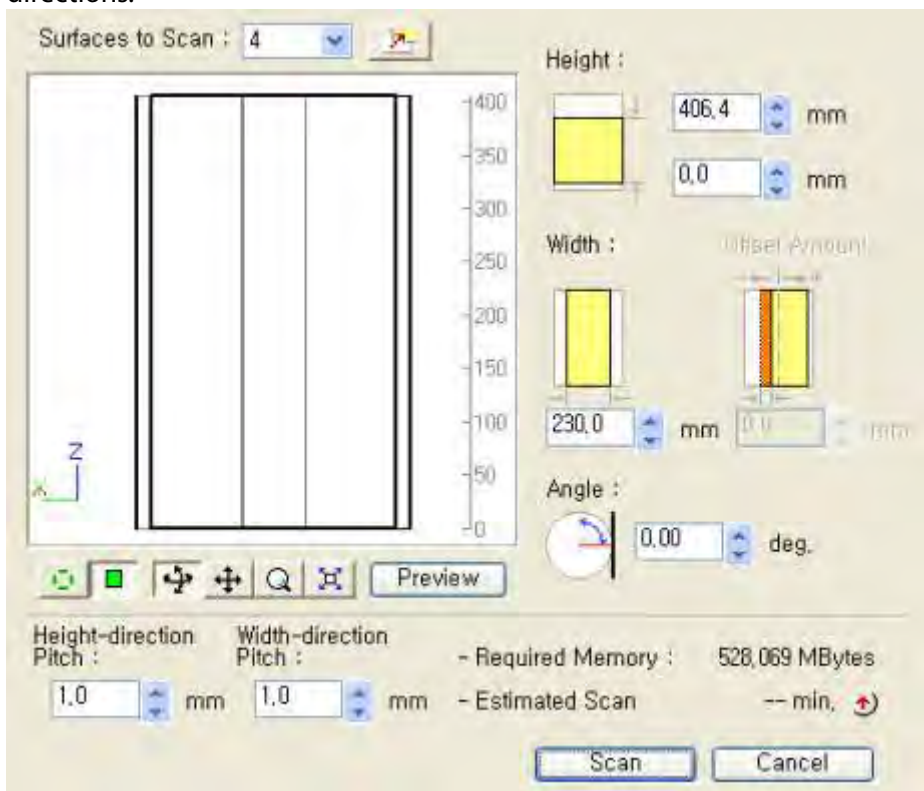
Rotational Scan – Set scanning options for scanning a model while it is rotating.



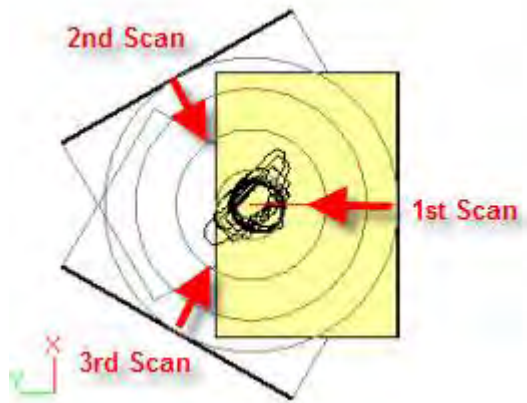
A scanning area is defined by the **Height** and **Circumferential** input boxes. If a scanning size cannot be initially defined, click the **Preview** button and an approximate height will be generated for a **Height** value.


Scanning pitch is defined in the **Circumferential Pitch** and **Height-direction Pitch** input boxes. After scan settings are define scanning time can be estimated by using the up arrow button. To start scanning, click the **Scan** button.

Planar Scan – Makes a square of the scanning area and scans a model in several rotational directions.



A total number of scans can be defined in the **Surfaces to Scan** input box. If 3 is input, a model will be scanned 3 times.

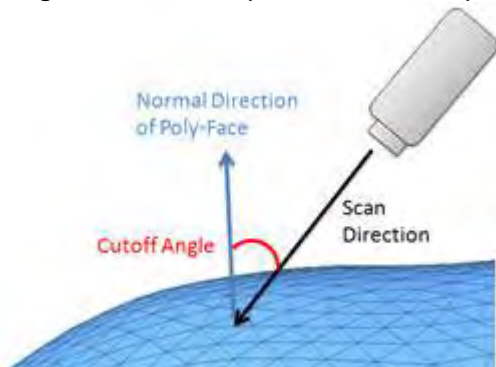


To change an initial scanning direction, click the  button. A scanning area is defined by the **Height** and **Width** input boxes. If a scanning size cannot be initially defined, click the **Preview** button and an approximate height will be generated for a **Height** value.

Scanning pitch is defined by the **Circumferential Pitch** and **Width-direction Pitch** input boxes. After scan settings are defined scanning time can be estimated by using the up arrow button. To start scanning, click the **Scan** button.

Points Only – Imports scan data as a point cloud.

Cutoff Angle - Specifies an angle between a scan direction and a normal direction of poly-face mesh to be generated. If the angle is larger than a user defined value, a poly face mesh will not be generated. This option is useful for preventing scan data from containing unwanted spikes.



Insert > Scanner Direct Control > Konica Minolta RANGE5/RANGE7 (64 bit only)

To execute the **RANGE5** and **RANGE7** command, previously, **RangeViewer** should be installed on your machine.

Options in Detail

Connect – Connects to a RangeViewer after setting a Path.

Points Only – Imports only point data from **RangeViewer**.

Suppress – Used to suppress data when acquiring point cloud or mesh data of large sizes from a scanner. The suppressed data will be imported and loaded on the hard disk.

RangeViewer Path – Automatically sets a default installation path when the application is first installed. If the installation folder is changed or move, this option can be used to set the correct path for **RANGEVIEWER.exe**.

Insert > Scanner Direct Control > 3D Digital Scanner (32 bit only)

RealScan should be installed in advance to directly control a 3D digital scanner.

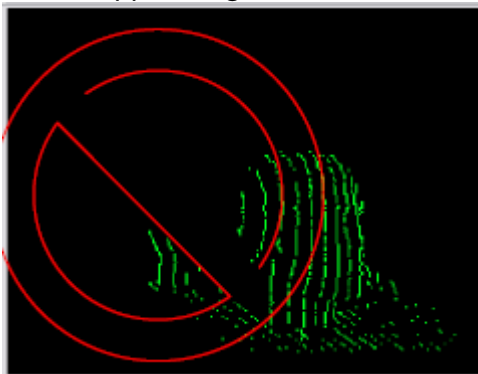
Supported devices

Device	32-bit	64-bit
3D Digital Scanner	Yes	No

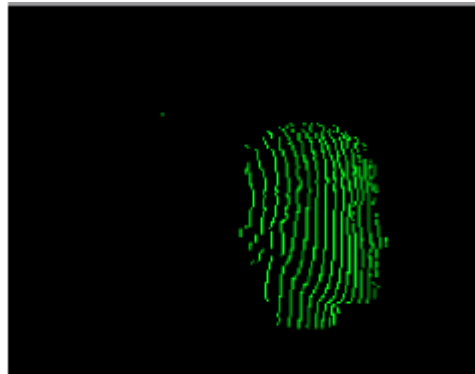
Options in Detail

Scanner – Chooses a scanner to directly control in the application.

Gain / Threshold – Allows users to specify scan density. The number of lines is adjustable up to 1000 and the higher the density of points, the longer it will take to scan. Along with the resolution control, users can also adjust gain and stripe threshold settings. The stripe threshold and gain settings allow users to adjust camera sensitivity to capture more or less points on the surface of an object to be scanned. Darker objects will need higher gain settings so that the laser will appear brighter to the camera.



Noisy Scan



Good Scan

1000 / 255 Points / Lines – Controls distribution points consisting of the number of points per line by the number of lines per scan. There are 2 modes for scanning, **255 Points** or **1000 Points**. The resolution of a scan can be changed by adjusting the number of **Points** and **Lines**. The default number of points per laser stripe will be 255 in a fast scan mode. EScan models can only scan in modes of 255 points, while Optix series scanners can scan up to 1000 points. The number of lines can be increased up to 1000.

Z Range Cut Off – A **Z Range Cut Off** setting is provided to help remove the foreground and backdrop from a model. To eliminate the foreground and backdrop points, reduce the maximum Z and increase the minimum Z using the **Set Cutoff Plane** sliders. The **Far** slider sets the location of the far cutoff plane and the **Near** slider sets the location of the near cutoff plane. Users can adjust scan volume in real time when the **Preview** button is used. To open Z limits, click the **Extend DOF** button. To return back to calibrated limits, push the button again.

X Range Limits – The **X-Range Limits** allows users to control an area being scanned in the X axis. A laser sweeps left to right but the length of each sweep can be narrowed if a user is scanning a smaller object. The **Laser On** button will turn on the laser to allow users to see the laser position. To adjust limits, move the slider bar to a desired laser position and click the **L** or **R** button to set either the left or right limits of scanning. Clicking the double arrow button will move the laser to either extreme of the scanning field. To open up the X limits again, click the **Reset FOV** button.

Points Only – Creates a point cloud from a scan. To get mesh data from scanning, turn this option off.

Preview / Scan – Toggles the preview mode or take scans.

Insert > Scanner Direct Control > Shining3D

To directly control a Shining 3D scanner, Shining 3D's application should be installed in advance.

Supported devices

Device	32-bit	64-bit
Shining3D	Yes	Yes

Note:

If the application cannot find a Shining 3D application, a dialog box will be displayed to manually choose the Shining 3D's application execution file (.exe) when the **Insert > Scanner Direct Control > Shining3D** command is executed.

Note:

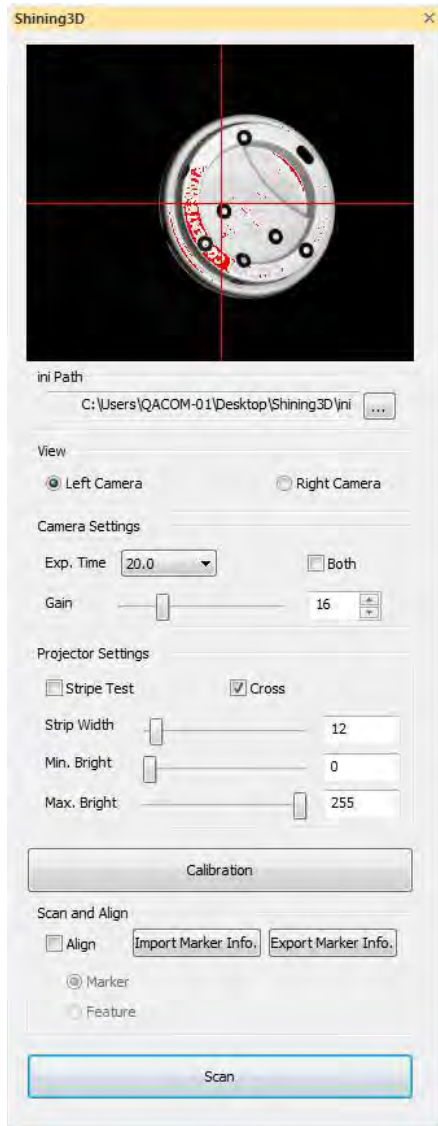
Scanner Direct Control Interface for Shining 3D is supported in the 32-bit version of Geomagic Design X only.

Options in Detail


1. Run Application

Manually choose the directory where the Shining 3D application execution file (.exe) and configuration files are located. Once the directory is set, it will be preserved for when the application is run again.

2. General



Ini Path – Shows the path where the configuration file (.ini) is located.

 – Manually chooses the location of the configuration file (.ini).

View – Shows the view of the Left Camera or the Right Camera.

Camera Settings – Specifies camera settings.

Exp. Time – Specifies the exposure time of camera.

Both – Applies camera settings to both the Left Camera and the Right Camera.

Gain – Adjusts camera sensitivity to capture more or less points on the surface of an object. Darker objects will need higher gain settings so that the laser will appear brighter to the camera.

Projector Settings – Specifies projector settings.

Stripe Test – Projects stripes on a target object.

Cross – Projects a crossed center on a target object.

Stripe Width – Sets the width of projected stripes by adjusting a slider.

Min. / Max. Bright – Sets the range of brightness by adjusting a slider. (0~255)

Calibration – Shows the Sensor Calibration dialog box for Shining 3D.

Scan and Align – Scans and aligns scan data.

Align – Aligns scan data.

Maker – Aligns scan data by using detected markers used during scanning.

Feature – Aligns scan data by using feature information. The application uses the Local Based On Auto Guess method for quick and accurate alignment. For more information, see [Tools > Scan Tools > Align Between Scan Data](#).

Import Marker Info. – Imports pre-defined markers information.

Export Marker Info. – Exports detected marker information during the scanning.

Scan – Starts scanning.

7.5. Tools

The **Tools** menu features commands to edit existing features or create a new feature.

Tools are divided into 5 tool groups used to:

Managing Features

- **Create new 3D Scan Data** (Related to [Scan Tools](#))
Creates and edits new 3D scan data from raw 3D scan data
- **Edit Mesh** (Related to [Mesh Tools](#))
Edits mesh in the Mesh mode
- **Manage Regions** (Related to [Region Tools](#))
Creates new regions on a mesh or edits them in the Region Group mode
- **Edit Point Clouds** (Related to [Point Cloud Tools](#))
Edits point clouds in the Point Cloud mode
- **Manage Texture** (Related to [Texture Tools](#))
Adds a new texture on mesh or edits it in the Mesh mode

Managing 2D Sketches

- **Create 2D Sketch Entities** (Related to [Sketch Entities](#))
Creates 2D sketch entities such as lines, arcs, circles, rectangles, and ellipses
- **Edit 2D Sketch Entities** (Related to [Sketch Tools](#))
Edits existing 2D sketch entities by filleting, offsetting, trimming, etc.
- **Use 2D Sketch Options** (Related to [Sketch Options](#))
Executes 2D sketch options while creating 2D sketch entities
- **Add Dimension Constraints** (Related to [Smart Dimension](#))
Measures dimensions and sets constraints for 2D sketch entities

Managing 3D Sketches

- **Create 3D Sketch Entities** (Related to [3D Sketch Entities](#))
Creates 3D sketch entities such as splines, sections, and boundaries
- **Edit 2D Sketch Entities** (Related to [3D Sketch Tools](#))
Edits existing 2D sketch entities by offsetting, trimming, extending, etc.

Managing Bodies

- **Manage Bodies** (Related to [Body Tools](#))
Finds defects in existing bodies

Aligning Features

- **Align Coordinates** (Related to [Align](#))
Moves features by aligning their coordinates

7.5.1. Tools > Scan Tools

The **Scan Tools** menu features various commands used to create new meshes from raw scan data as well as editing existing 3D scan data.

Scan Tools are divided into 2 tool groups used to:

Creating a New 3D Scan Data

- **[Mesh Buildup Wizard](#)**
Creates defect-free and watertight mesh models from raw 3D scan data through mesh building stages
- **[Triangulate](#)**
Creates mesh from raw point clouds
- **[Align Between Scan Data](#)**
Aligns 3D scan data
- **[Merge](#)**
Merges 3D scan data into a single mesh
- **[Combine](#)**
Combines multiple 3D scan data files into a single 3D scan data file
- **[Decimate Meshes](#)**
Reduces the number of poly-faces in a mesh

Editing 3D Scan Data

- **[Normal Information Manager](#)**
Manages normal information in 3D scan data
- **[Transform Scan Data](#)**

Transforms 3D scan data

- **Average Meshes**

Creates averaged master meshes or feature shapes from 3D scan data

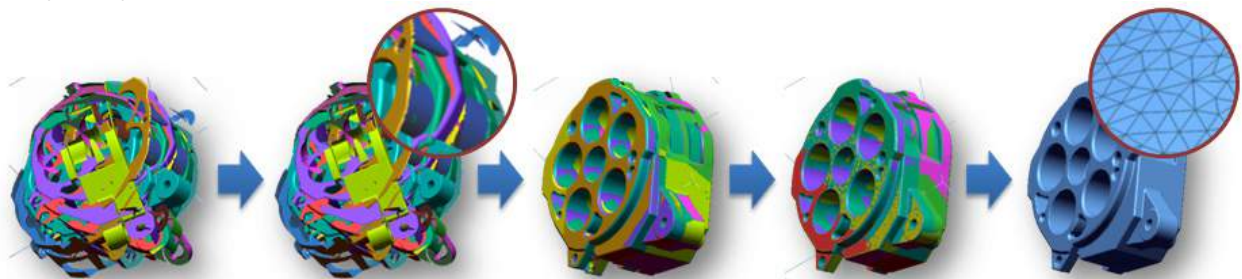
- **Boolean**

Creates new mesh by applying various geometric / mathematic operations

Tools > Scan Tools > Mesh Buildup Wizard

What

The **Mesh Buildup Wizard** command is a wizard style interface for creating defect-free and watertight mesh models from raw 3D scan data. The command consists of 3-5 stages that can be executed in just few clicks, enabling the speedy creation of optimized mesh for use in final reverse design stages.



Why

Mesh Buildup Wizard command is useful for:

- Creating a watertight mesh from raw 3D scan data
- Managing large amounts of 3D scan data and generating high quality mesh from it
- Applying proper 3D scan data processing methods to 3D scan data based on its type
- Reducing tedious and time consuming 3D scan data processing tasks

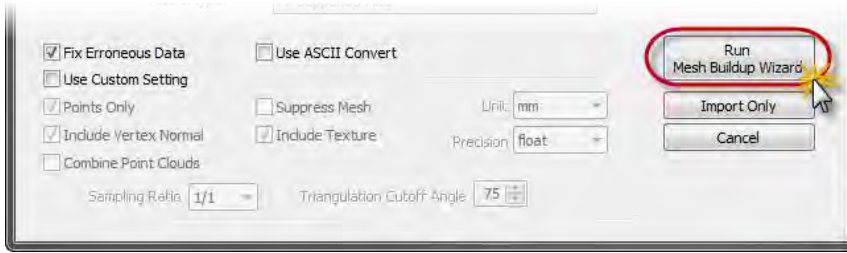
How to Use

A. How to create mesh from unaligned raw point clouds

1. Importing 3D scan data by using Mesh Buildup Wizard

Go to **Insert > Import**, or click the icon in the toolbar.

Select 3D scan data and click the **Run Mesh Buildup Wizard** button in the **Import** dialog box.



Note:

The **Mesh Buildup Wizard** can be executed after importing scan data by going to **Tools > Scan Tools > Mesh Buildup Wizard** in the menu. Multiple meshes or point clouds or a single mesh or point cloud can be used as target 3D scan data.

2. Stage 1/5 – Data Preparation: Defining data properties

Check **Small/Medium Size Object Scanner** as the target 3D scanner type.

Note:

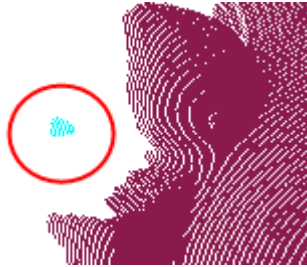
This example assumes that the target scan data is a 3D point cloud to be scanned as a **Small/Medium Size Object** scanner and not aligned.

Uncheck the “**Is Pre-Aligned?**” option and click the **Next Stage** button.



3. Stage 2/5 – Data Editing: Deleting unnecessary poly-vertices/faces

Select unnecessary clusters or outer poly-vertices.



Press the **Del** key or click the **Delete** button to remove them.

Note:

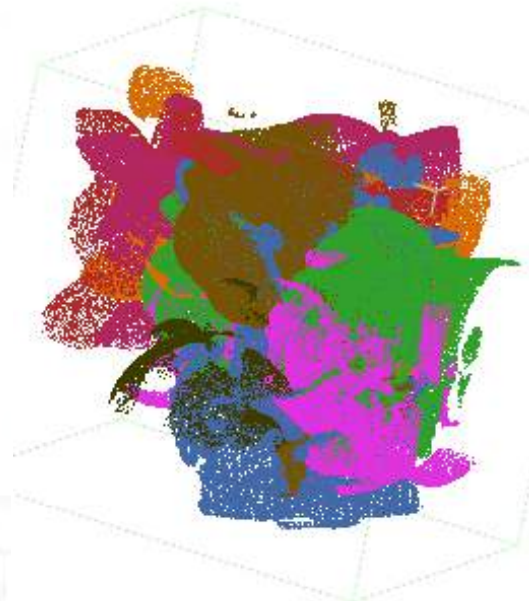
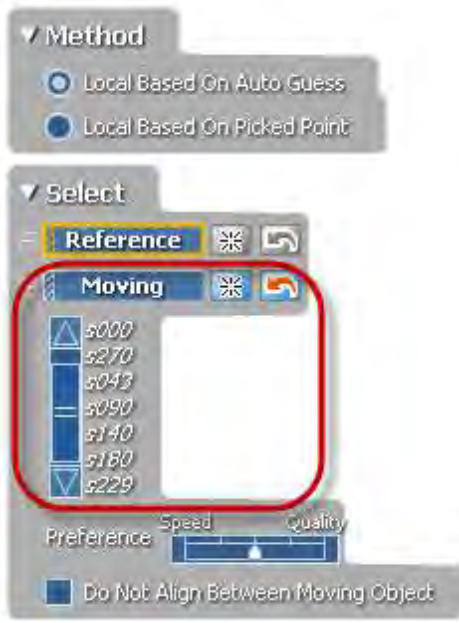
Checking specific 3D scan data or using the **Show This Only** option in the pop-up menu in the **Entity Thumbnails** pane can assist in displaying only desired data while editing 3D scan data.



Click the **Next Stage** button.

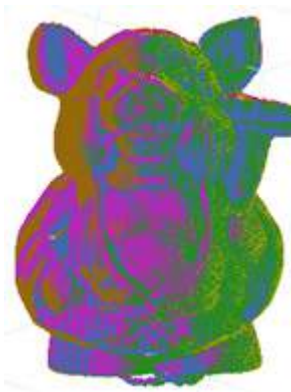
4. Stage 3/5 – Data Pre-Aligning: Aligning 3D scan data automatically or manually

Choose the **Local Based On Auto Guess** option to automatically align scans.
Check if all scans are selected as **Moving** scans.



Click the **Apply** button.

All scans will be automatically aligned.



Note:

If scans fail to align, use the **Local Based On Picked Point** option to manually align them.

Click the **Next Stage** button.

- 5. Stage 4/5 – Best-Fit Aligning:** Using the previous rough alignment, this stage performs a best-fit alignment to accurately align scans.

Check that all scans are selected for **Moving** and then click the **Apply** button.



Click the **Next Stage** button.

- 6. Stage 5/5 – Data Merging:** Merging scans and generating a single mesh

To make a watertight mesh, use the **Remesh And Fill All Holes To Create Watertight Mesh** option.

Click the **Apply** button and check the result.



7. Click **OK** to complete the command.

B. How to create a mesh from pre-aligned raw 3D point clouds

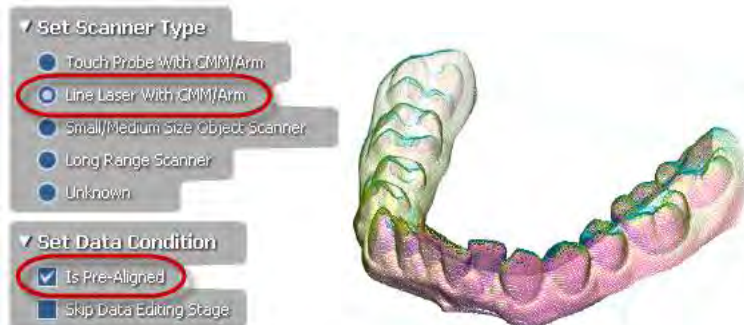
1. Run the **Mesh Buildup Wizard** command by going to **Tools > Scan Tools > Mesh Buildup Wizard** in the menu. Multiple meshes or point clouds or a single mesh or point cloud can be used as target 3D scan data in the **Mesh Buildup Wizard**.
2. **Stage 1/4 – Data Preparation:** Defining data properties

Select **Line Laser With CMM/Arm** as the target 3D scanner type.

Note:

This example assumes that the target scan data are 3D point clouds to be scanned by a **Line Laser with CMM/Arm** scanner and are already pre-aligned.

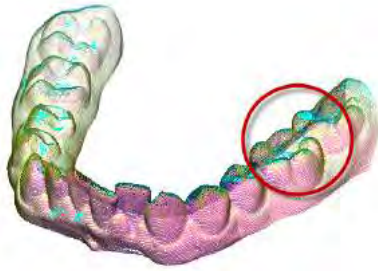
Check the **“Is Pre-Aligned?”** option and click the **Next Stage** button to continue.



3. **Stage 2/4 – Data Editing:** Deleting unnecessary poly-vertices/faces

Set the **Max. Poly-Vertices/Faces Count Per Noisy Cluster** option to **100** and click the **Find Noisy Cluster** button.

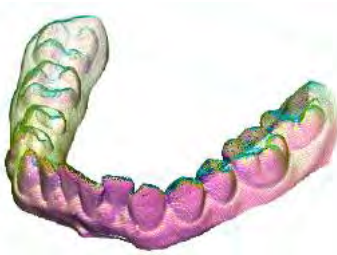
Check that noisy clusters are selected and listed under the **Select** button.



Press the **Del** key or click the **Delete** button to remove them.
Click the **Next Stage** button.

- 4. Stage 3/4 – Best-Fit Aligning:** If the 3D scan data has been pre-aligned, this stage accurately aligns scans

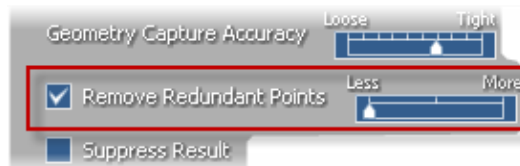
Select all scans for **Moving**.



Click the **Apply** button.
Check the results and click the **Next Stage** button.

- 5. Stage 5/5 – Data Merging:** Merging scans and generating a single mesh

Check the **Remove Redundant Points** option and reduce the ratio by adjusting the slider towards **Less**.



To make a watertight mesh, check the **Remesh And Fill All Holes To Create Watertight Mesh** option. Click the **Apply** button and check the result.

Note:

If the results are not satisfactory they can be manually edited by using the **Edit Tools**.



- 6.** Click **OK** to complete the command.

Options in Detail

Stages

This command consists of the following stages:

- **Data Preparation stage**

Analyzes the state of 3D scan data and defines the next process according to the type of target 3D scan data

- **Data Editing stage**

Removes noisy clusters and unnecessary point clouds or meshes

- **Data Pre-Aligning stage**

Quickly aligns multiple scans

- **Best-Fit Aligning stage**

Accurately aligns multiple scans by using geometry shape information

- **Data Merging stage**

Merges scans and creates an optimized mesh.

Note:

The number of stages for this command can change depend on the target source data.

Stage 1/5 – Data Preparation

This stage prepares data by selecting target 3D scan data and setting its type and alignment status. Depending on the settings selected in this stage, future stages will be automatically organized.

Select – Selects point clouds or meshes. If multiple scan data are selected as the target scan data, the alignment stage will be the following stage.

Set Scanner Type – Chooses a type of 3D scanner for target 3D scan data that can be viewed in the mouse help option. Depending on the selected type the **Mesh Buildup Wizard** will generate a watertight mesh using the most appropriate mesh generation algorithms.

Touch Probe With CMM/Arm – Sets **Touch Probe With CMM/Arm** as the scanner type for target 3D scan data.

Line Laser With CMM/Arm – Sets **Line Laser With CMM/Arm** as the scanner type for target 3D scan data.

Small/Medium Size Object Scanner – Sets **Small/Medium Size Object Scanner** as the scanner type for target 3D scan data.

Long Range Scanner – Sets **Long Range Scanner** as the scanner type for target 3D scan data that is scanned by long-range 3D laser scanners.

Unknown – Sets a generic scanner type for target 3D scan data that cannot be defined by a specific scanner type or has been mixed with different scan types.

Set Data Condition – Sets the current status of 3D scan data.

Is Pre-Aligned? – Indicates target 3D scan data has been pre-aligned.

Skip Data Editing Stage? – Skips the data editing stages.

Stage 2/5 – Data Editing

This stage removes unnecessary poly-vertices or faces manually or automatically. After selecting unwanted data use the **Del** key or click the **Delete** button to remove them.

Remove Unwanted Data – Removes unwanted poly-vertices or faces in 3D scan data using various editing tools.

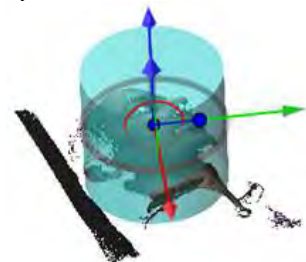
Select – Selects unwanted poly-vertices/faces manually.



Define Wanted Region By Box – Defines a working volume by resizing a box. After defining a working volume, click the **Delete** button to remove outer poly-vertices or faces from the box.



Define Wanted Region By Scanner Range – Defines a working volume by resizing a cylinder. The radius of the cylinder can be manipulated by a gray circle. After defining the working volume, click the **Delete** button to remove outer poly-vertices or faces from the cylinder.



Find Floor – Selects a seed point on a scan data used to expand and select flat areas. After finding the flat areas, click the **Delete** button to remove the selected poly-vertices or faces.



Find All Boundary – Selects all boundaries and is available for mesh data only. After finding boundaries, click the **Delete** button to remove poly-faces neighboring the boundaries.



Find Noisy Cluster – Finds small groups of poly-vertices or faces that are unnecessary in constructing a mesh by specify a cluster size next to the **Max. Poly-Vertices/Faces Count Per Noisy Cluster** options. After finding noisy clusters, click the **Delete** button to remove the selected poly-vertices or faces.



Stage 3/5 – Data Pre-Aligning

This stage aligns scans manually or automatically. The **Local Based On Auto Guess** option intelligently compares features on each 3D scan data file and aligns them. If scans fail in auto-guess aligning, they can be manually aligned by using the **Local Based On Picked Point** option. For information about aligning between scan data, see [Tools > Scan Tools > Align Between Scan Data](#).

Note:

This stage is accessed when the **Is Pre-Aligned?** A option has been unchecked in the first stage (Data Preparation). If scans have already been pre-aligned, skip this stage by clicking the **Next Stage** button.

Stage 4/5 – Best-Fit Aligning

If scans are roughly aligned or the **Data Pre-Aligning** stage is used, it will be necessary for scans to be aligned accurately. This stage uses the best-fit alignment algorithm to match scans closely and reduce deviation.

Select – Selects point clouds or meshes.

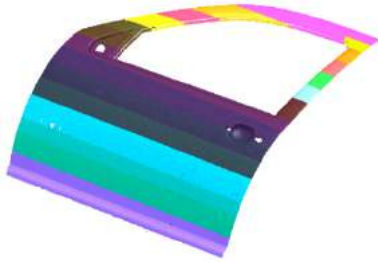
Reference – Selects scans to set for **Reference**. Their position will be fixed and other **Moving** scans will be transformed. A reference scan is not necessary.

Moving – Selects scans to set for **Moving**. The moving scans will be transformed to the **Reference**.

Options – Sets alignment the options.

Sampling Ratio – Samples data points by a specified value. When the ratio is 100%, all selected data will be used. When the ratio is 50%, half of the selected data will be used.

Apply Only Small Transform – Applies only a small transform when overlapping areas are assumed to be well aligned or if an alignment is difficult due to lack of feature shapes that can be used.



Lack of Feature Shapes



Patterned Feature Shapes

Do Not Align Between Moving Object – Applies alignment by scan groups.



Reference Group



Moving Group – aligned scans

Update View & Histogram – Displays the alignment process and the updating histogram in the Model View in real-time.

Edit Tools – Removes unwanted poly-vertices/faces in 3D scan data by using various editing tools and corrects the normal information of 3D scan data.

Note:

This command can be used in this stage as well as in the Data Pre-Aligning stage (3/5 stage) and in the Data Merging stage (5/5 stage).

Remove Unwanted Data – Selects unwanted poly-vertices or faces that can be deleted by pressing the **Del** key or clicking the **Delete** button.

Note:

Unwanted scan data can be selected manually or automatically by selecting or using the **Detect Floor** and **All Boundaries** buttons.

Reverse Normal Of Entire Mesh/Point Cloud – Reverses the normal information of an entire target 3D scan data.

Reverse Normal Of Cluster – Reverses the normal information of a poly-face cluster.

Fix Normal Of Mesh – Corrects the normal information of a target mesh.

Stage 5/5 – Data Merging

This stage merges 3D scan data and generates them into a single mesh.

Methods

This command features 2 methods:

- **Mesh Construction**

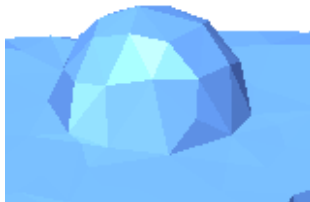
Generates a polygonal mesh by following the geometric shape of a point cloud

- **New Mesh Construction**

Generates a polygonal mesh by using voxel data structures

Mesh Construction – Generates poly-faces by following the geometric shape of a point cloud. This method is useful for generating polygonal mesh from a point cloud that has multiple combined directional scans that cannot be projected onto a virtual plane or sphere or when results are required to be constructed by poly-faces with uniform poly-edge lengths.

Geometry Capture Accuracy – Defines geometry capture accuracy. When the slider is moved towards **Loose**, less poly-faces will be used to create a shape. When the slider is moved towards **Tight**, more poly-faces will be used to create a shape.



Loose



Tight

Scanner Accuracy – Specifies the accuracy of a scanner. Specify a value by referring to the specification of a particular scanner. For example, if target 3D scan data is scanned by a small/medium object scanner, the default value of 0.05 mm is suitable. If target 3D scan data is scanned by a long-range laser scanner, increase the value based on the precision of the scanner.

Note:

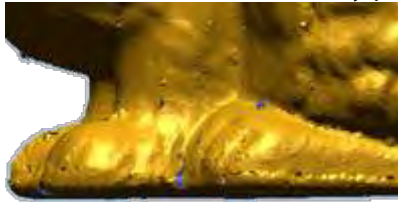
In general, if scan data is clean with a low level of noise, has a regular density, and also has satisfactory align accuracy, then using the maximum value of the 'Scanner Accuracy' slider will preserve and use most points just as they are.

Each decreased step in the slider will decrease the creation of meshes by approximately a half of the previous step. The result will not be a uniform reduction and high curvature regions will be preserved, which is similar to applying a 50% decimation.

For example, the following combinations create almost the same results:

- 'Scanner Accuracy' is 0.05mm and 'Geometry Capture Accuracy' is at the 4th step
- 'Scanner Accuracy' is 0.0333mm (0.05 / 1.5) and 'Geometry Capture Accuracy' is at the 3th step
- 'Scanner Accuracy' is 0.075mm (0.05 x 1.5) and 'Geometry Capture Accuracy' is at the 5th step

Reduce Noise – Reduces noisy poly-faces.



Without Reduce Noise option



With Reduce Noise option

Remesh And Fill All Holes To Create Watertight Mesh – Generates a watertight mesh that has no holes and or defects.



Non-watertight Mesh



Watertight Mesh

New Mesh Construction – Generates poly-faces by using voxel data structures. This method is useful for generating polygonal mesh from a point cloud that does not have many open boundaries. This method is also quicker and can produce better results than the **Mesh Construction** method.

Note:

To get satisfactory results from the New Mesh Construction method, scan data must have normal information and using a mesh that has few open boundaries is ideal because the algorithm extends boundaries beyond their initial settings. The New Mesh Construction method keeps normal information consistent on connected triangles and can be useful for long range scan data and scan data with no boundaries.

Global Shape Fitting Resolution – Controls how many voxel structures are used to create a resulting mesh. The higher the voxel structure count, the more control points can be used to represent the original shape of the scan data.

Shape Fitting To Source Data – Defines geometry capture accuracy. When the slider is moved towards **Loose**, less poly-faces will be used to create a shape. When the slider is moved towards **Tight**, more poly-faces will be used to create a shape.

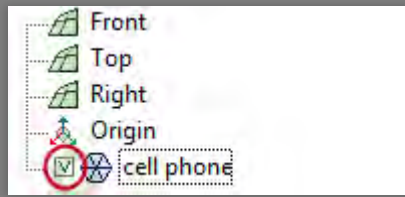
Suppress Result In Feature Tree – Suppresses results from the **Feature Tree**.

Note:

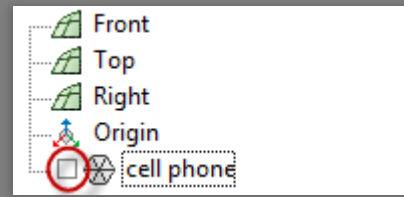
The Suppress and Unsuppress mechanisms manage data memory by swapping data

dynamically between physical memory and a hard disk. This function is useful for effectively managing and handling a large 3D scan data file.

Created Features can be suppressed or unsuppressed by checking the box next to the feature name in the Feature Tree.



Unsuppressed



Suppressed

Noise Reduction Sampling – Reduces noise using a sampling algorithm.

Create Watertight Mesh – Closes boundaries using the curvature of scan data.

Curved – Creates a closed curved boundary at open boundaries.

Flat – Creates a closed flat boundary at open boundaries.

See Also

[Tools > Scan Tools > Triangulate](#)

[Tools > Scan Tools > Align Between Scan Data](#)

[Tools > Scan Tools > Merge](#)

[Tools > Mesh Tools > Fix Normal](#)

[Tools > Mesh Tools > Global Remesh](#)

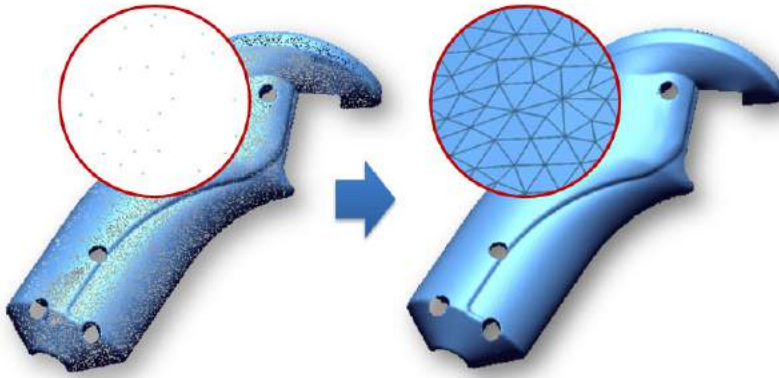
[Tools > Mesh Tools > Fill Holes](#)

Tools > Scan Tools > Triangulate

What

The **Triangulate** command creates poly-faces by connecting points within 3D scan data to construct mesh. A target can be entire point clouds or individually selected poly-vertices in a point cloud.

Point clouds scanned and produced by 3D scanners can be used directly for measuring and visualizing. Using only a point cloud can be difficult to represent a shape and lacks information for further design stages. Triangulation is a typical process used for creating geometric shape information from a raw point cloud.



Why

The **Triangulate** command is useful for:

- Converting point cloud to 3D mesh models with geometric shape information
- Creating basic geometric shape information used for creating CAD model
- Creating basic geometric shape information used for evaluating the shape of a model
- Creating basic geometric shape information used for modifying an original feature shape

How to Use

A. How to create a mesh model from point cloud which side-wall shapes exist

1. Go to **Tools > Scan Tools > Triangulate** or click the icon in the toolbar.
2. Click the **Point Cloud** button and select a point cloud.



Note:

This example assumes that a target point cloud is a single point cloud combined by multiple scans that have been scanned in different scanning directions and that it cannot be projected onto any virtual plane.

3. Choose the **3D Triangulate** method. The **Estimate** button automatically calculates the distance between points and displays them in the **Distance Criteria To Connect Vertices** box.
4. Click **OK** to complete the command. Check the results.



B. How to create a mesh model from point clouds which side-wall shapes doesn't exist

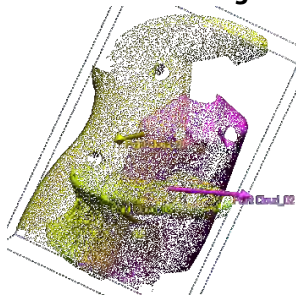
1. Go to **Tools > Scan Tools > Triangulate** or click the icon in the toolbar.
2. Click the **Point Cloud** button and select a point cloud.



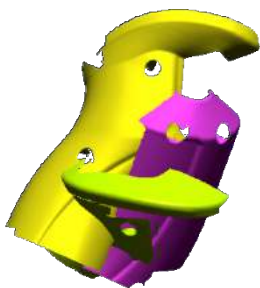
Note:

This example assumes that target point clouds are multiple scans that have been scanned in different scanning directions and that they can be projected onto a virtual plane.

3. Choose the **2D Triangulate** method.



4. Click **OK** to complete the command. Check the results.



Options in Detail

Target Entities

Point Clouds – Selects point clouds or poly-vertices.

Note:

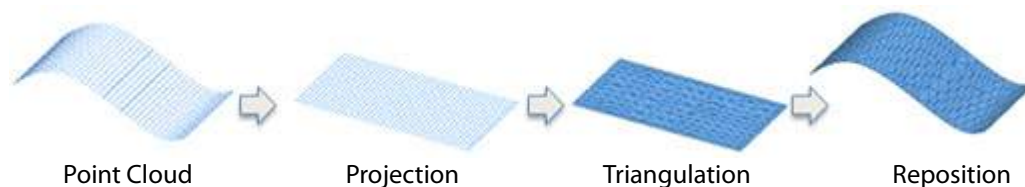
This option allows the selection of not only multiple point clouds, but also poly-vertices. Selected point clouds and poly-vertices can be used to create a polygonal mesh on partial areas of point clouds.

Methods

This command features 4 methods:

- **2D Triangulate**
Generates a polygonal mesh by projecting a point cloud onto a virtual plane or sphere
- **3D Triangulate**
Generates a polygonal mesh by connecting points in a point cloud in 3D space
- **Mesh Construction**
Generates a polygonal mesh by following the geometric shape of a point cloud
- **New Mesh Construction**
Generates a polygonal mesh by using voxel data structures

2D Triangulate – Generates a polygonal mesh from a point cloud by using projection and repositioning. Poly-vertices in three dimensional space are projected onto a plane or sphere specified by a user and triangulated to projected vertices which are repositioned to their original position.



Planar – Projects poly-vertices onto a virtual plane and generates poly-faces. This method can be used when 3D scan data is scanned in a single direction.



Scan Direction – Projects in a scanning direction shown as an arrow on a point cloud in the Model View. Reverse the projection direction by clicking an arrow on

the model. If point clouds do not have normal information, click the **Find Scan Direction** button.

Current Viewing Direction – Projects in the current viewing direction in the Model View.

Spherical – Projects poly-vertices onto a virtual sphere from the center point of 3D scan data and generates poly-faces. This method is used to triangulate sphere-like 3D scan data files obtained by specific scanner types like long-range laser scanners.



Input Center Manually – Specifies a center position manually.

Note:

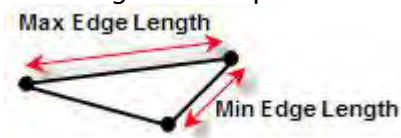
The center point of 3D scan data follows the origin in the world coordinate system (X: 0, Y: 0, Z: 0) by default. In case where the 3D scan data was transformed before triangulation, input new X, Y, Z coordinate values according to the transformation information.

Face Removal Criteria – Filters irregular poly-faces while executing 2D triangulation.

Maximum Edge Length Is Larger Than – Removes poly-faces that have a longest edge length longer than a specified value.

Area Is Larger Than – Removes poly-faces that have an area larger than a specified value.

Max/Min Edge Length Ratio Is Larger Than – Removes poly-faces that have a maximum and minimum length ratio between its longest edge and shortest edge that is larger than a specified value.



Angle Between Projection Direction And Face Normal Direction Is Larger Than – Removes poly-faces that have an angle between a projection direction and a poly-face normal that is larger than a specified criteria.

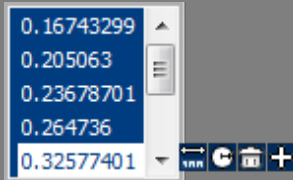
3D Triangulate – Generates a polygonal mesh from a point cloud by connecting poly-vertices. This method starts generating small poly-faces from closest poly-vertices and propagates triangulation to adjacent poly-vertices. This method is useful for generating polygonal mesh from a point cloud that has multiple scans combined from different directions that cannot be projected onto a virtual plane or sphere.



Distance Criteria To Convert Vertices – Specifies distances between poly-vertices to connect them and generate poly-faces.

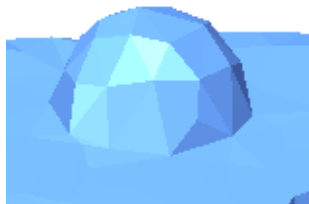
Note:

The **Estimate** button estimates distances between poly-vertices and displays the results in a criteria list box. The results are representative values of average distance of a point cloud. To edit values, double-click it. If another value needed, click the **Add** button.



Mesh Construction – Generates poly-faces by following the geometric shape of a point cloud. This method is useful for generating polygonal mesh from a point cloud that has multiple combined directional scans that cannot be projected onto a virtual plane or sphere or when results are required to be constructed by poly-faces with uniform poly-edge lengths.

Geometry Capture Accuracy – Defines geometry capture accuracy. When the slider is moved towards **Loose**, less poly-faces will be used to create a shape. When the slider is moved towards **Tight**, more poly-faces will be used to create a shape.



Loose



Tight

Scanner Accuracy – Specifies the accuracy of a scanner. Specify a value by referring to the specification of a particular scanner. For example, if target 3D scan data is scanned by a small/medium object scanner, the default value of 0.05 mm is suitable. If target 3D scan data is scanned by a long-range laser scanner, increase the value based on the precision of the scanner.

Note:

In general, if scan data is clean with a low level of noise, has a regular density, and also has satisfactory align accuracy, then using the maximum value of the 'Scanner Accuracy' slider will preserve and use most points just as they are.

Each decreased step in the slider will decrease the creation of meshes by approximately a half of the previous step. The result will not be a uniform reduction and high curvature regions will be preserved, which is similar to applying a 50% decimation.

For example, the following combinations create almost the same results:

- 'Scanner Accuracy' is 0.05mm and 'Geometry Capture Accuracy' is at the 4th step
- 'Scanner Accuracy' is 0.0333mm (0.05 / 1.5) and 'Geometry Capture Accuracy' is at the 3th step
- 'Scanner Accuracy' is 0.075mm (0.05 x 1.5) and 'Geometry Capture Accuracy' is at the 5th step

New Mesh Construction – Generates poly-faces by using voxel data structures. This method is useful for generating polygonal mesh from a point cloud that does not have many open boundaries. This method is also quicker and can produce better results than the **Mesh Construction** method.

Note:

To get satisfactory results from the New Mesh Construction method, scan data must have normal information and using a mesh that has few open boundaries is ideal because the algorithm extends boundaries beyond their initial settings. The New Mesh Construction method keeps normal information consistent on connected triangles and can be useful for long range scan data and scan data with no boundaries.

Global Shape Fitting Resolution – Controls how many voxel structures are used to create a resulting mesh. The higher the voxel structure count, the more control points can be used to represent the original shape of the scan data.

Shape Fitting To Source Data – Defines geometry capture accuracy. When the slider is moved towards **Loose**, less poly-faces will be used to create a shape. When the slider is moved towards **Tight**, more poly-faces will be used to create a shape.

More Options

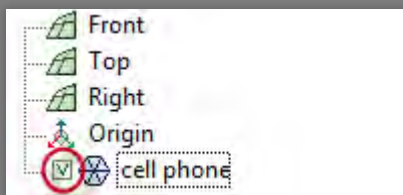
Remove Original Data – Removes original point cloud after triangulation.

Suppress Result In Feature Tree – Suppresses results from the **Feature Tree**.

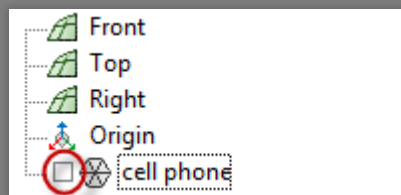
Note:

The Suppress and Unsuppress mechanisms manage data memory by swapping data dynamically between physical memory and a hard disk. This function is useful for effectively managing and handling a large 3D scan data file.

Created Features can be suppressed or unsuppressed by checking the box next to the feature name in the Feature Tree.



Unsuppressed

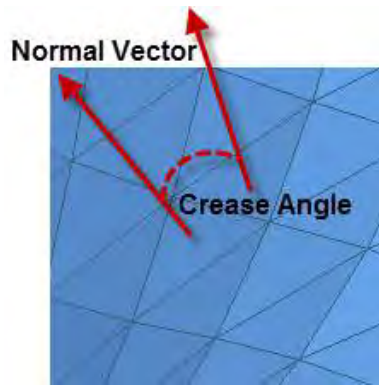


Suppressed

Sampling Ratio – Samples data points by a specified value. When the ratio is 100%, all selected data will be used. When the ratio is 50%, half of the selected data will be used.

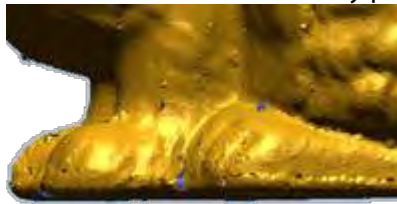
Assume Vertices Are Densely Placed Along A Direction – Samples redundant poly-vertices closed by a certain distance for better triangulation results. Check this option when target 3D scan data is scanned by a line laser with a CMM/Arm device.

Crease Angle – Specifies a maximum angle between normal vectors of adjacent poly-faces to avoid generating abnormal poly-faces.



Angle between Normal Vectors of Adjacent Poly-Faces

Reduce Noise – Reduces noisy poly-faces.



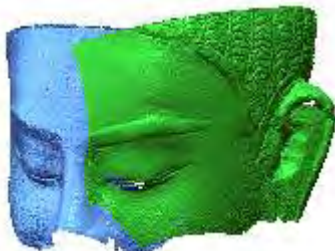
Without Reduce Noise option



With Reduce Noise option

Remove Redundant Points – Removes redundant poly-vertices closed by a certain distance for better triangulation results.

Make Single Mesh – Merges selected point clouds into a single mesh after triangulation and is only available if two or more point clouds exist.



Without Make Single Mesh



With Make Single Mesh

Noise Reduction Sampling – Reduces noise using a sampling algorithm.

Create Watertight Mesh – Closes boundaries using the curvature of scan data.

Curved – Creates a closed curved boundary at open boundaries.

Flat – Creates a closed flat boundary at open boundaries.

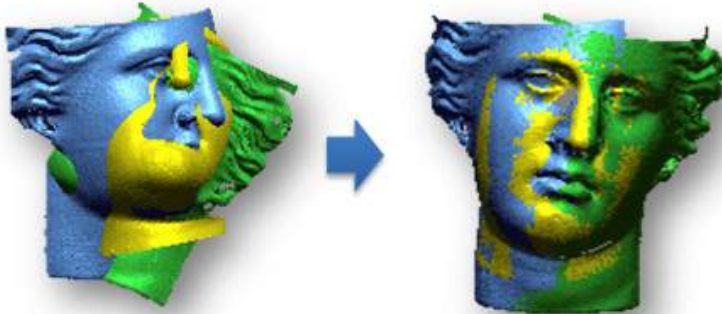
See Also

[Tools > Scan Tools > Mesh Buildup Wizard](#)

Tools > Scan Tools > Align Between Scan Data

What

The **Align Between Scan Data** command aligns two or more 3D scan data sets scanned in different scanning directions based on geometric feature shape information.



Why

The **Align Between Scan Data** command is useful for:

- Completing a model from multiple 3D scan data files scanned in different scanning directions
- Effectively managing a large amount of 3D scan data and re-arranging them

How to Use

A. How to align multiple 3D scan data files automatically

1. Go to **Tools > Scan Tools > Align Between Scan Data** or click the icon in the toolbar.

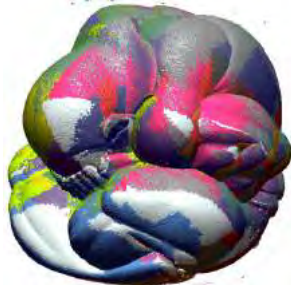
Note:

To apply this command to 3D scan data, the target 3D scan data should not have child entities.

2. Check the **Local Based On Auto Guess** option, and then select all scans for **Moving**.



3. Click the **Apply** button and check the results.



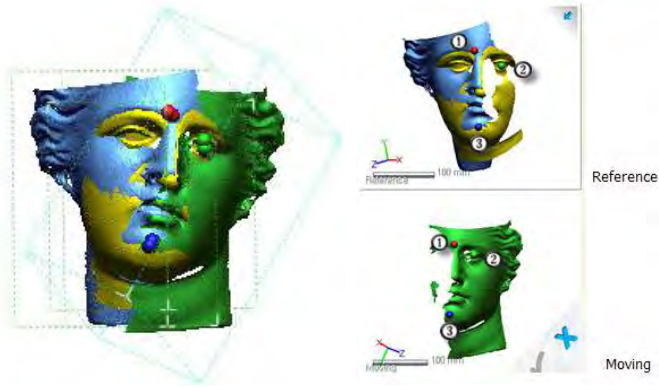
B. How to align multiple 3D scan data manually

1. If scans fail during auto guessing, manually align them using the **Local Based On Picked Point** method.

Check the **Local Based On Picked Point** option and select aligned scans for **Reference** and select failed scan shots for **Moving**.



2. Manually pick corresponding points on 3D scan data in the **Reference** and **Moving** window. To cancel the last picking point, go to **Edit > Undo** or use the **Ctrl + Z** keyboard shortcut.



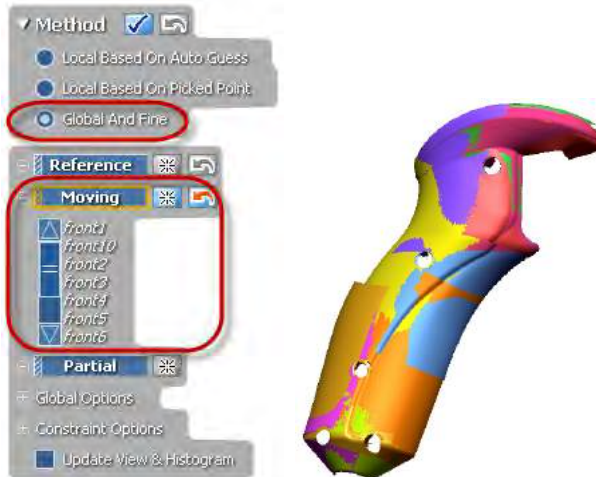
3. Click the **Apply** button to apply manual alignment.



C. How to align multiple 3D scan data files accurately

1. Accurately align roughly aligned scans by using the **Local Based On Auto Guess** or **Local Based On Picked Point** methods.

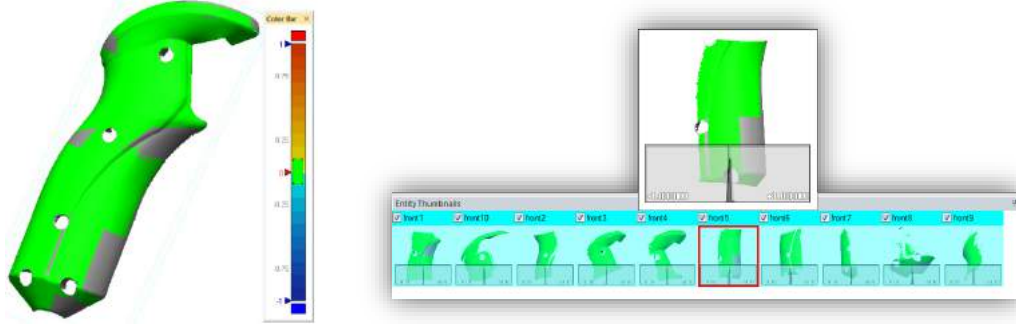
Check the **Global And Fine** option and select all scans for **Moving**.



2. Click the **Apply** button.



3. Click **Avg. Deviation** in the **Accuracy Analyzer™** panel to confirm deviation among meshes. Deviation results and a histogram can be displayed by selecting entities in the **Entity Thumbnail** pane.



4. Click **OK** to complete the command.

Options in Detail

Methods

This command features 3 methods:

- **Local Based On Auto Guess**
Automatically aligns scans based on geometric feature shape information
 - **Local Based On Picked Point**
Manually aligns scans by using corresponding points
 - **Global And Fine**
Aligns scans based on geometric feature shape information with the lowest amount of deviation possible in overlapped areas
-

Local Based On Auto Guess – Automatically aligns scans.

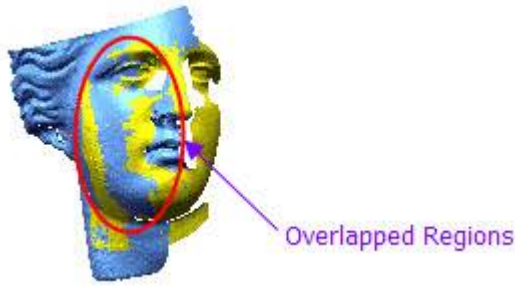
Reference – Selects scans to define as reference entities.

Moving – Selects scans to define as moving entities. One or more scans should be registered for **Moving** to execute the command.

More Options


Preference – Adjusts operation preferences. To quickly align scans, move the slider towards **Speed**, and if it fails, move the slider towards **Quality**.

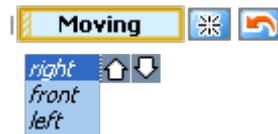
If Features Are Sufficient In Overlapped Regions – Used when scans have sufficient features in overlapped areas. If overlapped areas are flat or smooth without features, do not use this option.



Alignment Order – When the target scans are scanned in a certain scanning order, use this option for more accurate alignment.

By Name aligns by naming order.

By Selection aligns by selection order. The selection order can be changed in the list box under **Moving** using the **Up** and **Down** buttons .



Do Not Align Between Moving Object – Applies alignment by scan groups.



Reference Group



Moving Group – aligned scans

Local Based On Picked Point – Manually aligns scans. Select scans for **Reference** and for **Moving**, and pick corresponding points in the **Reference** and **Moving** windows.

The corresponding points are required to be defined for Reference and Moving entities to apply this method. This method can be used when scans fail in the **Local Based On Auto Guess** method or when scans are needed to be aligned manually.



Refine Alignment – Applies best-fit alignment to areas around defined corresponding points after the **Local Based On Picked Point** method is applied.

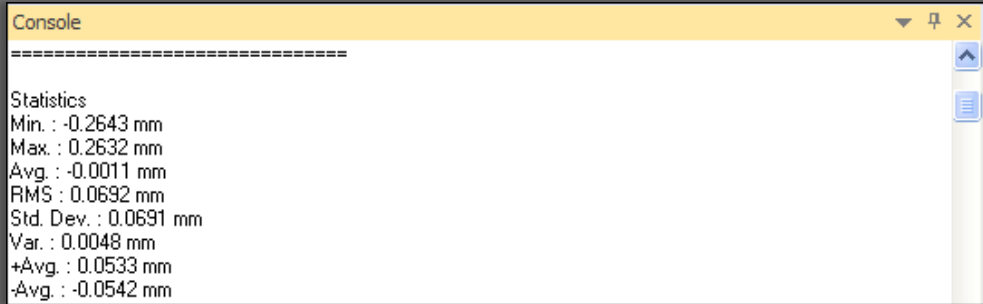
Global And Fine – Aligns scans based on geometric feature shape information with the lowest amount of deviation possible in overlapping areas. This method can be used after the **Local Based On Auto Guess** or **Local Based On Picked Point** methods are applied to scans for more accurate alignment.

Partial – Selects scans to define as partial entities.

Note:

Poly-vertices selected from at least two different 3D scan data files are need as **Partial** to apply this command. The partially selected poly-vertices are used to calculating the fine alignment.

Statistical results such as average, standard deviation, RMS for **Global And Fine** alignment are listed in the **Console** panel after alignment.



```
Console
-----
Statistics
Min. : -0.2643 mm
Max. : 0.2632 mm
Avg. : -0.0011 mm
RMS : 0.0692 mm
Std. Dev. : 0.0691 mm
Var. : 0.0048 mm
+Avg. : 0.0533 mm
-Avg. : -0.0542 mm
```

Global Options

Sampling Ratio – Samples data points by a specified value. When the ratio is 100%, all selected data will be used. When the ratio is 50%, half of the selected data will be used.

Max. Iteration Count – Specifies a maximum iteration value.

Max. Average Deviation – Specifies a maximum average deviation.

Note:

The **Max.Iteration Count** and **Max.Average Deviation** options interactively affect each other. For example, if both options are applied to the alignment, it performs until the defined maximum average deviation is satisfied within the maximum iteration count.

Apply Only Small Transform – Applies small transform only if overlapped areas are assumed to be well aligned or if alignment is difficult to be performed due to lack of feature shapes that can be referred to on the overlapped areas.

Use Only Reliable Scan Data – Uses only reliable 3D scan data or only effective overlapped scan areas for better alignment.

Constraint Options – Fixes the translation or rotation of the X, Y, or Z axes for **Moving** entities.

Update View & Histogram – Used to view the alignment process in the Model View and to update the histogram in real-time.

See Also

[Tools > Scan Tools > Mesh Buildup Wizard](#)
[Tools > Scan Tools > Transform Scan Data](#)

Tools > Scan Tools > Merge

What

The **Merge** command merges multiple point clouds or meshes and creates a single mesh. During the merging process, overlapped areas will be removed efficiently and neighboring boundaries are stitched together.



Why

The **Merge** command is useful for:

- Merging multiple 3D scan data into a single mesh

How to Use

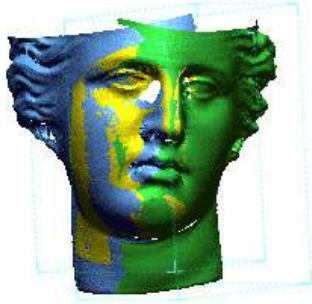
How to merge multiple 3D scan data into a single mesh

1. Go to **Tools > Scan Tools > Merge** or click the icon in the toolbar.

Note:

To apply this command to 3D scan data, a target 3D scan data should not have child entities.

2. Click the **Mesh / Point Cloud** button and select entities to merge them.



3. Choose **Surface Merge** and click the **OK** to complete the command.



Note:

Deviation between original data and a merged mesh can be viewed in the **Accuracy Analyzer**[™] panel.

Options in Detail

Target Entities

Mesh / Point Cloud – Selects meshes or point clouds.

Methods

This command features 3 methods:

- **Surface Merge**

Merges scans by their boundaries by removing overlapping areas

- **Volume Merge**

Generates polygonal mesh by reconstructing geometries scans with 3D volume

- **Mesh Construction**

Generates a polygonal mesh by following the geometric shape of a target 3D scan data

Surface Merge – Detects overlapping areas in target scans and merges the scans by their boundaries by removing the overlapping areas. This method can be applied to meshes only. It is useful for preserving the original position of poly-faces on a target 3D scan data and merging them by deleting only the overlapping areas.



Geometry Capture Accuracy – Defines geometry capture accuracy. When the slider is moved towards **Loose**, less poly-faces will be used to create a shape. When the slider is moved towards **Tight**, more poly-faces will be used to create a shape.



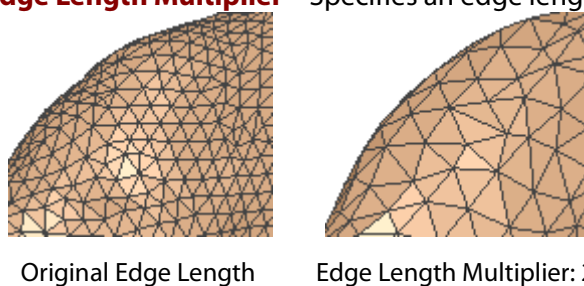
Estimated Num. Of Faces – Estimates the total number of poly-faces after a merging is completed.

Volume Merge – Divides the 3D volume of target scans by a specified resolution and generates a polygonal mesh by reconstructing geometries of the scans within the volume. This method can be applied to meshes and point clouds that have normal information. It is useful for generating a smooth polygonal mesh when 3D scan data has complex boundaries or has been badly aligned.



Current Average Edge Length – Shows the current average edge length of poly-edges from selected meshes.

Edge Length Multiplier – Specifies an edge length multiplier.



Target Average Edge Length – Defines the average length of poly-edges after merging using a specified value. The size of poly-faces of results will be determined according to a defined target average edge length. The average edge length of poly-faces can be control by either the **Edge Length Multiplier** or the **Target Average Edge Length** options.



Mesh Construction – Combines target scans and reconstructs a polygonal mesh by following the geometric shape of the target 3D scan data. This method can be applied to both point clouds and meshes. It executes the **Triangulate** and **Merge** methods simultaneously by following the geometric shape of scan data.

Scanner Accuracy – Specifies the accuracy of a scanner. Specify a value by referring to the specification of a particular scanner. For example, if target 3D scan data is scanned by a small/medium object scanner, the default value of 0.05 mm is suitable. If target 3D scan data is scanned by a long-range laser scanner, increase the value based on the precision of the scanner.

Note:

In general, if scan data is clean with a low level of noise, has a regular density, and also has satisfactory align accuracy, then using the maximum value of the 'Geometry Capture Accuracy' slider will preserve and use most points just as they are.

Each decreased step in the slider will decrease the creation of meshes by approximately a half of the previous step. The result will not be a uniform reduction and high curvature regions will be preserved, which is similar to applying a 50% decimation.

For example, the following combinations create almost the same results:

- 'Scanner Accuracy' is 0.05mm and 'Geometry Capture Accuracy' is at the 4th step
- 'Scanner Accuracy' is 0.0333mm (0.05 / 1.5) and 'Geometry Capture Accuracy' is at the 3th step
- 'Scanner Accuracy' is 0.075mm (0.05 x 1.5) and 'Geometry Capture Accuracy' is at the 5th step

More Options

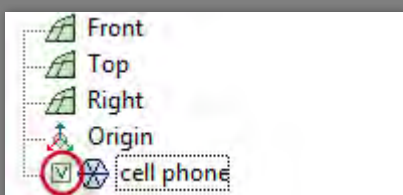
Remove Original Data – Removes original 3D scan data after merging.

Suppress Result – Suppresses results from the **Feature Tree**.

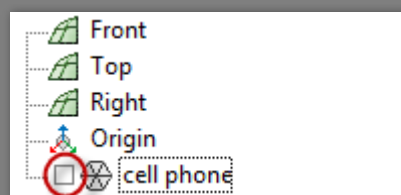
Note:

The Suppress and Unsuppress mechanisms manage data memory by swapping data dynamically between physical memory and a hard disk. This function is useful for effectively managing and handling a large 3D scan data file.

Created Features can be suppressed or unsuppressed by checking the box next to the feature name in the Feature Tree.



Unsuppressed



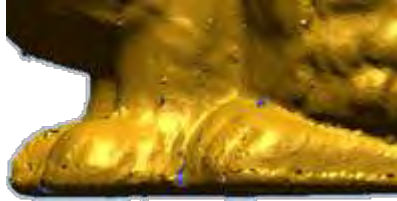
Suppressed

Scale Of Overlap Region Deviation Criteria – Sets the scale deviation criteria for overlapping areas. When target 3D scan data has a thinner thickness than its average edge length, scale the deviation criteria down to avoid merging poly-faces between the front and back sides.

Large Data Mode – Internally divides target 3D scan data into sub models and manages large data files by unloading and loading between the physical memory and the hard drive during calculation.

Remove Sharp Poly-Faces – Removes sharp poly-faces that may be generated during the merging.

Reduce Noise – Reduces noisy poly-faces. If you use clean mesh that have no noise. Uncheck this option to avoid noisy results.



Without Reduce Noise option



With Reduce Noise option

Remove Redundant Points – Removes redundant poly-vertices closed by a certain distance for better triangulation results.

Create More Dense Mesh Around Boundary – Creates dense poly-faces around the boundary when constructing a mesh.

See Also

[Tools > Scan Tools > Mesh Buildup Wizard](#)

[Tools > Scan Tools > Combine](#)

Tools > Scan Tools > Combine

What

The **Combine** command combines selected point clouds or meshes as a single point cloud or a mesh without reconstructing entities.



Why

The **Combine** command is useful for:

- Combining multiple point clouds into a single point cloud
- Combining multiple meshes into a single mesh

How to Use

How to combine multiple meshes into a single mesh

1. Go to **Tools > Scan Tools > Combine** or click the icon in the toolbar.

Note:

To apply this command to 3D scan data, the target 3D scan data should not have child entities.

2. Select meshes.



3. Click **OK** to complete the command.



Options in Detail

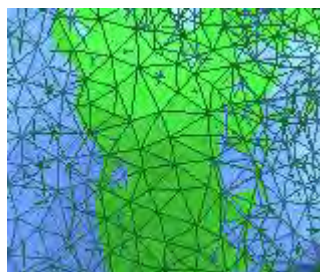
Type To Combine

Choose either a Point Cloud or Mesh as an entity type.

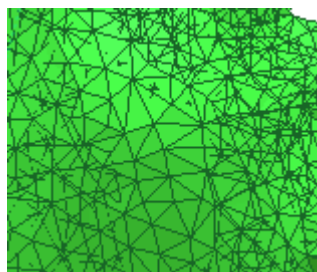
Target Entities

Selects point clouds or meshes.

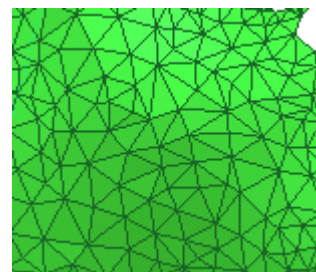
Remove Overlap Region – Removes overlapped areas.



Before Combining



Combine Without Remove Overlapped Areas



Combine With Remove Overlapped Areas

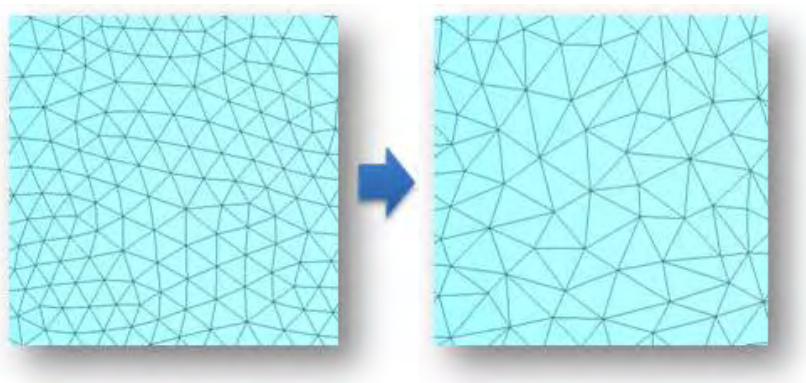
See Also

[Tools > Scan Tools > Mesh Buildup Wizard](#)
[Tools > Scan Tools > Merge](#)

Tools > Scan Tools > Decimate Meshes

What

The **Decimate Meshes** command reduces the number of poly-faces by merging poly-vertices while preserving geometric feature shapes. This command can be applied to multiple 3D scan data at one time.



Why

The **Decimate Meshes** command is useful for:

- Reducing the number of poly-faces
- Increasing the size of poly-faces from meshes
- Dealing with a large number of meshes in follow-up modeling processes while maintaining an original shape

How to Use

How to reduce the number of poly-faces of a mesh

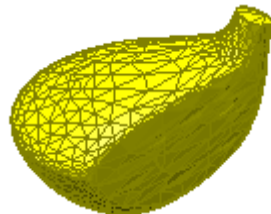
1. Go to **Tools > Scan Tools > Decimate Meshes** or click the icon in the toolbar.



Note:

To apply this command to meshes, the target meshes should not have child entities.

2. Check the **Poly-Face Count** option and set the **Reduction Ratio** to **70 %**.
3. Click **OK** to complete the command.



Options in Detail

Target Entities

Target Mesh – Selects target meshes.

Methods

This command features 2 methods:

- **Poly-Face Count**
Reduces the number of poly-faces by controlling the target poly-face count
 - **Allowable Deviation**
Reduces the number of poly-faces within an allowable deviation
-

Poly-Face Count – Reduces the number of poly-faces by using parameters specified by the **Target Poly-Face Count** or **Reduction Ratio** options.

Target Poly-Face Count – Determines the target number of poly-faces. This option can be used only when a single mesh is selected.

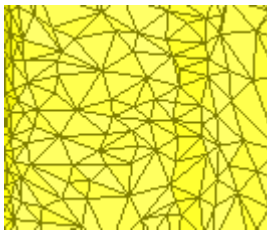


Original Face Count: 14,703

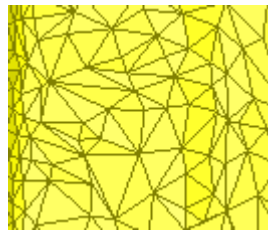


Target Face Count: 5,228

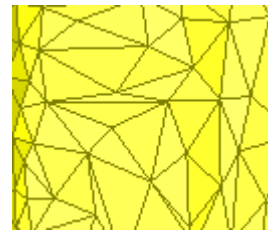
Reduction Ratio – Determines reduction ratio.



Original Data

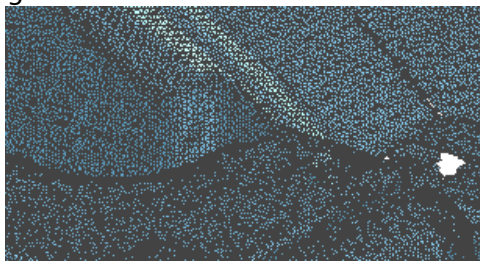


Reduction Ratio: 70%

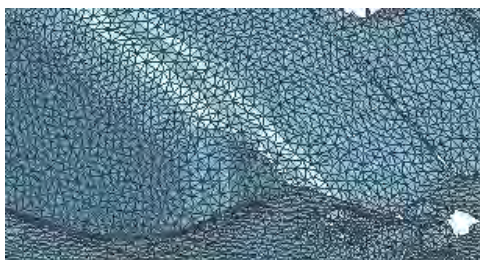


Reduction Ratio: 30%

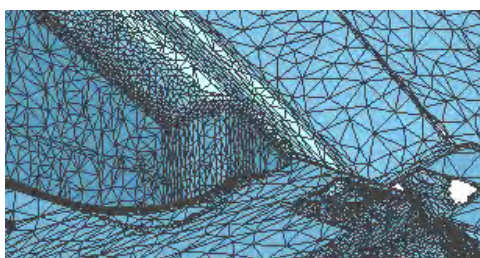
High Curvature Area Resolution – Determines resolution in high curvature areas of a target mesh.



Original Data



High Curvature Area Resolution: Sparse

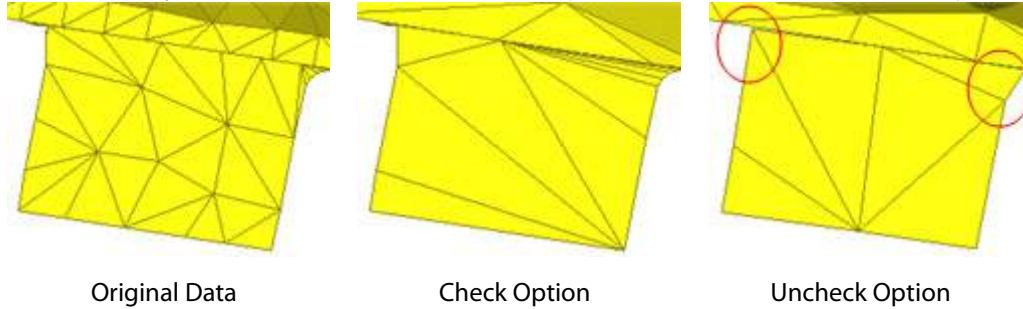


Allowable Deviation – Reduces the number of poly-faces within an allowable deviation between an original shape and decimated results.

More Options

Do Not Move Poly-Vertices – Preserves the original position of poly-vertices during decimation.

Do Not Modify Boundaries – Preserves the original shape of boundaries during decimation.



Regularize Poly-Faces – Regenerates poly-faces with uniform poly-edge lengths.

Note:

The calculation time for this command may be highly increased with this option.

Large Data Mode – Internally divides target 3D scan data into sub models by unloading and loading between the physical memory and the hard drive during the calculation. This option is useful when dealing with huge numbers of meshes.

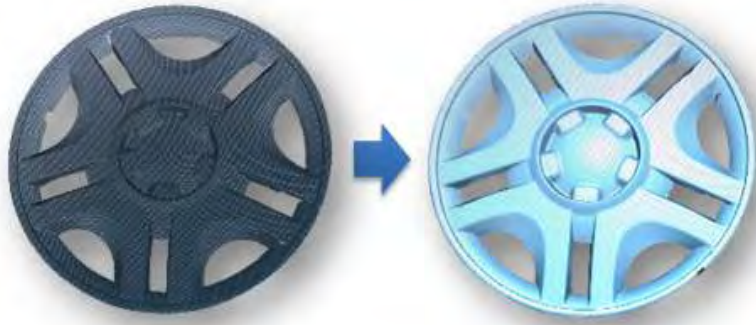
See Also

[Tools > Scan Tools > Mesh Buildup Wizard](#)
[Tools > Mesh Tools > Decimate](#)

Tools > Scan Tools > Normal Information Manager

What

The **Normal Information Manager** command generates, reverses, and removes normal information from 3D scan data.



Why

The **Normal Information Manager** command is useful for:

- Correcting the normal information of scan data
- Creating a defect-free mesh that has no abnormal poly-faces

How to Use

How to generate normal information into point clouds

1. Go to **Tools > Scan Tools > Normal Information Manager** or click the icon in the toolbar.

Note:

To apply this command to 3D scan data, the target 3D scan data should not have child entities.

2. Select all point clouds.



3. Click **OK** to complete the command.



Options in Detail

Methods

This command features 3 methods:

- **Generate**
Generates normal information
 - **Reverse**
Reverses the normal information of 3D scan data
 - **Remove**
Removes normal information from point clouds
-

Generate – Generates normal information into a target point cloud.

Except Point Clouds Which Have Vertex Normal – Preserves existing normal information and is useful when target point clouds are mixed with multiple point clouds that have normal information and do not have normal information.

Reverse – Flips the normal information of point clouds, meshes, and poly-vertices/faces.

Remove – Removes normal information from a point cloud.

See Also

[Tools > Scan Tools > Mesh Buildup Wizard](#)

[Tools > Mesh Tools > Find Defect](#)

[Tools > Mesh Tools > Healing Wizard](#)

[Tools > Mesh Tools > Fix Normal](#)

Tools > Scan Tools > Transform Scan Data

What

The **Transform Scan Data** command transforms 3D scan data by rotating, translating, scaling or using a transformation matrix.



Why

The **Transform Scan Data** command is useful for:

- Transforming scan data to a specific position
- Scale scan data up or down

How to Use

How to transform 3D scan data to a specific position

1. Go to **Tools > Scan Tools > Transform Scan Data** or click the icon in the toolbar.

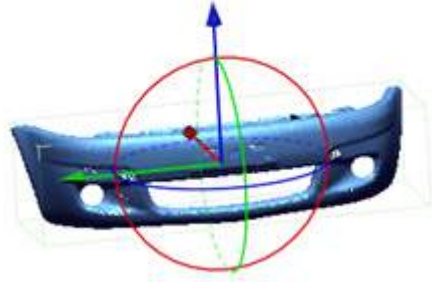
Note:

To apply this command to 3D scan data, the target 3D scan data should not have child entities.

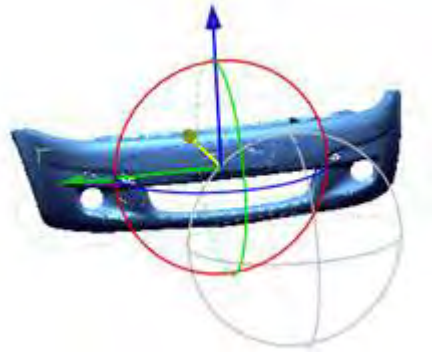
2. Select point clouds or meshes.



3. Click the **Next Stage** button.



4. Transform the scan data using the manipulator on the screen.



5. Click the **OK** button to complete the command.

Options in Detail

Target Entities

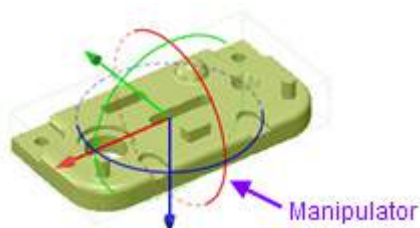
Target Scan Data – Selects a point cloud or mesh.

Methods

This command features 3 methods:

- **Rotate And Translate**
Transforms 3D scan data by rotating and translating
 - **Scale**
Transforms 3D scan data by scaling
 - **By Matrix**
Transforms 3D scan data by using a transformation matrix
-

Rotate And Translate – Transforms target 3D scan data by rotating and translating with manipulators for the X, Y and Z coordinates.



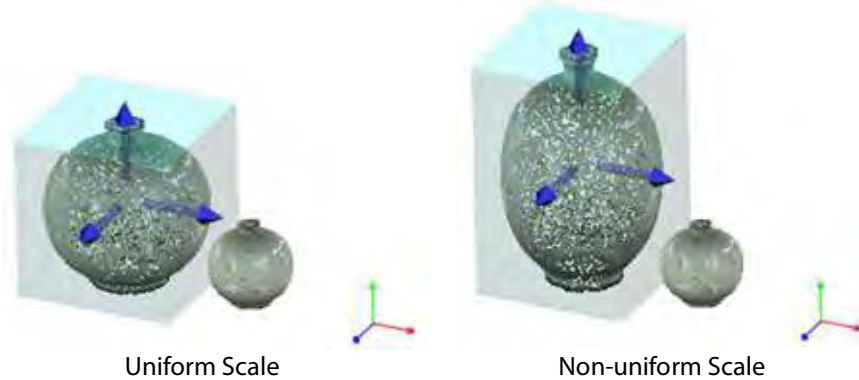
Use Local Coordinate – Uses an existing Ref.Coordinate as a local coordinate.

Rotate – Selects rotational values for the X, Y and Z coordinates.

Translate – Selects translation values for the X, Y and Z coordinates.

Scale – Scales scan data from its center, global origin, or by a custom position.

Uniform Scale – Scales target scan data along the X, Y and Z directions concurrently.



Unit Converter – Converts scale value using the **From** and **To** options.

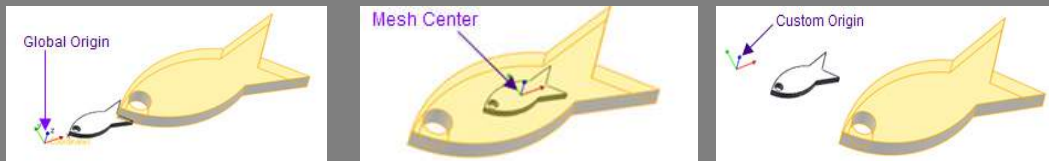
Note:

When the Uniform Scale is manually defined, the Unit Converter will be changed by default. The default unit follows the Default Measure Unit defined in the General Tab within Preferences (**File > Preferences**).

Scale Center – Selects a center position of scale.

Note:

Global Origin, the global coordinate, **Mesh Center**, the center of mesh, or **Custom**, the custom origin, can be defined.



By Matrix – Defines the transformational values in a transformation matrix.

2	0	0	300
0	1	0	0
0	0	1	0
0	0	0	1



Apply Matrix File – Import a transformation matrix file using the **Matrix File Open** button and applies it to the current coordinates of a model.

Set – Applies an imported matrix.

Multiply – Multiplies an imported matrix.

Inverse Multiply – Inverses and multiplies an imported matrix.

Transpose – Transposes an imported matrix.

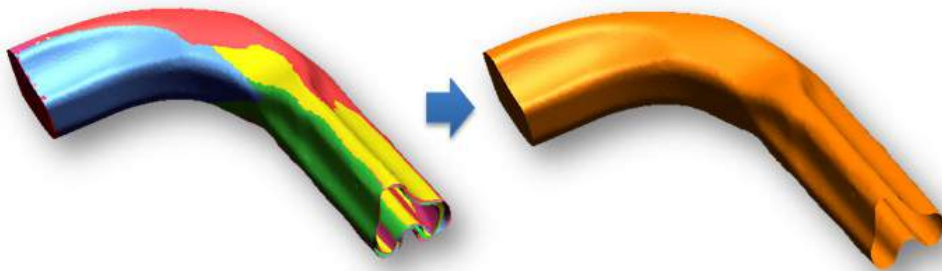
See Also

[User Interface > Docking Bars > Properties](#)
[Tools > Mesh Tools > Mirror](#)
[Insert > Modeling Feature > Transform Body](#)

Tools > Scan Tools > Average Meshes

What

The **Average Meshes** command averages meshes and creates a mesh or feature shape that can be used as a master part.



Why

The **Average Meshes** command is useful for:

- Creating a master part model from multiple 3D scans
- Finding a master feature shape from patterned feature shapes

How to Use

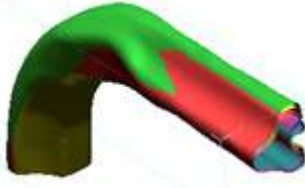
A. How to create a master part model from multiple 3D scans

1. Go to **Tools > Scan Tools > Average Meshes** or click the icon in the toolbar.

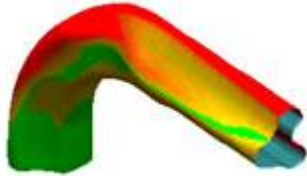
Note:

To apply this command to meshes, the target meshes should not have child entities.

2. Select all target meshes.



3. Click the **Next Stage** button. The result will be shown with its deviation color map.

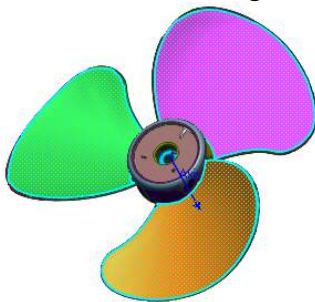


4. Click the **OK** button to complete the command. Hide other meshes.



B. How to find a master feature shape from patterned feature shapes

1. Go to **Tools > Scan Tools > Average Meshes** or click the icon in the toolbar.
2. Select all freeform regions.

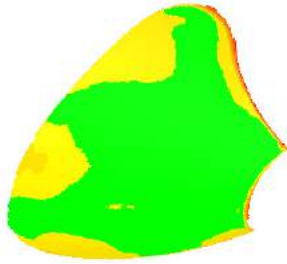


3. Check the **Align Patterned Targets** option and then check the **Circular Pattern** option.
4. Click the **Estimate** button to define a rotation axis and its center position.
5. Check the **Refine Alignment** option to align selected feature shapes.

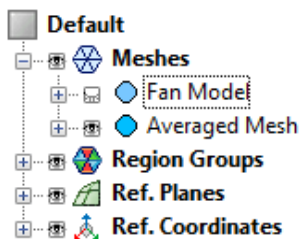
Note:

Selected feature shapes will be aligned by using a defined axis, axis center, and offset angle to average and create a master feature shape from them.

3. Click the **Next Stage** button. The result will be shown with its deviation color map.



4. Click the **OK** button to complete the command. Hide the original mesh.



Options in Detail

Stages

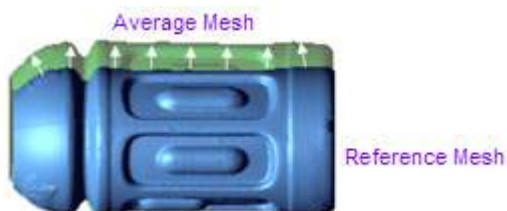
This command consists of 2 stages :

- **Define Targets stage**
Defines target meshes or feature shapes
 - **Check Deviation stage**
Checks results using a deviation color map
-

Stage 1/2 – Define Targets

Target – Selects meshes or feature regions.

Reference – Defines reference entities from selected entities. Otherwise, an entity that has a minimum deviation will be chosen as a reference entity from selected entities by default.



Note:

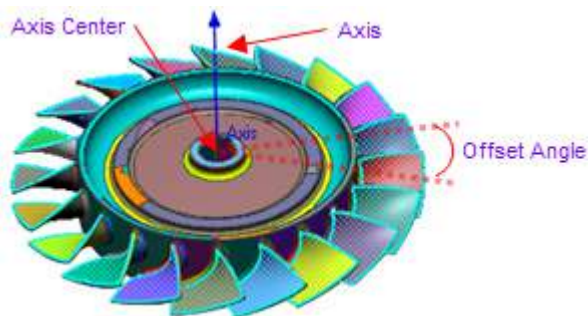
Other selected entities will be aligned into the reference and then an averaged entity will be created from the reference.

Max. Deviation – Specifies a maximum distance that can be deformed from a reference entity.

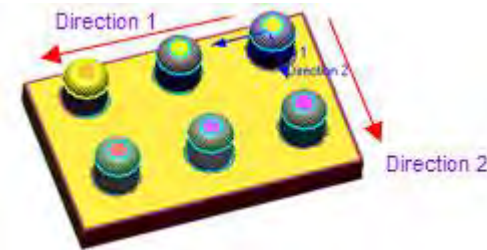
Weighted Average – Assigns more weight to poly-vertices that are located near the same position among target meshes and reduces the effect of outliers.

Align Patterned Targets – Transforms targets that are pattern shaped to a position and then calculated the averaged shape.

Circular Pattern – Specifies the parameters of a circular pattern such as: **Axis**, **Axis Center** and **Offset Angle**. These parameters can be automatically calculated by clicking the **Estimate** button.



Linear Pattern – Specifies the parameters of a linear pattern such as: **Direction 1** and **Direction 2**. These parameters can be automatically calculated by clicking the **Estimate** button.



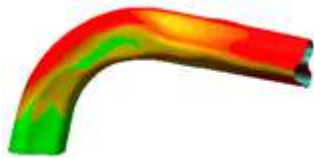
Refine Alignment – Performs a best-fit alignment to well align meshes after targets are internally collected on a position.

Use Selected Data Only – Defines partial areas on a reference entity. The best-fit alignment will be performed among target entities by using the selected partial areas only.

Stage 2/2 – Check Deviation

Target – Checks the deviations between each target entity and an averaged mesh.

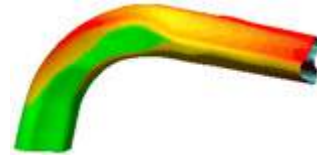
Statistics – Checks the statistical results calculated among selected target entities in the **Target** list. According to the **Deviation Type**, the statistical values and color map can be checked on a model.



Max. Error



Min. Error



Avg. Error

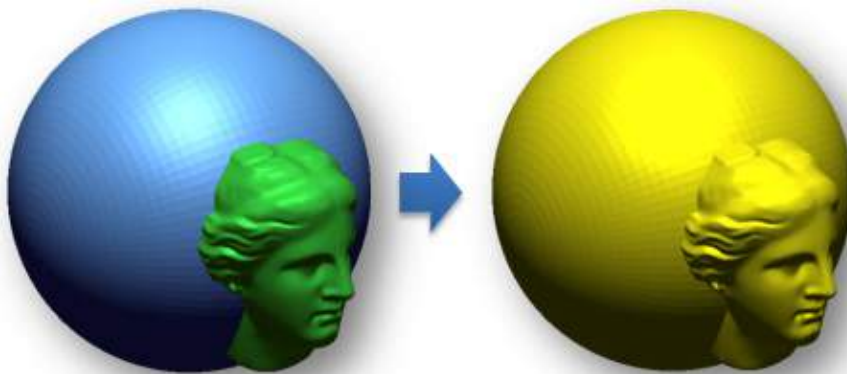
See Also

[Measure > Mesh Deviations](#)

Tools > Scan Tools > Boolean

What

The **Boolean** command uses the Boolean operation to create a difference, intersection, or union between two meshes.



Why

The **Boolean** command is useful for:

- Combining multiple scans into a single mesh model by using a Boolean operations such as Merge, Cut, or Intersect

How to Use

A. How to merge multiple 3D scans into a single mesh

1. Go to **Tools > Scan Tools > Boolean** or click the icon in the toolbar.

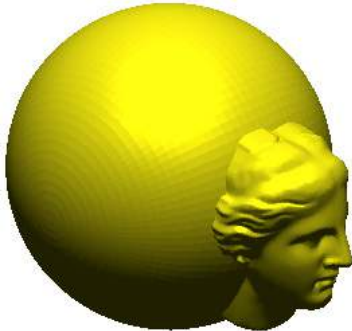
Note:

To apply this command to meshes, the target meshes should not have child entities.

2. Change the method to **Merge**. Define **Tool Mesh** and **Target Mesh**.

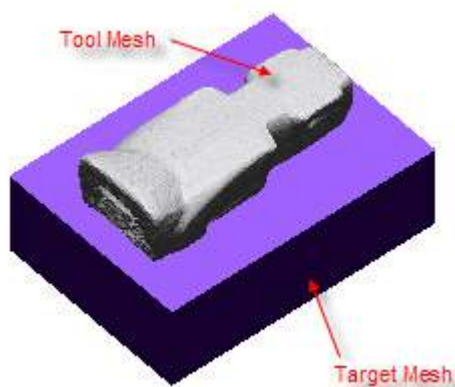


3. Check the **Remove Original Mesh** option to remove the original mesh after the operation. Click the **OK** button to complete the command.



B. How to cut a 3D scan with another 3D scan

1. Click **Tools > Scan Tools > Boolean** or click the icon in the toolbar.
2. Change the method to **Cut**. Define **Tool Mesh** and **Target Mesh**.



3. Click the **OK** button to complete the command.



Options in Detail

Methods

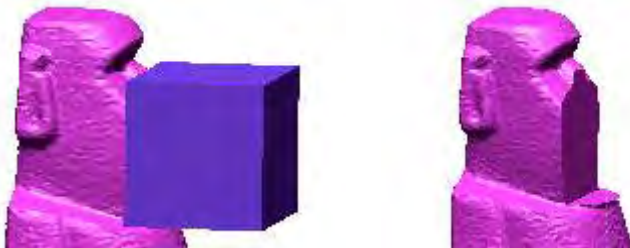
This command features 3 methods:

- **Merge**
Merges two meshes
 - **Cut**
Cuts a target mesh by a tool mesh
 - **Intersect**
Creates an intersected mesh between a tool mesh and a target mesh
-

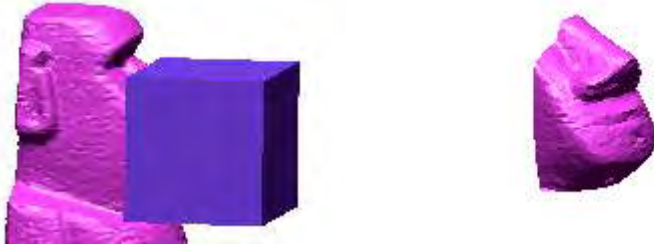
Merge – Merges meshes and create a single mesh.



Cut – Cuts a target mesh by a tool mesh.



Intersect – Creates an intersected mesh between a tool mesh and a target mesh.



More Options

Remove Original Data – Removes original 3D scan data after merging.

See Also

[Insert > Solid > Boolean](#)

7.5.2. Tools > Mesh Tools

The **Mesh Tools** menu features various commands used to edit, heal, and generate a complete mesh from existing 3D scan data.

Mesh Tools are divided into 4 tool groups used to:

Healing 3D Scan Data

- **Find Defect**
Finds defects in mesh
- **Healing Wizard**
Heals abnormal poly-faces and cleans defects in mesh
- **Fix Normal**
Corrects normal information of poly-faces in mesh
- **Edit Boundaries**
Edits mesh boundaries
- **Sew Boundaries**
Sews mesh boundaries

Optimizing 3D Scan Data

- **Retriangulate**
Regenerates poly-faces based on a mesh
- **Global Remesh**
Regenerates poly-faces with uniform poly-edge lengths on a mesh

- **Optimize Mesh**
Optimizes mesh based on geometric feature shapes
- **Edit Poly-Edge**
Edits poly-edges on a mesh
- **Virtual Rescan**
Rescans feature shapes and regenerates mesh
- **Rewrap**
Rewraps feature shapes and regenerates poly-faces on a mesh

Editing 3D Scan Data

- **Fill Holes**
Fills missing holes in mesh
- **Defeature**
Deletes feature shapes and regenerates poly-faces on a mesh
- **Remove Marker**
Removes scanned markers from a mesh
- **Add Poly-Face**
Partially adds poly-faces on a mesh
- **Smooth**
Smooths and reduces the roughness of a mesh
- **Smart Brush**
Interactively edits poly-faces by using a paint brush selection mode
- **Decimate**
Merges and reduces the number of poly-faces
- **Subdivide**
Divides and increases the number of poly-faces
- **Enhance Shape**
Enhances feature shapes of a mesh

Modifying 3D Scan Data

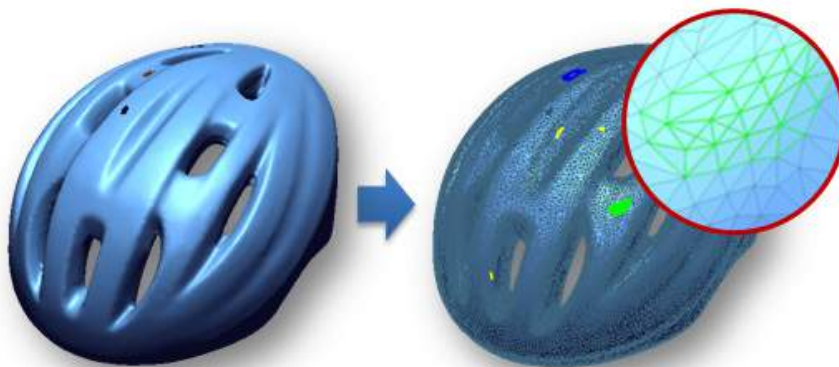
- **Refit**
Refits a mesh to a reference entity
- **Insert**
Converts other entities and inserts into a mesh
- **Split**
Splits a mesh by a tool entity

- **Trim**
Trims poly-faces on a mesh
- **Mirror**
Mirrors part of a mesh
- **Offset**
Increases or decreases the size of a mesh by a certain distance
- **Thicken**
Thickens mesh
- **Convert Body**
Converts bodies into mesh

Tools > Mesh Tools > Find Defect

What

The **Find Defect** command detects four types of defects: **Non-Manifold Poly-Faces**, **Crossing Poly-Faces**, **Redundant Poly-Faces** and **Tangled or Reversed Poly-Faces**.



Why

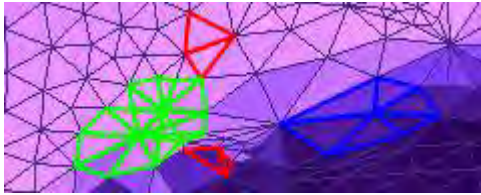
The **Find Defect** command is useful for:

- Finding defects in a mesh and deleting them
- Determining the quality of a generated mesh

How to Use

How to Find defects in a mesh and delete them

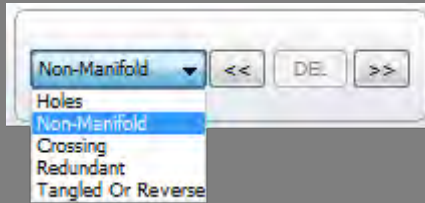
1. Go to **Tools > Mesh Tools > Find Defect** or click the icon in the toolbar.



2. Check that detected defects have different colors and how many defects exist in the mesh.

Note:

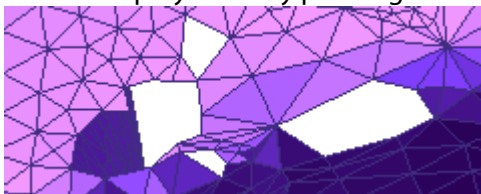
To navigate defects, click the left or right << >> buttons. Detected defects can be navigated and directly deleted by using the Navigation Tool Palette.



3. Click **OK** to complete the command. All defective abnormal poly-faces will be selected.



4. Delete the poly-faces by pressing the **Del** key or go to **Edit > Delete**.



Options in Detail

Non-Manifold Poly-Faces – Shows the total number of detected **Non-Manifold Poly-Faces** and enables the changing of their displayed colors.

Crossing Poly-Faces - Shows the total number of detected **Crossing Poly-Faces** and enables the changing of their displayed color.

Redundant Poly-Faces – Shows total number of detected **Redundant Poly-Faces** and enables the changing of their displayed color.

Tangled Or Reversed Poly-Faces – Shows total number of detected **Tangled or Reversed Poly-Faces** and enables the changing of their displayed color.

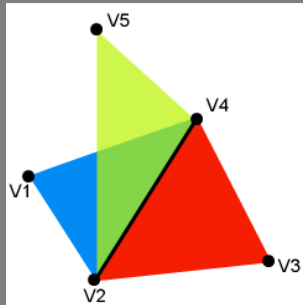
Note:

Non-manifold Poly-Faces, Redundant Poly-Faces, Crossing Poly-Faces and Tangled Or Reversed Poly-Faces are defined as below:

Non-manifold Poly-Face – When poly-faces share three or more poly-edges, it is referred to as non-manifold topology. In a pure mathematical term, to be a manifold topology, only two poly-faces can constitute a poly-edge. However, a poly-edge with only one poly-face is classified as a boundary poly-edge in the application and considered to be normal.

Non-manifold poly-faces may cause severe errors in various calculations, and even fatal errors in other applications such as **NC machining**.

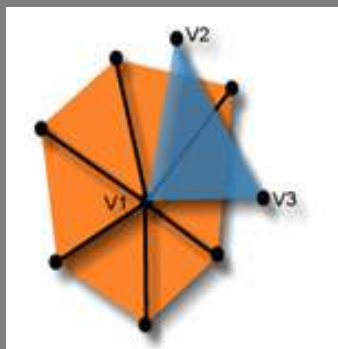
The right picture shows an example of the non-manifold topology, where poly-faces (**V1 V2 V4**), (**V2 V4 V5**), and (**V2 V4 V3**) share poly-edges (**V2 V4**).



Redundant Poly-Face – A redundant poly-face is a type of abnormal poly-face from the viewpoint of a poly-vertex. Ignoring boundaries, a poly-vertex must have an identical number of poly-faces and connecting poly-edges.

A poly-face violating this rule is called a redundant poly-face.

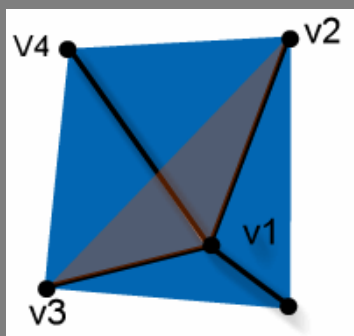
The following picture shows an example of a redundant poly-face (**V1 V2 V3**).



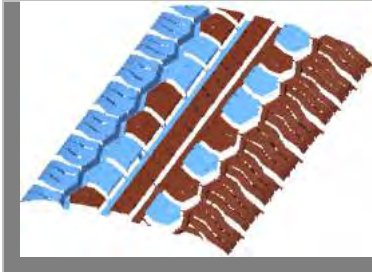
Crossing Poly-Face – When poly-edges sharing a poly-vertex intersect with one another, they are geometrically incorrect and their poly-faces are called crossing poly-faces.

The following image displays a crossing poly-face. The poly-edge (**V2 V3**) of poly-face (**V1 V2 V3**) is crossed with the poly-edge (**V1 V4**).

Although crossing poly-face is neither a topological definition nor a possible cause of serious problems, it may cause errors in geometrical calculations.



Tangled Or Reversed Poly-Face – Every poly-face has normal information. If the normal direction of poly-face is opposite to neighboring poly-face, it is a tangled or reversed poly-face.



See Also

[Tools > Scan Tools > Normal Information Manager](#)

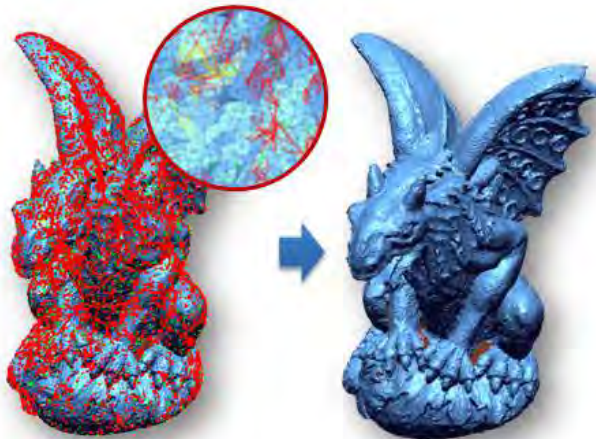
[Tools > Mesh Tools > Healing Wizard](#)

[Tools > Mesh Tools > Fix Normal](#)

Tools > Mesh Tools > Healing Wizard

What

The **Healing Wizard** command detects abnormal poly-faces in a mesh and automatically heals them. When 3D scan data that has been scanned in different directions by using a 3D scanner are aligned and merged into a single mesh, some abnormal poly-faces such as folded poly-faces, dangling poly-faces, and small noisy clusters may occur. To optimize the mesh, the defects on the mesh need to be healed.



Why

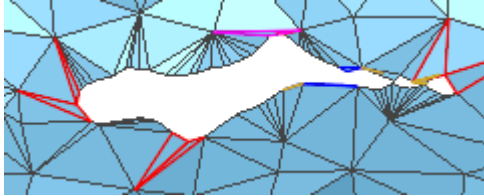
The **Healing Wizard** command is useful for:

- Creating an optimal mesh model without any abnormal poly-faces
- Healing defects

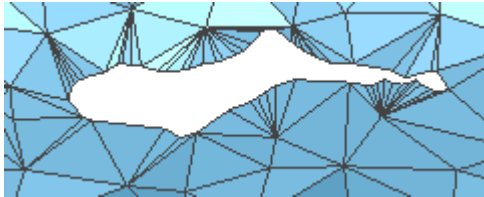
How to Use

How to heal abnormal poly-faces and clean defects from a mesh

1. Go to **Tools > Mesh Tools > Healing Wizard** or click on the icon in the toolbar.
2. Abnormal poly-faces will be automatically detected in a mesh and the results will be displayed with different colors. Check detected abnormal poly-faces and click the **OK** button to heal them at all the same time.



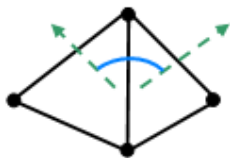
3. Check that all detected defects are healed from the mesh.



Options in Detail

Folded Poly-Faces – Automatically checks if poly-faces have been folded in the same place on a mesh and that the angle between poly-faces is smaller than a specified value.

Angle Is Smaller Than – Specifies angle criteria to detect folded poly-faces.



Angle Between Poly-faces

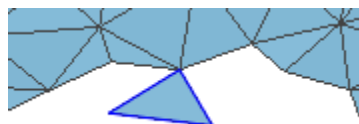
Note:

When the angle criteria are changed, the number of detected folded poly-faces will be automatically updated.

Dangling Poly-Faces – Automatically checks if 2 or 3 sided opened poly-faces exist on the boundaries of a mesh.



2 Side Open



3 Side Open

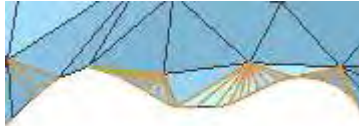
Small Clusters – Checks if a group of connected poly-faces, known as small clusters, have less poly-faces than a specified value exist on a mesh.



Small Clusters

Poly-Face Count In A Cluster – Specifies count criteria used to detect small clusters.

Small Poly-Faces – Checks if small poly-faces exist that have a smaller area than specified area criteria.



Small Poly-faces

Area Is Smaller Than – Specifies area criteria used to detect small poly-faces.

Non-Manifold Poly-Faces – Checks if non-manifold poly-faces and redundant poly-faces exist on a mesh. For more information, see [Tools > Mesh Tools > Find Defect](#).

Crossing Poly-Faces – Checks if crossing poly-faces exist in a mesh.

Method – Contains 3 methods used to heal crossing poly-faces.

Smooth – Smooths poly-faces around crossing poly-faces.

Merge Poly-Vertices – Merges poly-vertices of crossing poly-faces.

Delete And Fill Hole – Deletes crossing poly-faces and fills the deleted area.

Small Tunnels – Detects poly-faces shaped like a tunnel or handle, known as small handles. This option is automatically checked if small tunnels that have lesser poly-faces than the count criteria exist on the mesh.



Tunnel Shape

Poly-Face Count In Tunnel – Specifies the count criteria used to detect small tunnels.

See Also

[Tools > Scan Tools > Normal Information Manager](#)

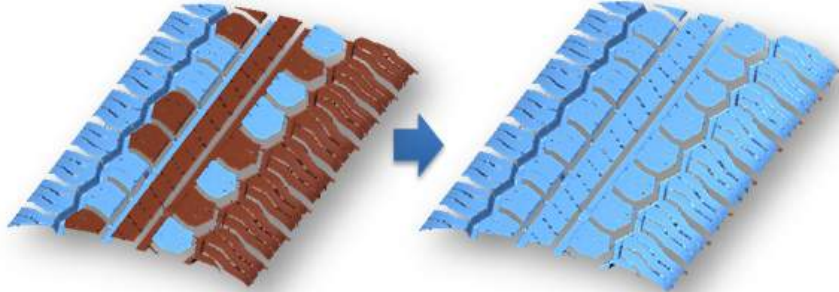
[Tools > Mesh Tools > Find Defect](#)

[Tools > Mesh Tools > Fix Normal](#)

Tools > Mesh Tools > Fix Normal

What

The **Fix Normal** command harmonizes normal information of poly-faces in a mesh.



Why

The **Fix Normal** command is useful for:

- Correcting the normal information of poly-faces to create an optimal mesh
- Fixing the normal information of poly-faces in tessellated CAD mesh that has been imported from neutral file formats such as IGES or STEP

How to Use

How to correct normal information of poly-faces in a mesh

1. Go to **Tools > Mesh Tools > Fix Normal** or click the icon in the toolbar.



2. Choose the **Automatic** method and check the **Propagate Topologically** option. To reverse the normal direction manually, choose the **Interactive** method and pick a seed poly-face on the mesh.
3. Pick a seed poly-face on a mesh and check the results.



Options in Detail

Methods

This tool features 3 methods:

- **Automatic**
Automatically fixes the normal information of poly-faces in a mesh
 - **Interactive**
Reverses the normal information of connected poly-faces
 - **Entire Mesh**
Reverses the normal information of entire poly-faces in a mesh
-

Automatic – Fixes the normal information of poly-faces in a mesh by propagating topologically, propagating geometrically, or repairing tangled poly-faces.



Propagate Topologically



Propagate Geometrically



Tangled Poly-Face

Propagate Topologically – Fixes the normal information of poly-faces inside of its boundary.

Note:

Check the **Propagate Topologically** option if the mesh has no non-manifold poly-faces and the boundaries exist in the mesh.

Propagate Geometrically – Fixes the normal information of poly-faces regardless of its boundaries. Check the **Propagate Geometrically** option of a mesh that has disjointed poly-edges and non-manifold poly-faces and boundaries that do not exist in the mesh.

Repair Tangled Poly-Faces – Repairs twisted and tangled topology and fixes the normal information of poly-faces. The **Repair Tangled Poly-Faces** option is used to correct the twisted topology that may occur when triangulating 3D scan data which has thin walls.

Interactive – Reverses the normal information of connected poly-faces.

Entire Mesh – Reverses the normal information of entire poly-faces in mesh.

See Also

[Tools > Scan Tools > Normal Information Manager](#)

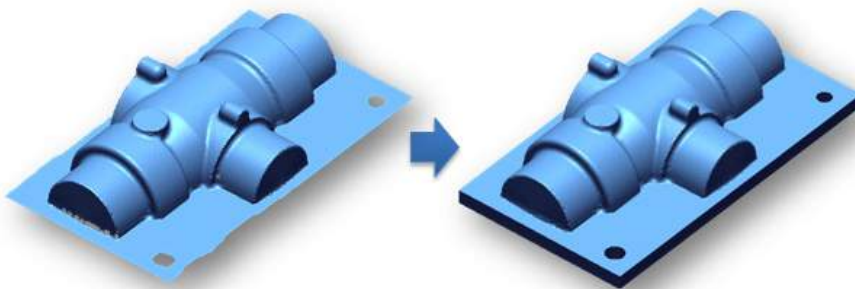
[Tools > Mesh Tools > Find Defect](#)

[Tools > Mesh Tools > Healing Wizard](#)

Tools > Mesh Tools > Edit Boundaries

What

The **Edit Boundaries** command edits the boundaries of mesh. This command features multiple boundary editing methods such as **Smooth**, **Shrink**, **Fit**, **Extend**, **Extrude**, and **Fill**.



Why

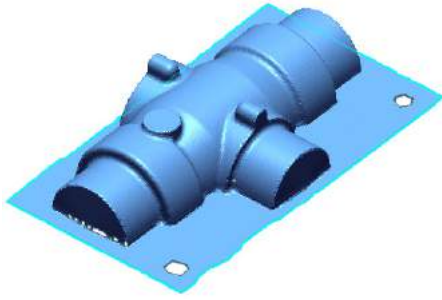
The **Edit Boundaries** command is useful for:

- Reducing the effect of noise and roughness on mesh boundaries
- Modifying the feature shapes of mesh
- Creating a CAD-like mesh

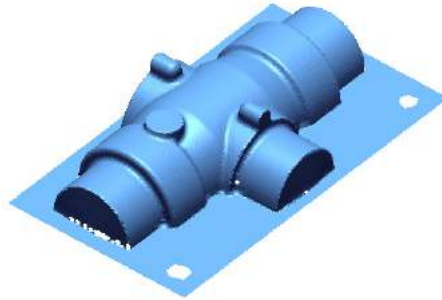
How to Use

How to edit mesh boundaries

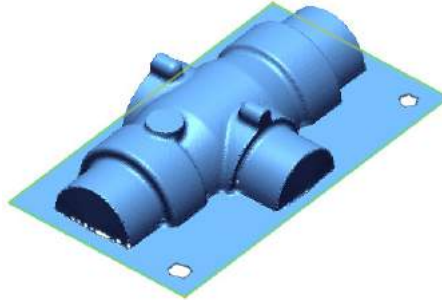
1. Go to **Tools > Mesh Tools > Edit Boundaries** or click the icon in the toolbar.
2. Choose the outer boundary of a mesh as the target Boundaries.



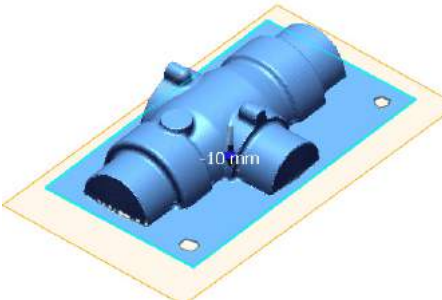
3. Choose the **Fit** method and define the **Fitting Shape** as **Rectangle**.
4. Click the **OK** button and check the results.



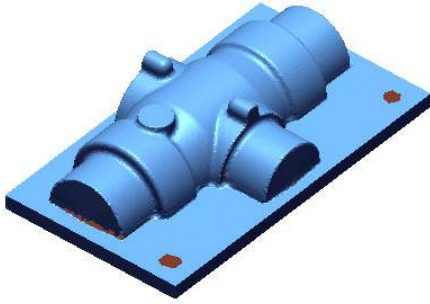
5. Run the **Edit Boundaries** command again.
6. Choose the fitted outer boundary as the target Boundaries.



7. Choose the **Extrude** method and set the **Distance** to **-10mm** and then check the **Cap** option to close the bottom of the mesh.



8. Click the **OK** button to complete the command.



Options in Detail

Target Entities

Boundaries – Selects boundaries of mesh. The **Split Boundary** button splits a boundary at selected positions. This button is useful for partially editing boundaries of a mesh.



Before splitting boundary



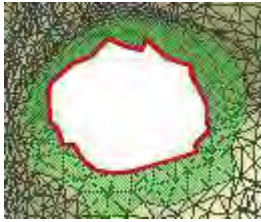
After splitting boundary

Methods

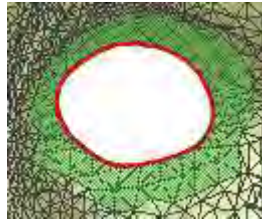
This command features 6 methods:

- **Smooth**
Reduces the effects of roughness on a boundary
 - **Shrink**
Removes poly-faces from boundary
 - **Fit**
Fits a boundary to a specific feature shape
 - **Extend**
Extends a boundary by a specific distance
 - **Extrude**
Extrudes a boundary by a specific distance
 - **Fill**
Fills poly-faces in a boundary
-

Smooth – Reduces noises and effect of roughness in boundary of mesh.



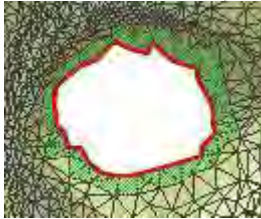
Original Boundary



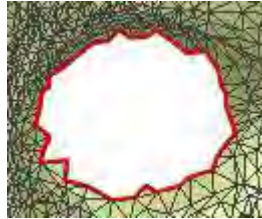
Smooth Boundary

Select N-Depth From Boundary – Specifies the influence range of the smoothing effect from a target boundary. The influence range will be highlighted on a mesh interactively with this option.

Shrink – Removes poly-faces from a boundary.



Original Boundary



Shrink Boundary

Note:

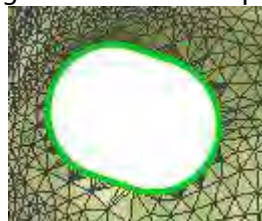
3D scan data may have roughness and noise in the area near a boundary caused by shadows or vibrations during the scanning process. These defects may adversely affect the alignment between scan data as well as the merging process. To get better results in scan data processing, defects in the boundaries of mesh should be cleaned.

Select N-Depth From Boundary – Specifies the influence range of the shrinking effect from a target boundary. The influence range will be highlighted on a mesh interactively with this option.

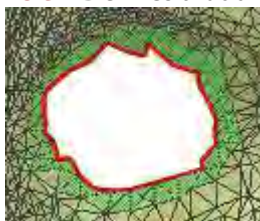
Fit – Fits a boundary to specific feature shapes such as **Circle**, **Slot**, **Rectangle**, **Polygon** and **Freeform**.

Fitting Shape – Defines specific features for a boundary to be fit to.

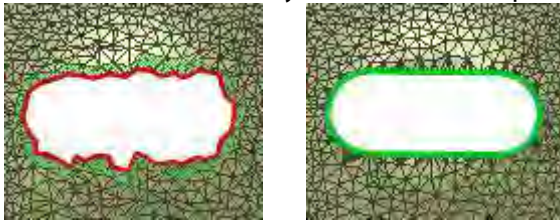
Auto – Automatically recognizes a feature shape within an allowable tolerance.



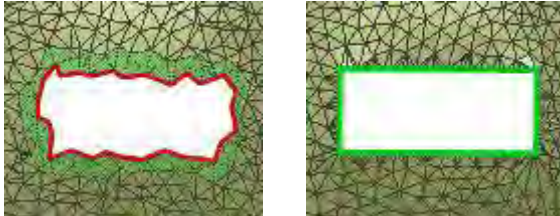
Circle – Defines a boundary fitted feature shape as a circle.



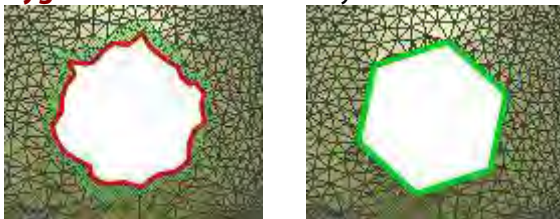
Slot – Defines a boundary fitted feature shape as a slot.



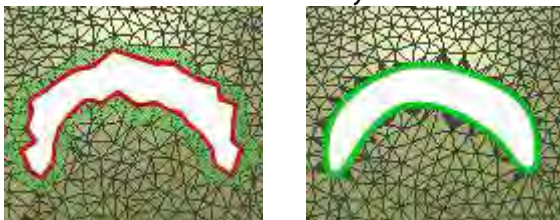
Rectangle – Defines a boundary fitted feature shape as a rectangle.



Polygon – Defines a boundary fitted feature shape as a polygon.



Freeform – Defines a boundary fitted feature shape as a freeform shape.

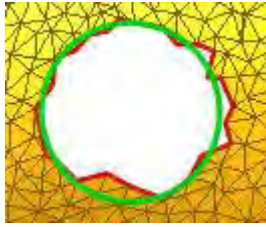


Allowable Tolerance – Specifies allowable tolerance for extracting a feature shape from a boundary. If any feature shape cannot be matched with a boundary a freeform shape will be chosen.

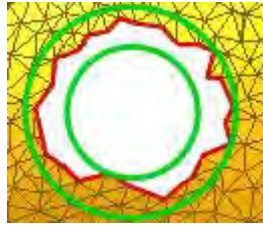
Note:

This option can be used only when the Fitting Shape option is set to **Auto**. Click the **Estimate** button and the best tolerance will be automatically calculated based on the shape of the selected target boundaries.

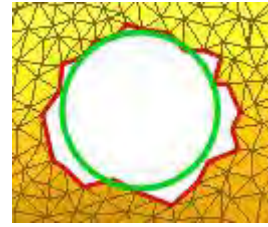
Fitting Type – Creates a best-fitted value by the least square fitting method when the **Best Fit** option is selected. The least squares method is the approximate solution from distributed data. The **Min. Bound** option finds the smallest radial distance value between a pair of concentric circles, and **Max. Bound** finds the largest value. The **Hole** method is available when the fitting type is set to **Auto** or **Circle**. This option finds an inscribed circle from poly-vertices around a boundary.



Best Fit



Min Boundary & Max Boundary



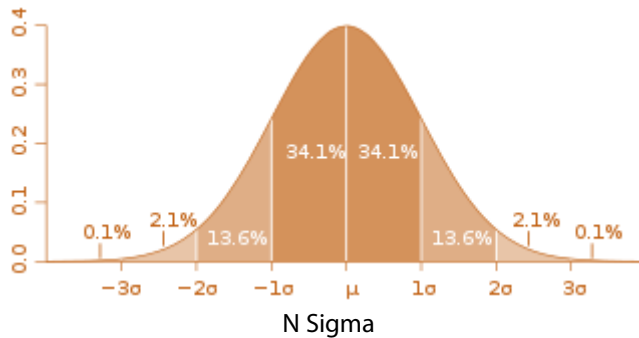
Hole

Remove Outlier – Removes outlier data to achieve more accurate fitting results.

The **N*Sigma** method requires an N value and excludes data that is larger than N times the standard deviation, also known as the sigma.

The **Ratio** method requires a ratio value and only data that falls into the selected value will be used.

The **Absolute Distance** method requires a distance and the fitting will be done by excluding any data outside of that distance.



Sampling Ratio – Samples data points by a specified value. When the ratio is 100%, all selected data will be used. When the ratio is 50%, half of the selected data will be used.

Use Only Effective Data – Uses only effective data when fitting. Effective data is the actual data points used in fitting after sampling and outlier removal.

Rounded Corner – Fits a boundary to a specific feature shape with rounded corners and is available when the fitting type is set to **Auto**, **Rectangle** or **Polygon**.



Non-Rounded Corner



Rounded Corner

No. Of Edges – Decides the number of edges for a polygonal feature shape and is available when the fitting type is set to **Auto** or **Polygon**.



Number of edges: 3



Number of edges: 5

User Defined Curves – Fits a boundary to user defined curves and is available when the fitting type is set to **Freeform**.

Note:

When using user-defined curves, the curves need to be prepared before the target mesh feature in the Feature Tree. For more information, see [Getting Started > Introduction to the Interface > User Interface Overview](#).

Smoothness – Defines the smoothness of a boundary shape by using a value between the Min and Max limits. It is available when the fitting type is set to **Freeform** and user-defined curves are not selected.

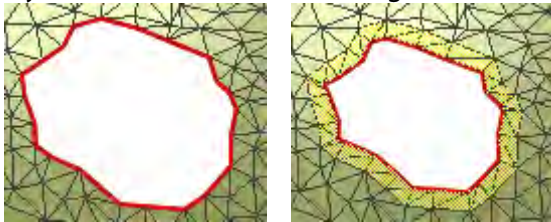


Min. Smoothness



Max. Smoothness

Extend – Adds poly-faces around a boundary and extends the shape. The extending distance of poly-faces can be controlled using **Distance** under **Extend Options**.



Distance – Specifies an extending distance.

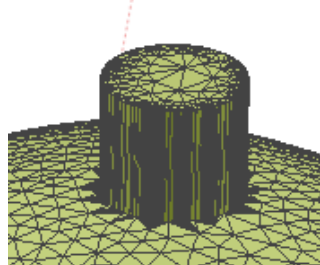
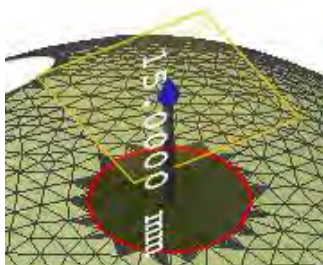
Refine And Smooth Extended Mesh – Refines poly-faces in an extended area by removing crossing poly-faces that can be generated during the extension, and smooths poly-faces.

No. Of Smoothing Iterations – Specifies the number of smoothing iterations.

Extrude – Extrudes a boundary by a specified distance or up to a specified plane. The **Blind**, **Up To Plane**, and **Custom** extrusion methods are available.

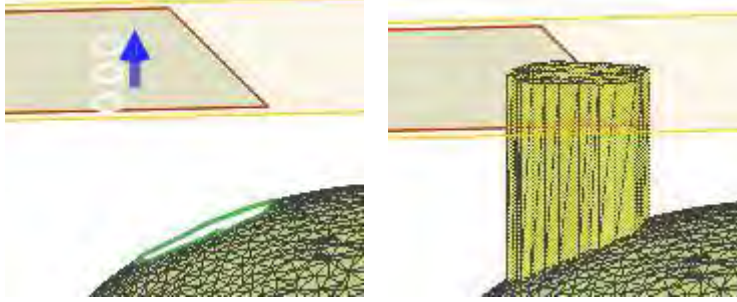
Method – Defines extrusion methods.

Blind – Extrudes a boundary by a specified distance.

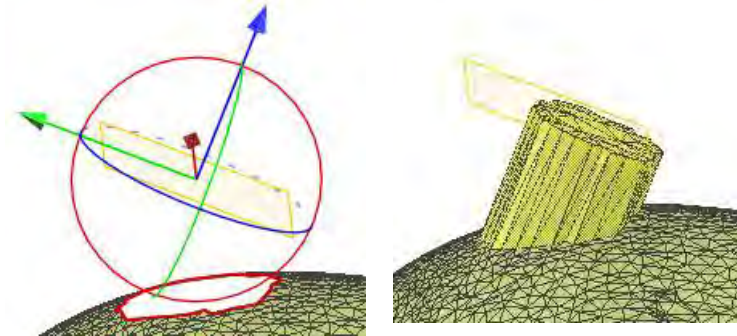


Cap – Closes a boundary at the end of an extrusion.

Up To Plane – Extrudes a boundary up to a specified plane. Ref.Plane, 2D Sketch, and planar face can be selected as a target plane.



Custom – Extrudes a boundary up to a specific position. The specific position can be defined by using a manipulator for the X, Y, and Z coordinates.



Fill – Fills poly-faces in a boundary. This option is similar to the **Fill Hole** command. For more information, see [Tools > Mesh Tools > Fill Holes](#).

Flat – Fills a boundary by using flat poly-faces.

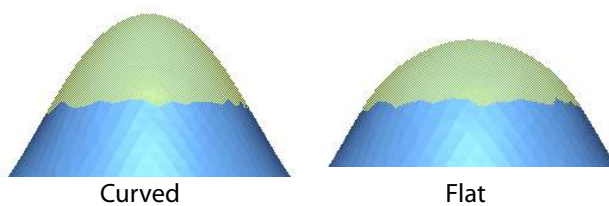
Curvature – Fills a boundary by following the curvature flow of poly-faces around a boundary.

Merge Splitted Boundaries – Merges split boundaries into a single boundary.



Smooth Boundary – Refines a boundary for better filling results.

Flatness Of Filled Holes – Adjusts the flatness of a resulting shape in a filled boundary.



See Also

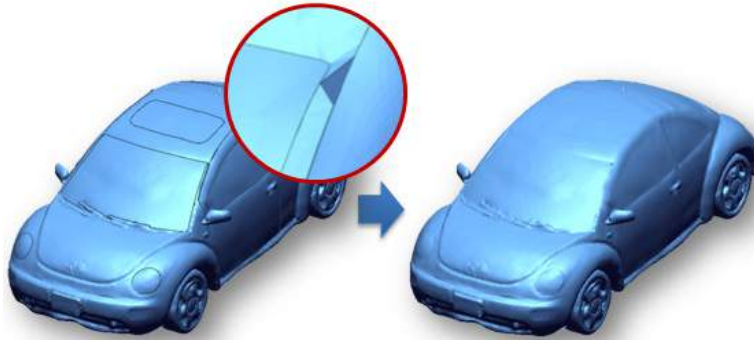
[Tools > Mesh Tools > Sew Boundaries](#)

[Tools > Mesh Tools > Fill Holes](#)

Tools > Mesh Tools > Sew Boundaries

What

The **Sew Boundaries** command sews narrow line type holes in boundaries.



Why

The **Sew Boundaries** command is useful for:

- Creating an optimal mesh model without any torn boundaries
- Sewing torn boundaries in tessellated CAD mesh that have been imported from neutral file formats such as IGES or STEP

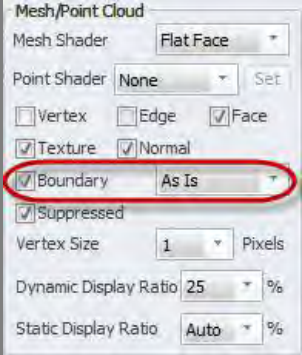
How to Use

How to sew the boundaries of a mesh

1. Go to **Tools > Mesh Tools > Sew Boundaries** or click the icon in the toolbar.

Note:

To check the boundaries of a mesh in the Model View, check the Boundary option in the Display tab.

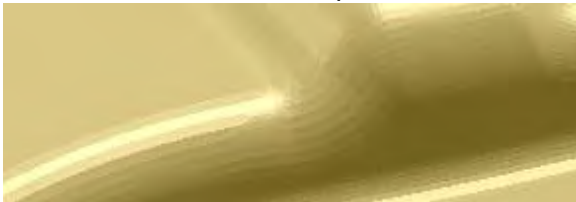


The screenshot shows the 'Mesh/Point Cloud' display options dialog. The 'Boundary' checkbox is checked, and the 'As Is' dropdown menu is highlighted with a red circle. Other options include 'Mesh Shader' (Flat Face), 'Point Shader' (None), 'Vertex' (unchecked), 'Edge' (unchecked), 'Face' (checked), 'Texture' (checked), 'Normal' (checked), 'Suppressed' (checked), 'Vertex Size' (1 Pixels), 'Dynamic Display Ratio' (25%), and 'Static Display Ratio' (Auto%).

2. Click the **Estimate** button in the **Sew If Gap Is Smaller Than** option to calculate an estimated distance from boundaries.



3. Click the **OK** button to complete the command.



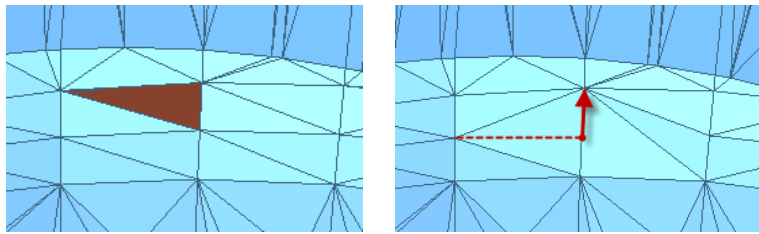
Options in Detail

Sew If Gap Is Smaller Than – Specifies a distance for sewing gaps.



More Options

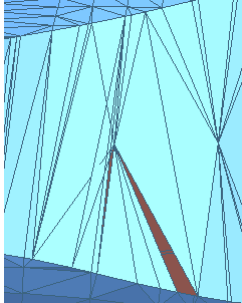
Merge Close Vertices – Merges poly-vertices in close proximity to each other under a specified distance.



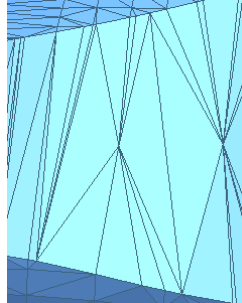
Before Merged

After Merged

Repair Spike Poly-Faces – Repairs spikey poly-faces.



Spike Poly-Faces



Repaired Spike Poly-Faces

See Also

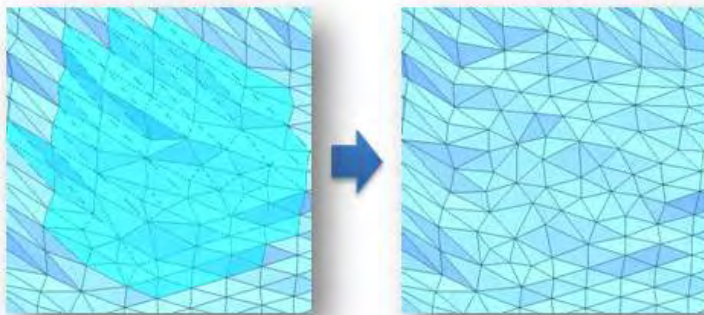
[Tools > Mesh Tools > Edit Boundaries](#)

[Tools > Mesh Tools > Fill Holes](#)

Tools > Mesh Tools > Retriangulate

What

The **Retriangulate** command removes selected poly-faces and regenerates poly-faces by equalizing poly-vertex balance.



Why

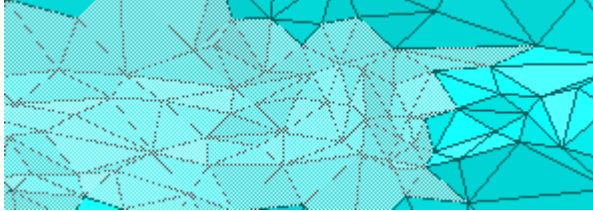
The **Retriangulate** command is useful for:

- Partially refining poly-faces on a mesh

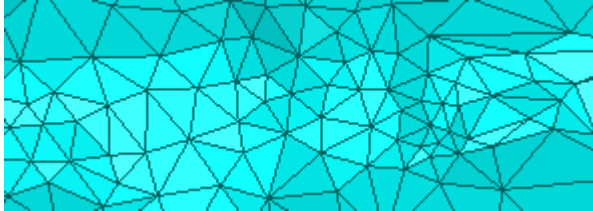
How to Use

How to recreate poly-faces from a mesh

1. Go to **Tools > Mesh Tools > Retriangulate** or click the icon in the toolbar.
2. Select poly-faces on a mesh.



3. Click the **OK** button to complete the command.



See Also

[Tools > Mesh Tools > Global Remesh](#)

[Tools > Mesh Tools > Optimize Mesh](#)

[Tools > Mesh Tools > Edit Poly-Edge](#)

[Tools > Mesh Tools > Virtual Rescan](#)

[Tools > Mesh Tools > Rewrap](#)

[Tools > Mesh Tools > Smart Brush](#)

Tools > Mesh Tools > Global Remesh

What

The **Global Remesh** command globally regenerates poly-faces with uniform poly-edge lengths on a mesh and improves the quality of the mesh. This command also cleans and manifolds solid mesh that has no boundaries.



Why

The **Global Remesh** command is useful for:

- Optimizing mesh so that it has poly-faces globally with uniform poly-edge lengths
- Optimizing mesh to be used for creating a high quality surface body
- Reducing the effects of defects and roughness on a mesh
- Creating a high quality mesh from combined mesh that is complex and overlapped

How to Use

A. How to recreate high quality poly-faces with uniform poly-edge lengths on a mesh

1. Go to **Tools > Mesh Tools > Global Remesh** or click the icon in the toolbar.



2. Check the **Make Clean And Manifold Solid Mesh** option to create an optimal mesh without any defects and holes.
3. Click the **Preview** button to check the results. If the results are not satisfactory, click **Cancel** and change the values in the option and try again.
4. Click the **OK** button to complete the command.

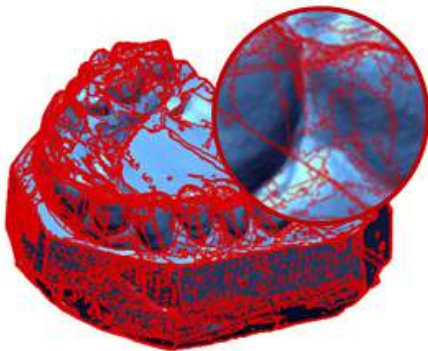


B. How to optimize mesh from combined mesh that is complex and overlapped

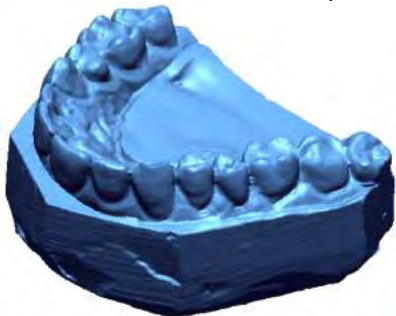
1. Go to **Tools > Mesh Tools > Global Remesh** or click the icon in the toolbar.



2. Check the **Make Clean And Manifold Solid Mesh** option to create an optimal mesh without any defects and holes. When that is complete check the **Assume Combined Multiple Scan Data Sets** option to merge combined and complex overlapped scan sets during the optimizing process.



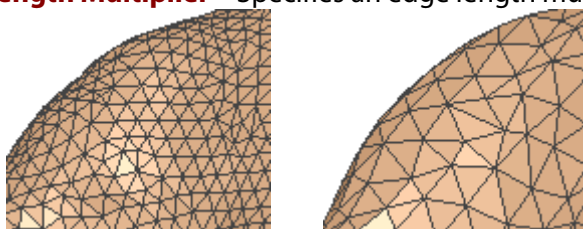
3. Click the **OK** button to complete the command.



Options in Detail

Current Average Edge Length Edge – Shows current average edge length of poly-edges from selected meshes.

Length Multiplier – Specifies an edge length multiplier.



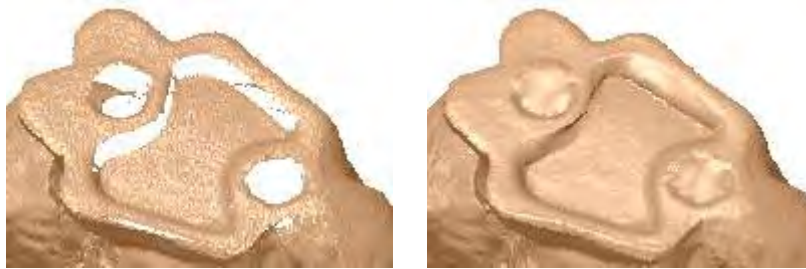
Original Edge Length Edge Length Multiplier: 2

Target Average Edge Length – Defines average length of poly-edges after merging.



More Options

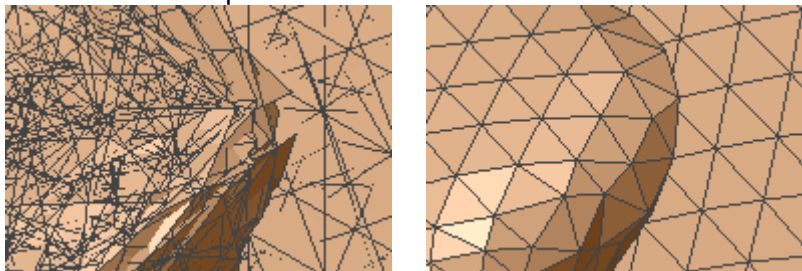
Make Clean And Manifold Solid Mesh – Generates a watertight mesh that has no holes or defects.



Original Data

Enable Option

Assume Combined Multiple Scan Data Sets – Merges combined and complexly overlapped scan data sets during the optimizing process. Overlapping areas are removed with this option.



Original Data

Enable Option

See Also

[Tools > Mesh Tools > Retriangulate](#)

[Tools > Mesh Tools > Optimize Mesh](#)

[Tools > Mesh Tools > Edit Poly-Edge](#)

[Tools > Mesh Tools > Virtual Rescan](#)

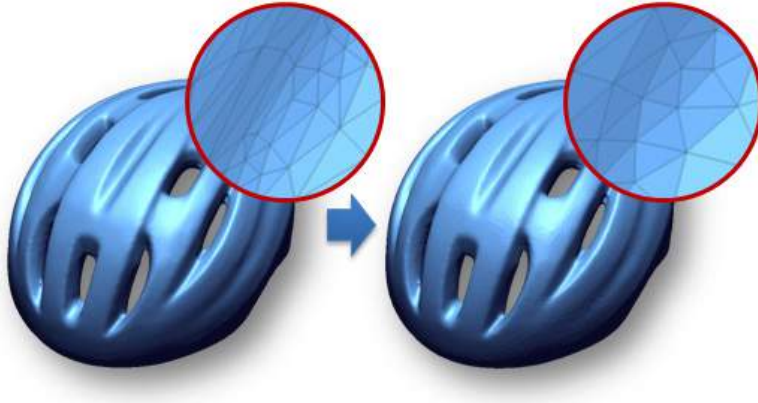
[Tools > Mesh Tools > Rewrap](#)

[Tools > Mesh Tools > Smart Brush](#)

Tools > Mesh Tools > Optimize Mesh

What

The **Optimize Mesh** command optimizes a mesh based on its feature shapes and improves the quality of the mesh. This command also offers advanced options to control the size of poly-faces and the effects of smoothing a model.



Why

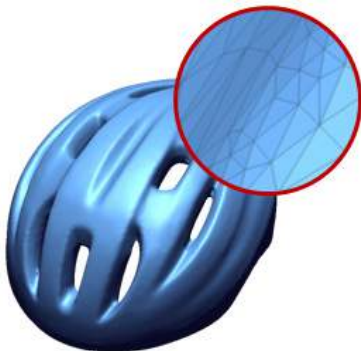
The **Optimize Mesh** command is useful for:

- Optimizing mesh based on its feature shapes
- Optimizing mesh to be used for creating a high quality surface body
- Creating optimized mesh to be used for Finite element analysis, also known as FEA

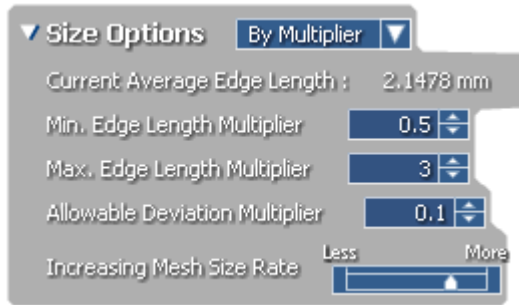
How to Use

A. How to optimize mesh based on its feature shapes to improve the quality

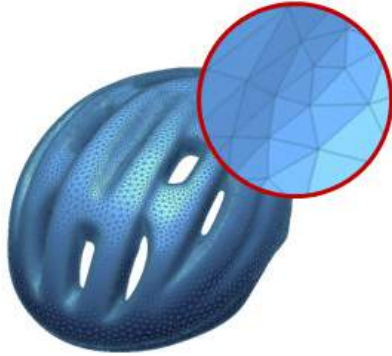
1. Go to **Tools > Mesh Tools > Optimize Mesh** or click the icon in the toolbar.



2. Choose the **High Quality Mesh Conversion** method and adjust options in the **Size Option** panel.

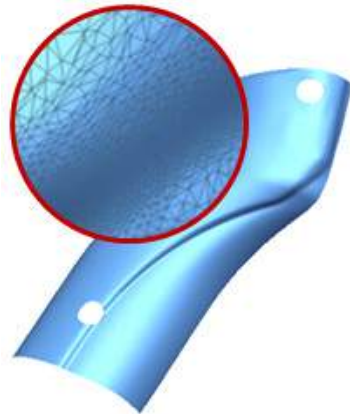


3. Click the **OK** button to complete the command.

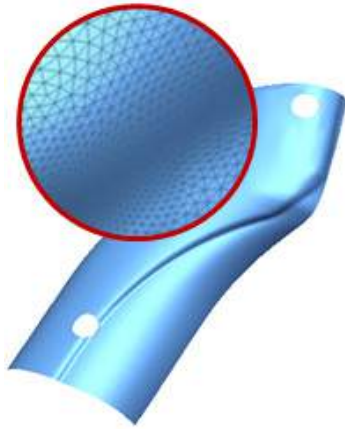


B. How to equalize poly-vertex balance in mesh

1. Go to **Tools > Mesh Tools > Optimize Mesh** or click the icon in the toolbar.



2. Choose the **Equalize Poly-Vertex Balance** method and set the **Max. No. Of Iterations** to **20**.
3. Check the **Smooth** option to improve the mesh quality.
4. Click the **OK** button to complete the command.



Options in Detail

Methods

This command features 3 methods:

- **High Quality Mesh Conversion**

Optimizes mesh based on its feature shapes and improves the quality of mesh

- **Improve Curvature Flow**

Improves curvature flow and refine the feature shapes

- **Equalize Poly-Vertex Balance**

Equalizes poly-vertex balance to create poly-faces with uniform poly-edge lengths

High Quality Mesh Conversion – Optimizes mesh based on feature shapes and improves the quality. It can also be used to control the size of poly-faces and the effects of smoothing a model.

Size Options – Choose either **By Multiplier** or **By Length** to define the target size of poly-faces.

Note:

To easily define the size of poly-faces and the criteria for regenerating poly-faces, choose the **By Multiplier** size option. To manually define the size of poly-faces, choose the **By Length** size option.



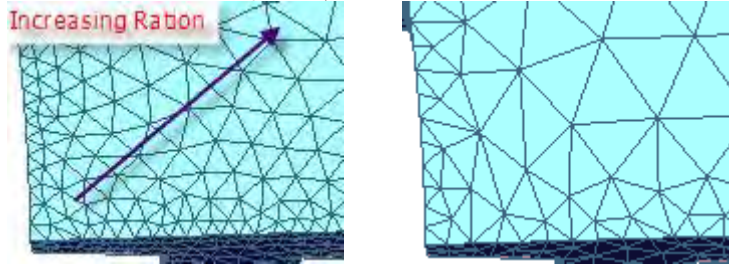
Current Average Edge Length – Shows the current average edge length of poly-edges from a target mesh.

Min. Edge Length Multiplier (Length) – Limits the minimum length of poly-edges in poly-faces.

Max. Edge Length Multiplier (Length) – Limits the maximum length of poly-edges in poly-faces.

Allowable Deviation Multiplier (Deviation) – Limits the deformation of poly-faces within the deviation between an original mesh and optimized mesh.

Increasing Mesh Size Rate – Determines the increasing size rate of poly-faces on a mesh from small to large poly-faces.



Increasing Ratio: Less

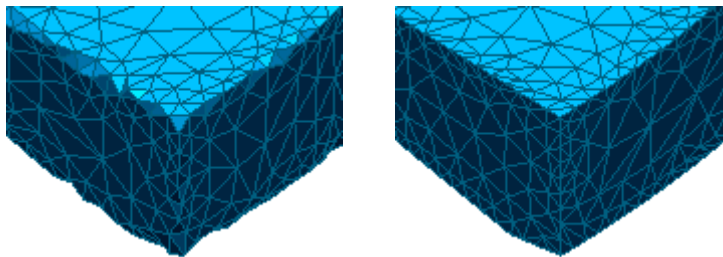
Increasing Ratio: More

No. Of Smoothing Iterations – Specifies the number of smoothing iterations. The smoothing will be applied to the mesh after the optimizing process.

Regularize The Connectivity By Edge-Flipping – Regularizes the connectivity of poly-faces by flipping poly-edges to improve the quality of mesh and equalize poly-vertex balance.

Preserve Boundary Poly-Vertices – Preserves the position of poly-vertices around boundaries.

Do Not Modify Sharp Edges – Preserves sharp poly-edges and is useful for optimizing CAD mesh converted from CAD data.



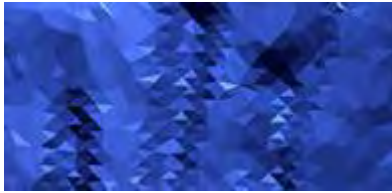
Disable Option

Enable Option

Sharp Edge Angle Criteria – Determines the criterion to judge shape poly-edges by angles. An edge is regarded as being sharp when the angle between two adjacent faces is bigger than a predefined value. Small is set as 40 degrees, large as 100 degrees and the middle default value is set at 70 degrees. Moving the slider towards small will preserve more edges as sharp edges.

Improve Curvature Flow – Improves curvature flow of poly-faces by using curvature information on a mesh and refines feature shapes.

Max. No. Of Iterations – Specifies the maximum number of improving iterations.



Original Data



Improved Curvature Flow

Curvature Sensitivity – Determines the curvature sensitivity. When the sensitivity is set to High, the number of switched poly-edges will increase and sharp edges will be generated.



Original Data



Switched Edges

Equalize Poly-Vertex Balance – Equalizes poly-vertex balance to create poly-faces with uniform poly-edge lengths on mesh.

Note:

Equalized vertex balance is when 6 poly-vertices have been uniformly organized around a center poly-vertex and the forces of their connectivity have been balanced.



Balanced Poly-Vertex

Max. No. Of Iterations – Specifies the maximum number of equalizing iterations.

Smooth – Applies smoothing to a mesh.

Modified Region Only – Applies smoothing to a modified area only.

See Also

[Tools > Mesh Tools > Retriangulate](#)

[Tools > Mesh Tools > Global Remesh](#)

[Tools > Mesh Tools > Edit Poly-Edge](#)

[Tools > Mesh Tools > Virtual Rescan](#)

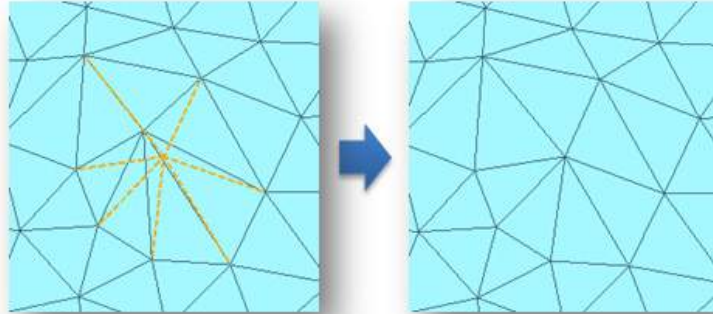
[Tools > Mesh Tools > Rewrap](#)

[Tools > Mesh Tools > Smart Brush](#)

Tools > Mesh Tools > Edit Poly-Edge

What

The **Edit Poly-Edge** command edits poly-edges manually by swapping, merging, subdividing, inserting, and moving poly-vertices.



Why

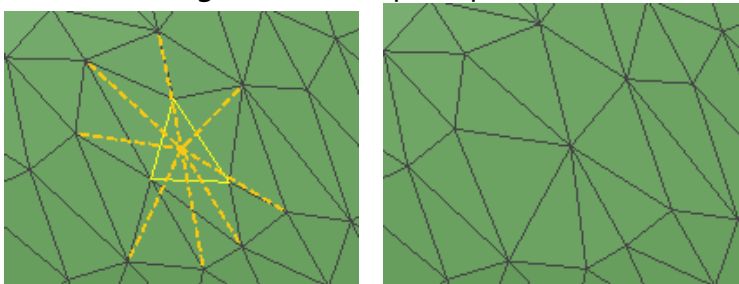
The **Edit Poly-Edge** command is useful for:

- Manually editing poly-edges, poly-vertices, or poly-faces
- Manually improving the topology of poly-faces
- Modifying mesh shapes with a low poly-faces count

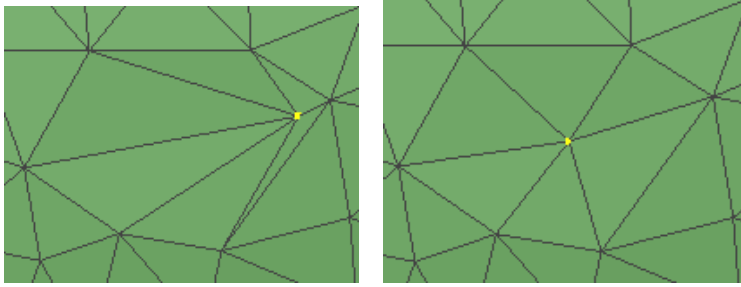
How to Use

How to edit poly-edges on a mesh

1. Go to **Tools > Mesh Tools > Edit Poly-Edge** or click the icon in the toolbar.
2. Choose the **Merge** method and pick a point on a mesh.



3. Choose the **Move** method and drag a poly-vertex on the mesh.



4. Click the **OK** button to complete the command.

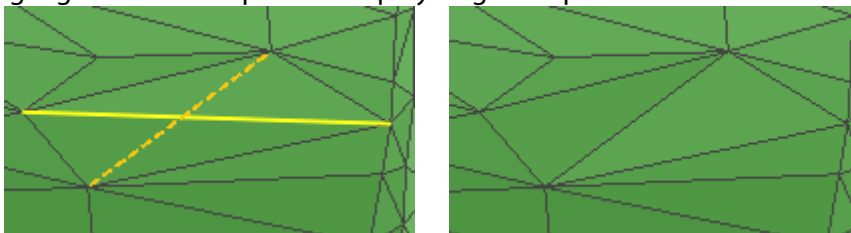
Options in Detail

Methods

This command features 3 methods:

- **Swap**
Flips poly-edges
 - **Merge**
Merges poly-vertices
 - **Subdivide**
Divides poly-faces
 - **Insert**
Inserts a poly-vertex on a poly-face
 - **Move**
Moves a poly-vertex
-

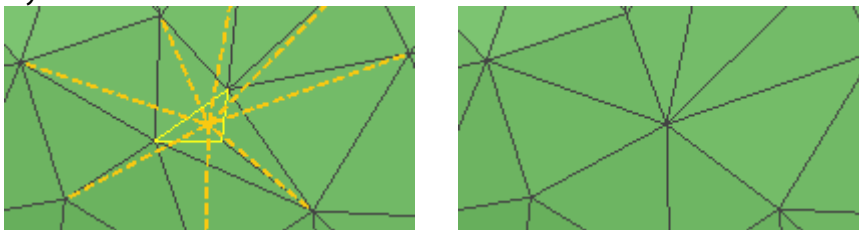
Swap – Flips a poly-edge. When the mouse cursor hovers over poly-edge, a preview will be highlighted. Select a previewed poly-edge to flip it.



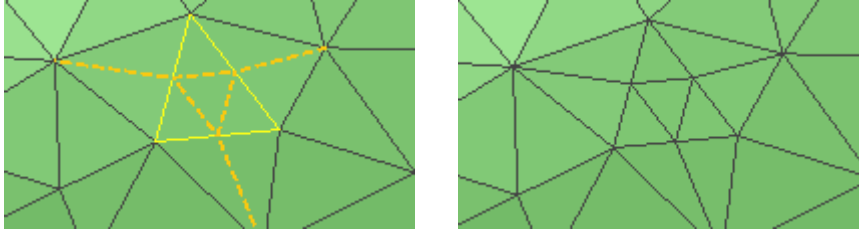
Before Swap

After Swap

Merge – Merges poly-vertices in to a single poly-vertex. This method is useful for removing small poly-faces.



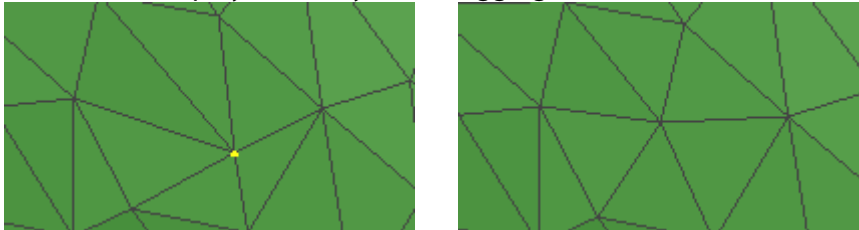
Subdivide – Divides poly-face. This method is useful when dividing large poly-faces.



Insert – Inserts poly-vertex on poly-face. This method divides poly-faces by inserting a poly-vertex.



Move – Moves a poly-vertex by click-dragging.



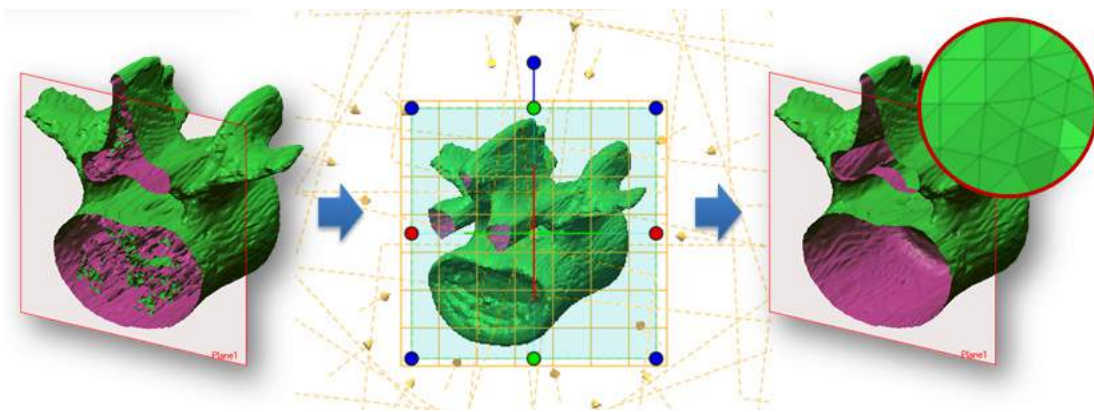
See Also

[Tools > Mesh Tools > Retriangulate](#)
[Tools > Mesh Tools > Global Remesh](#)
[Tools > Mesh Tools > Optimize Mesh](#)
[Tools > Mesh Tools > Virtual Rescan](#)
[Tools > Mesh Tools > Rewrap](#)
[Tools > Mesh Tools > Smart Brush](#)

Tools > Mesh Tools > Virtual Rescan

What

The **Virtual Rescan** command regenerates a mesh by combining feature shapes captured in the normal direction of each virtual plane. This command can be used to clean messy poly-faces inside of a mesh.



Why

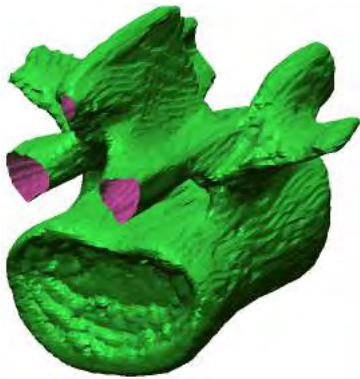
The **Virtual Rescan** command is useful for:

- Extracting exterior poly-faces from a volumetric mesh model that is mixed with inner and outer feature shapes
- Optimizing poly-faces with uniform lengths on a mesh

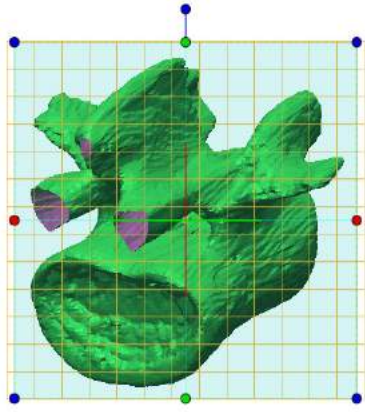
How to Use

How to rescan feature shapes and recreate a mesh

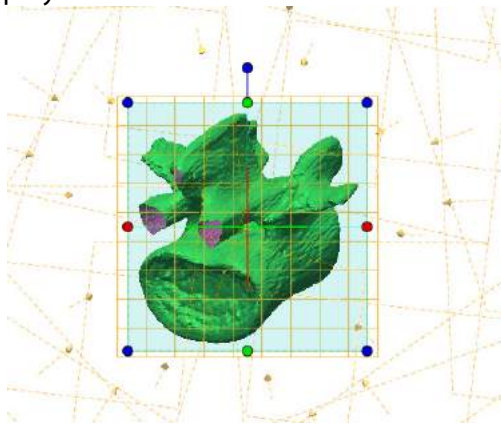
- 1 Align a model in the Model View to define a normal direction of a virtual plane. Go to
 - **Tools > Mesh Tools > Virtual Rescan** or click the icon in the toolbar.



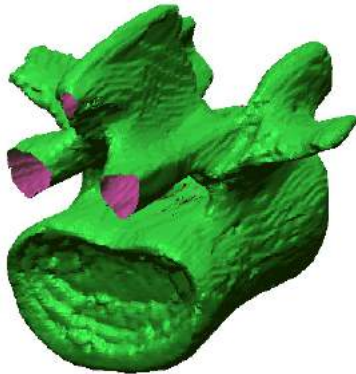
- 2 Check the **Max. Poly-Face Angle** option and set the value to **75°**.
 - Check the **Merge** option to merge feature shapes captured in the normal direction of each virtual plane.
- 2 Set the **Grid U/V Size** to **0.3mm**.
 -



- 3 Use the **Add Polyhedral Position** option to surround the target mesh with virtual planes.
- .
- 4 Change the **Type** to **Icosahedron (20)** and click the **OK** button.
- . Check the normal direction of virtual planes surrounding the target mesh by the defined polyhedral formation.

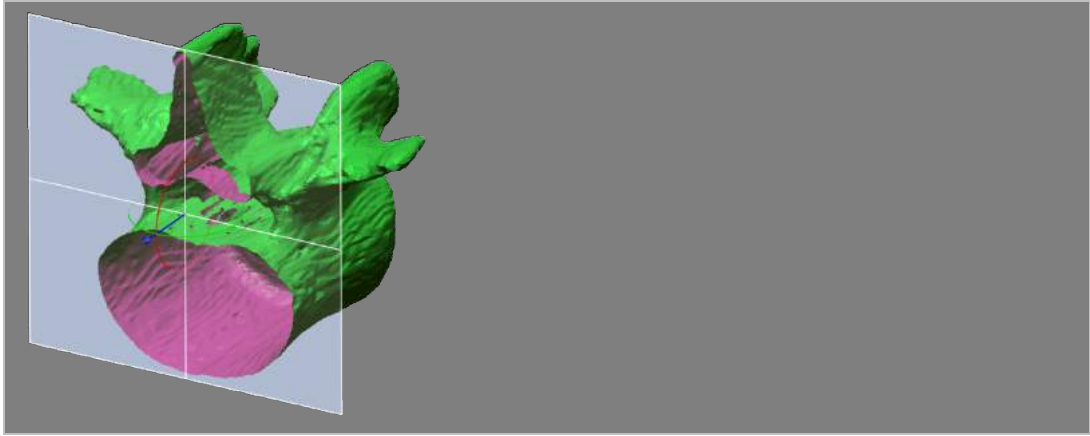


- 5 Click the **OK** button to complete the command.
- .



Note:

To check the inner part of a mesh, use the **View Clip** command. For more information, see [View > Clip](#).



Options in Detail

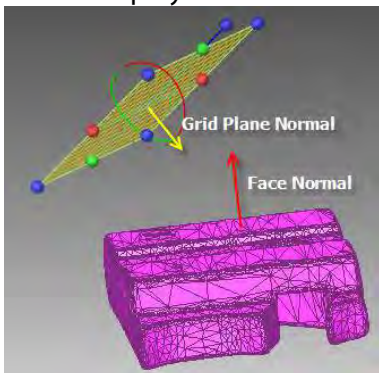
Rescan Options

Poly-Faces – Selects target poly-faces.

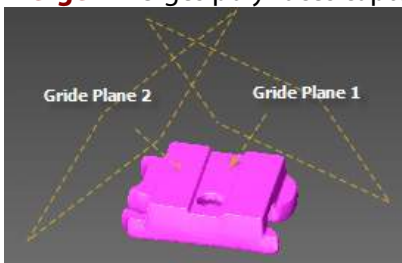
Note:

If poly-faces are not selected, the Virtual Rescan command will be applied to an entire mesh.

Max. Poly-Face Angle – Specifies a maximum poly-face angle between the direction of poly-faces and the normal direction of a virtual plane to avoid capturing slanted areas and creating abnormal poly-faces.

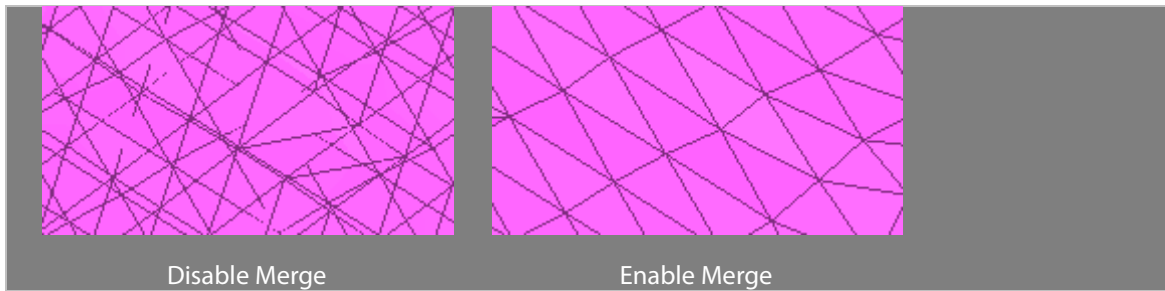


Merge – Merges poly-faces captured in the normal direction of each virtual plane.



Note:

If this option is disabled poly-faces will be combined.



Grid Plane Options

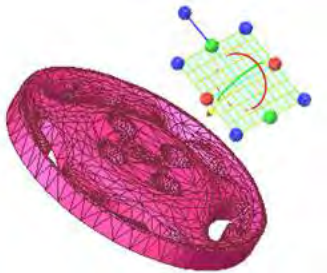
Plane – Specifies a size for **Grid U** and **V**. To

Note:

Click the **Add** button to add more grid planes in a different view direction. To remove existing planes, change the target plane in the Plane list and click the **Delete** button. If the normal direction of an added plane needs to be reversed, click the **Reverse** button. The **Select None** button resets a selected grid plane.

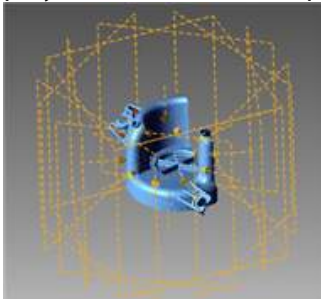
Reference – Defines the normal direction of a virtual plane by referring to the normal direction of an existing Ref.Plane, Sketch, or Face of a body.

Draw – Draws a rectangle on the screen to define a grid plane in a view direction.



Add Cylindrical Position

This option creates a cylindrical formation of virtual planes surrounding a target mesh or selected poly-faces. Check virtual planes and their normal direction by clicking the **Preview** button.



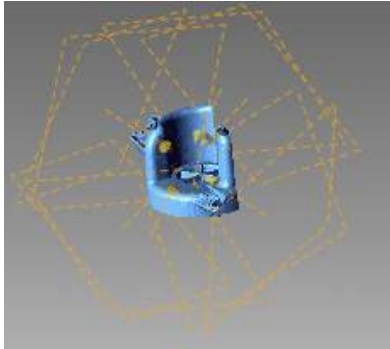
Axis – Specifies the direction of an axis of a cylindrical formation.

No. Of Planes – Specifies the number of virtual planes that cylindrically surround a target mesh or selected poly-faces.

Cap – Caps a feature shape on the top or the bottom of a target mesh or selected poly-faces.

Add Polyhedral Position

Creates a polyhedral formation of virtual planes surrounding a target mesh or selected poly-faces.



Position – Specifies the center position of a polyhedral formation.

Type – Selects a polyhedral formation type.

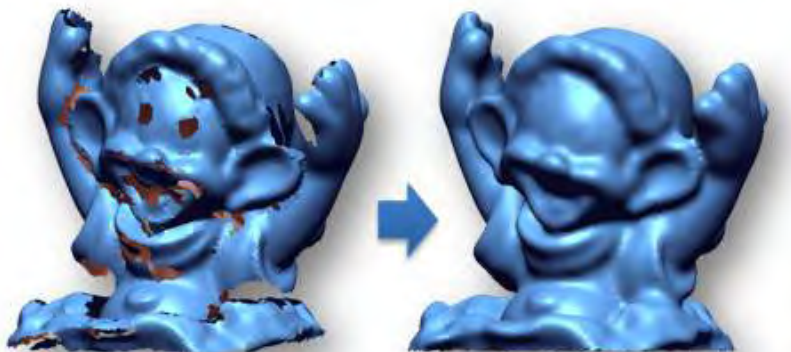
See Also

[Tools > Mesh Tools > Retriangulate](#)
[Tools > Mesh Tools > Edit Poly-Edge](#)

Tools > Mesh Tools > Rewrap

What

The **Rewrap** command fills missing areas on a mesh based on its geometric feature shapes. This command also smooths and improves the quality to a mesh.



Why

The **Rewrap** command is useful for:

- Creating a watertight mesh from raw 3D scan data that has missing areas and complex holes
- Filling holes with poly-faces based on geometric feature shapes and cleaning defects from a mesh at the same time

How to Use

How to rewrap feature shapes and recreate poly-faces on a mesh

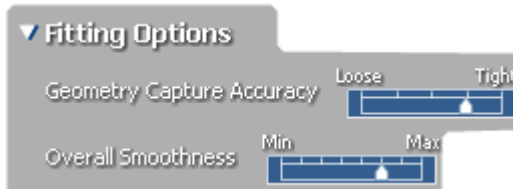
1. Go to **Tools > Mesh Tools > Rewrap** or click the icon in the toolbar.



Note:

Check the normal information of a target mesh before applying the **Rewrap** command. The target mesh should have correct normal information because the normal of poly-faces are used during rewrapping.

2. Increase the **Geometry Capture Accuracy** from **Loose** to **Tight** to tightly fit geometric feature shapes to a mesh and adjust the **Overall Smoothness** option.



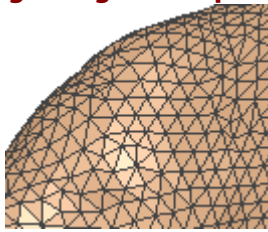
3. Click the **OK** button to complete the command.



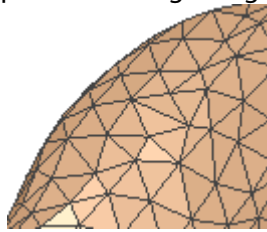
Options in Detail

Current Average Edge Length – Shows the current average edge length of poly-edges on a target mesh.

Edge Length Multiplier – Specifies an edge length multiplier.



Original Edge Length



Edge Length Multiplier: 2

Note:

To create a mesh with dense poly-faces, set a value smaller than 1. If the value is set larger than 1, the results will be more sparse than the original mesh.



Edge Length Multiplier = 1

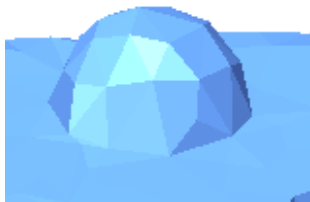


Edge Length Multiplier = 3

Target Average Edge Length – Defines the average length of poly-edges after rewrapping.



Geometry Capture Accuracy – Defines geometry capture accuracy. When the slider is moved towards **Loose**, less poly-faces will be used to create a shape. When the slider is moved towards **Tight**, more poly-faces will be used to create a shape.



Loose



Tight

Overall Smoothness – Applies a smoothing effect to the overall areas of a target mesh. The weight of the smoothness can be adjusted by moving the slider between Min. and Max.



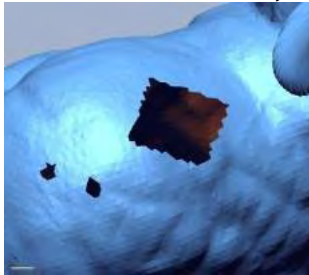
Smooth: Min



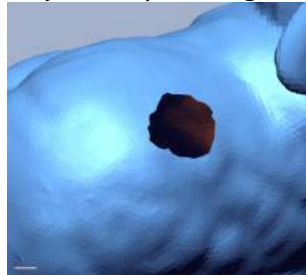
Smooth: 2 Step

More Option

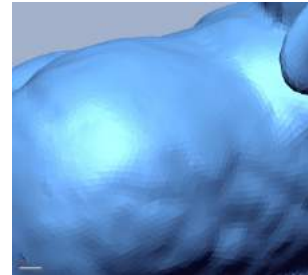
Extend Boundaries – Closes large holes by extending boundaries. The tightness or looseness of the tension in a boundary can be adjusted by moving the slider between Min. and Max.



Original Data



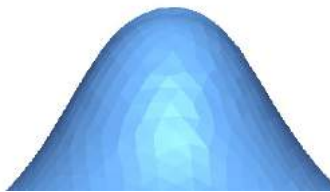
Uncheck Extend Boundary



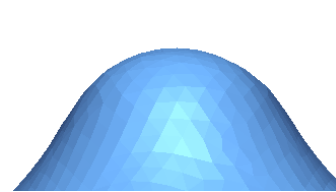
Check Extend Boundary



Original Data



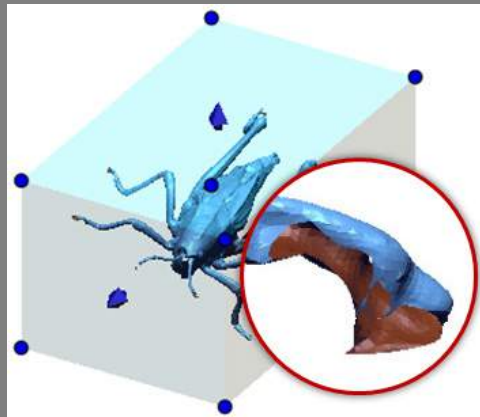
Curved



Flat

Note:

A volume manipulator will be displayed when the Extend Boundaries option is enabled. It limits the extension area by a user-defined volume. This is useful when dealing with a mesh that has boundary shapes opened toward the outside.

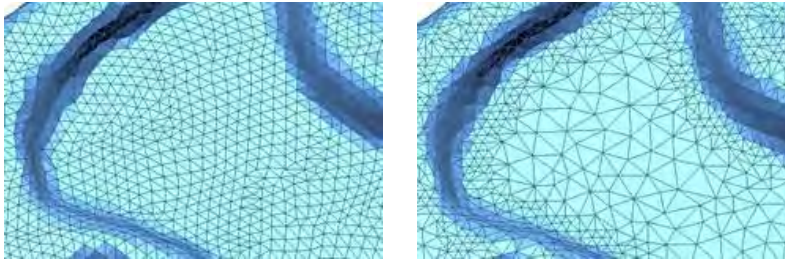


Before



After

Mesh Density In High Curvature Region – Controls the density of poly-faces in high curvature areas.



Uncheck

Check

See Also

[Tools > Mesh Tools > Retriangulate](#)

[Tools > Mesh Tools > Global Remesh](#)

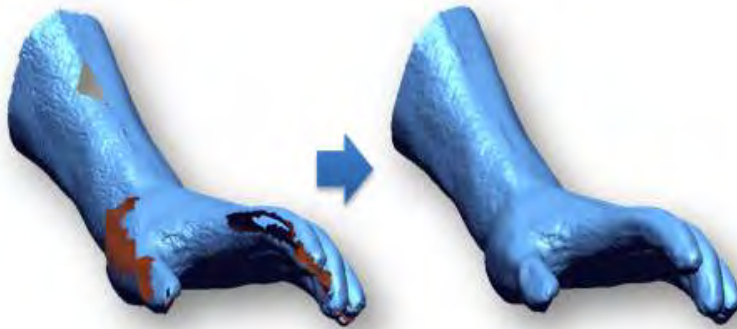
[Tools > Mesh Tools > Optimize Mesh](#)

[Tools > Mesh Tools > Fill Holes](#)

Tools > Mesh Tools > Fill Holes

What

The **Fill Holes** command fills in missing holes with poly-faces based on feature shapes of a mesh. This command also offers advanced editing commands to modify or remove feature shapes from boundaries.



Why

The **Fill Holes** command is useful for:

- Manually filling in holes with poly-faces based on geometric feature shapes

How to Use

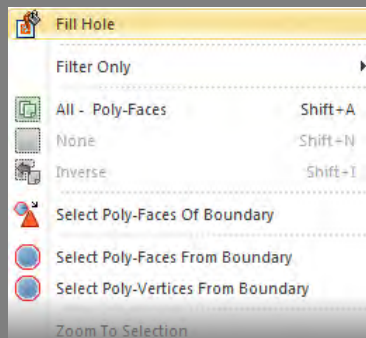
How to fill missing holes in mesh

- 1 Go to **Tools > Mesh Tools > Fill Holes** or click the icon in the toolbar.



Note:

The Fill Hole command can be accessed in the pop-up menu by selecting a boundary and clicking the RMB button. When a hole is filled using the pop-up menu, the Curvature method will be applied.



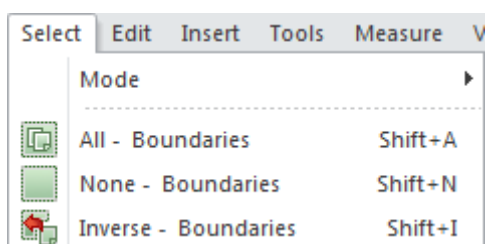
- 2 Select holes and choose the **Curvature** method.
- 3 Click the **OK** button to complete the command.



Options in Detail

Target Entities

Boundaries – Selects boundaries.



Editing Tools

This command features 6 editing commands:

- **Add Bridge**
Adds bridges in between boundaries
 - **Fill Gulf**
Fills poly-faces on indented boundaries
 - **Remove Peninsula**
Removes poly-faces on indented boundaries
 - **Remove Island**
Removes small clusters
 - **Smooth Boundary**
Applies smoothing to boundaries
 - **Delete Poly-Face**
Removes poly-faces
-

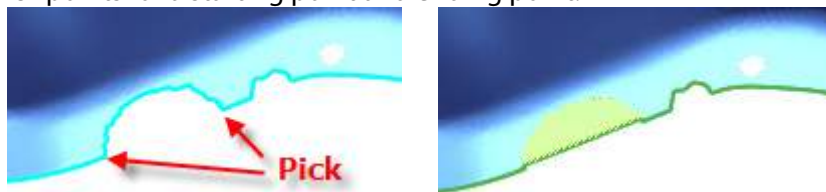
Add Bridge – Adds bridges between target boundaries by connecting two poly-edges. To add a bridge, drag a poly-edge from one side to the other.



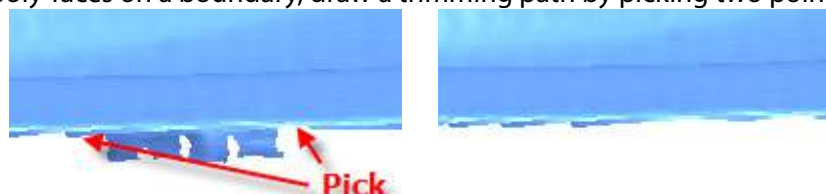
Note:

If a hole is too large and complex to fill with poly-faces, use the **Add Bridge** editing command to make the hole smaller and simpler.

Fill Gulf – Fills poly-faces on an indented boundary. To fill poly-faces on a gulf-like boundary, pick points for a starting point and ending point.



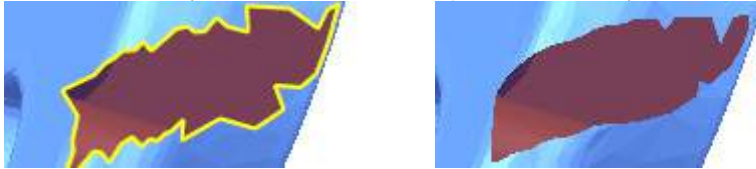
Remove Peninsula – Removes poly-faces on an indented boundary. To remove peninsula-like poly-faces on a boundary, draw a trimming path by picking two points.



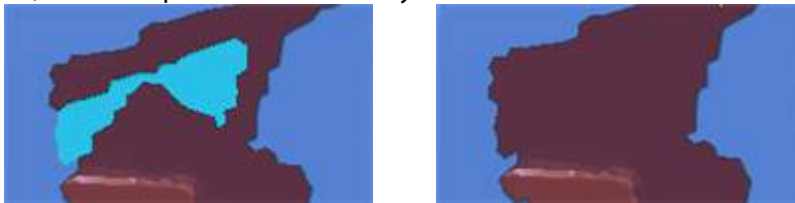
Remove Island – Removes small clusters. To remove small clusters, pick a point in the cluster.



Smooth Boundary – Applies smoothing to a boundary.



Delete Poly-Face – Removes poly-faces. To remove poly-faces, click this button and select poly-faces, and then press the **Delete** key.



Note:

Using the **Delete Poly-Face** command to delete poly-faces will not exit the **Fill Holes** command.

Methods

Flat – Fills target boundaries with flat poly-faces.



Curvature – Fills poly-faces in target boundaries using the curvature flow around the boundary.



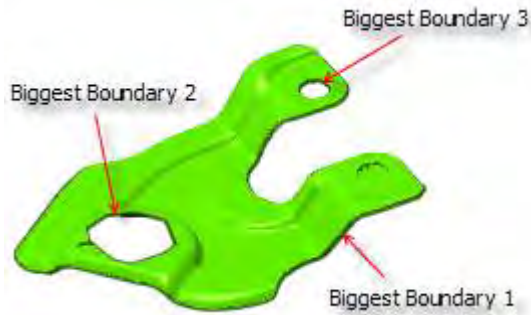
More Options

Hole Size Measure

Boundary Edge Count – Limits filling holes by the boundary edge count.

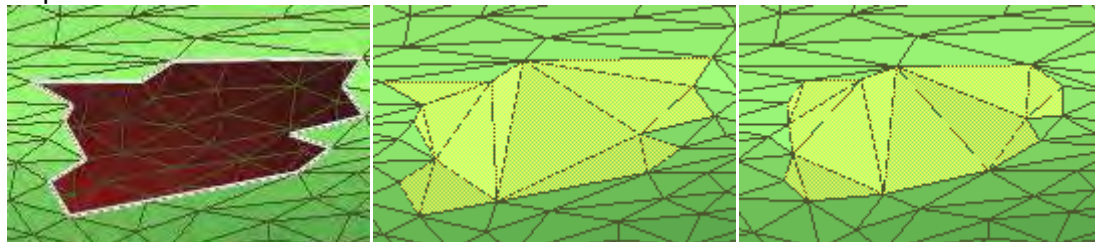
Perimeter – Limits filling holes by the length of boundaries.

Do Not Fill N-Biggest Holes – Preserves large holes. The biggest holes will be preserved in order of size according to a specified value. If the value is set to zero, the option will be ignored.



Do Not Fill If Hole Is Bigger Than – Preserve large holes. Holes with a boundary edge count or length of a boundary less than a specified value will not be filled.

Smooth Boundary – Applies smoothing to target boundaries. This option is useful for reducing noise and the effect of roughness in a target boundary and allows easy filling of poly-faces in complex boundaries.

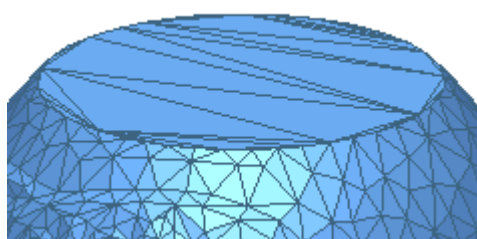


Original Data

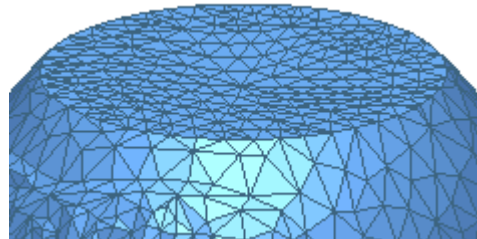
Disable Smooth Boundary

Enable Smooth Boundary

Refine Filled Holes – Refines poly-faces with uniform edge lengths in the filled area within boundary. This option is available when using the **Flat** method.



Without Refine Filled Holes



With Refine Filled Holes

Flatness Of Filled Holes – Adjusts the flatness of filled areas.

See Also

[Tools > Mesh Tools > Edit Boundaries](#)

[Tools > Mesh Tools > Global Remesh](#)

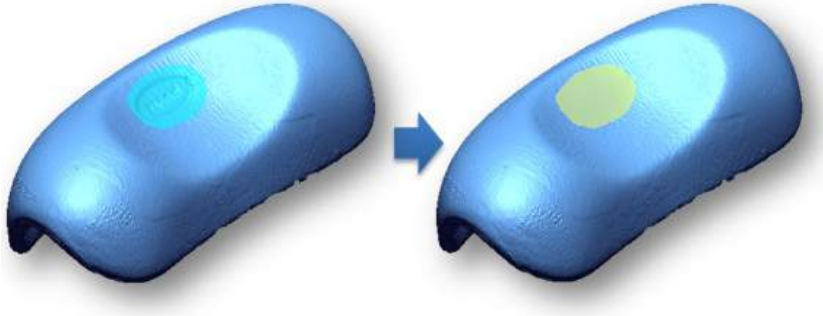
[Tools > Mesh Tools > Optimize Mesh](#)

[Tools > Mesh Tools > Rewrap](#)

Tools > Mesh Tools > Defeature

What

The **Defeature** command removes feature shapes and regenerates poly-faces on a mesh. **Defeature** removes selected areas first and applies the **Fill Holes** operation to the removed areas.



Why

The **Defeature** command is useful for:

- Removing feature shapes from a target mesh
- Removing unnecessary feature shapes and filling in poly-faces in empty areas to smoothly connect with adjacent feature shapes

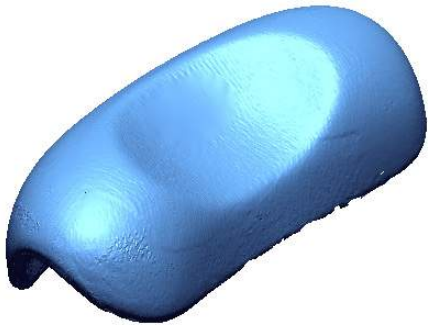
How to Use

How to delete feature shapes and recreate poly-faces on a mesh

1. Go to **Tools > Mesh Tools > Defeature** or click the icon in the toolbar.
2. Choose the **Fill** method. Select poly-faces to remove feature shapes.



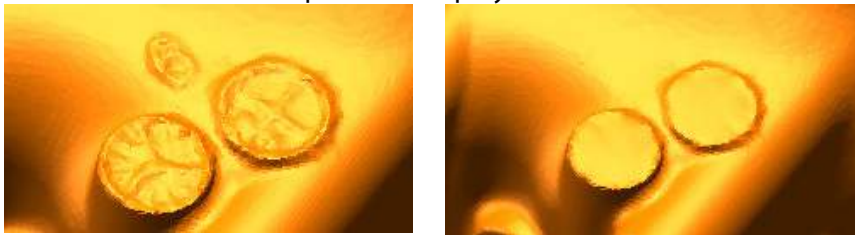
3. Click the **OK** button to complete the command.



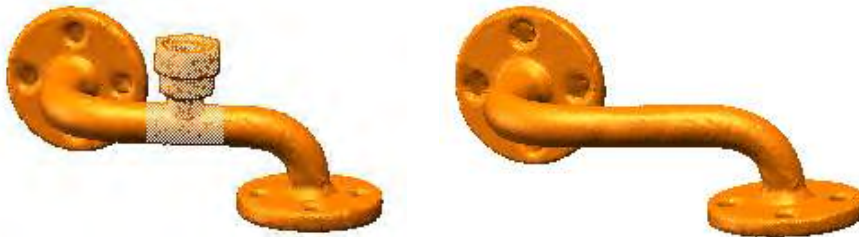
Options in Detail

Target Feature Shape

Fill – Removes feature shapes and fills poly-faces in the removed areas.



Bridge – Removes feature shapes on cylindrical areas and fills poly-faces in the removed areas by connecting with bridges.



Methods

Flat – Fills removed areas using flat poly-faces.

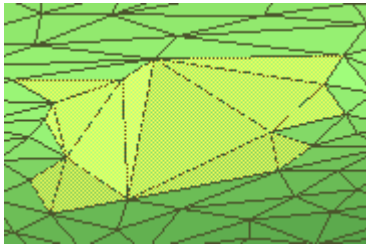


Curvature – Fills removed areas using the curvature flow around a boundary.

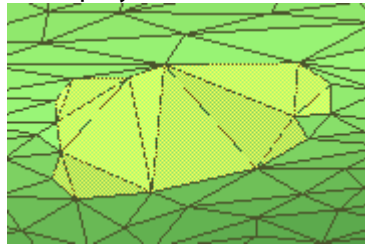


More Options

Smooth Boundary Of Defeatured Area – Applies smoothing to boundaries. This option is useful for reducing the effect of roughness around boundaries that have been manually selected and removed, and easily fills in poly-faces in boundaries.



Disable Smooth Boundary



Enable Smooth Boundary

See Also

[Edit > Delete](#)

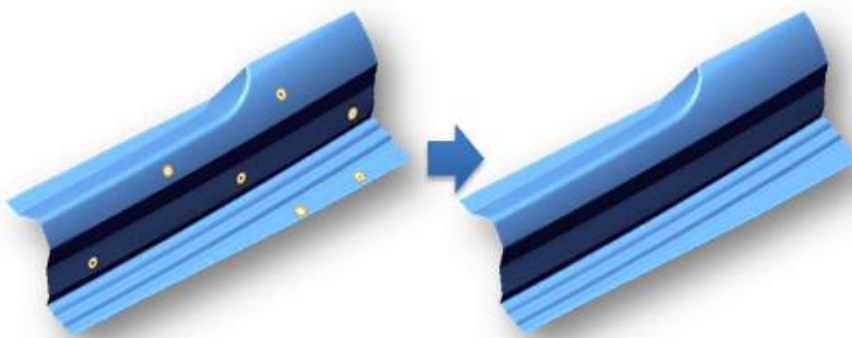
[Tools > Mesh Tools > Retriangulate](#)

[Tools > Mesh Tools > Fill Holes](#)

Tools > Mesh Tools > Remove Marker

What

When scanning an object in different directions, markers are used to recognize feature shapes of 3D scan data and align scans accurately. After aligning and merging scans to mesh, the markers are required to be removed for creating an optimal mesh. The **Remove Marker** command removes scanned markers from a mesh.



Why

The **Remove Marker** command is useful for:

- Finding and removing scan markers

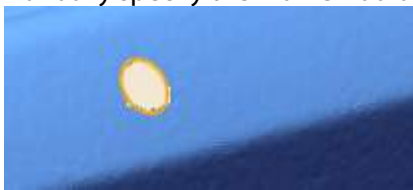
How to Use

How to remove scan markers from a mesh

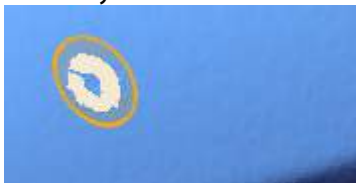
1. Go to **Tools > Mesh Tools > Remove Marker** or click the icon in the toolbar.



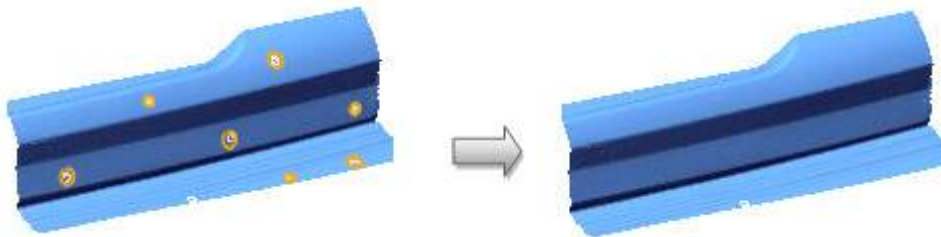
2. Manually specify the marker radius as **10 mm** and then click the **Find Markers** button.



3. Check the detected markers. If markers cannot be located by using **Find Marker** button with a specified marker radius, pick the boundary of a marker on the mesh to select them manually.



4. Click the **OK** button to complete the command.



Options in Detail

Marker Radius – Specifies the radius of a target marker. Click the **Find Markers** button to find markers with the specified radius.

Markers – Shows detected markers in a list. Click the **From File** button to import an AICON file (.obc) that contains center coordinates of markers. This information will be used to find markers in a mesh.

See Also

[Edit > Delete](#)

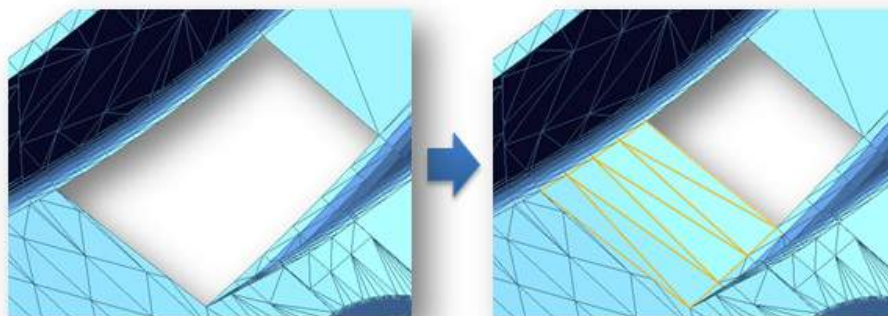
[Tools > Mesh Tools > Fill Holes](#)

[Tools > Mesh Tools > Defeature](#)

Tools > Mesh Tools > Add Poly-Face

What

The **Add Poly-Face** command creates a new poly-face manually by picking poly-vertices on a mesh.



Why

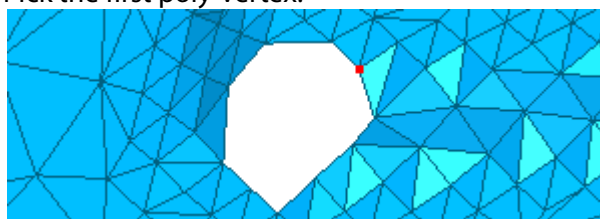
The **Add Poly-Face** command is useful for:

- Manually adding poly-faces
- Filling poly-faces in the boundaries of mesh that has been converted from CAD data

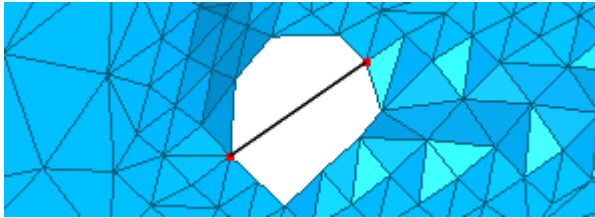
How to Use

How to partially add poly-faces on a mesh

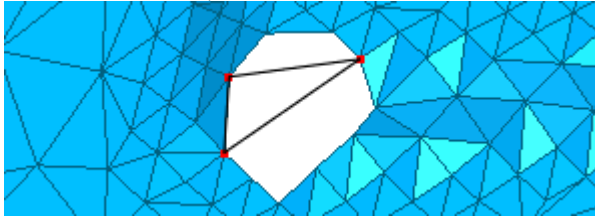
1. Go to **Tools > Mesh Tools > Add Poly-Face** or click the icon in the toolbar.
2. Pick the first poly-vertex.



3. Pick the second poly-vertex.



4. Pick the third poly-vertex



5. Click the **OK** button to complete the command.



Options in Details

Target Entities

Poly-Vertices – Selects three poly-vertices on a mesh. To cancel all selections, click the **Select None** button. The **Deselect Last** button is used to cancel the last selection.

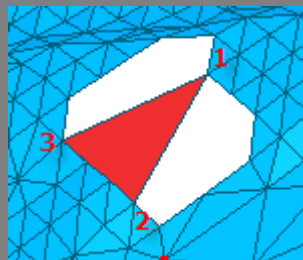
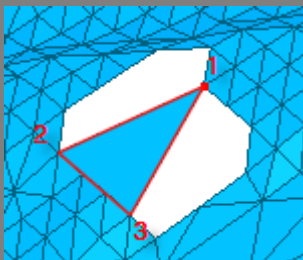
New Poly-Face Count – Shows the number of newly added poly-faces.

More Options

Match Poly-Face Normal – Matches the normal direction of added poly-faces to their adjacent poly-faces.

Note:

The normal direction of created poly-faces follows the order of selection in either the clockwise or counterclockwise directions. To match a normal direction with adjacent poly-faces regardless of selection order, use this option.



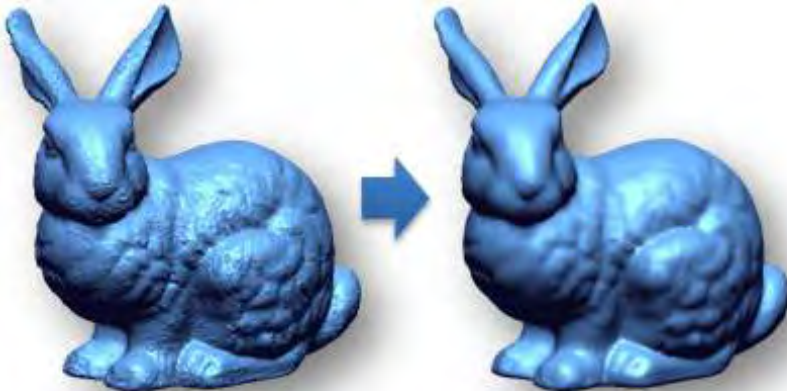
See Also

[Tools > Mesh Tools > Fill Holes](#)

Tools > Mesh Tools > Smooth

What

The **Smooth** command reduces the effect of noise and roughness in a mesh. It can be applied to the entire mesh or partially selected poly-faces.



Why

The **Smooth** command is useful for:

- Reducing the effects of noise and roughness in a target mesh or partially selected poly-faces
- Improving the quality of a mesh

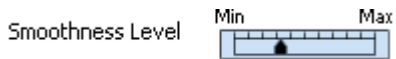
How to Use

How to smooth and reduce the roughness of a mesh

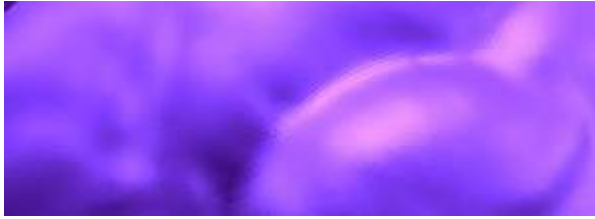
1. Go to **Tools > Mesh Tools > Smooth** or click the icon in the toolbar.



2. Adjust the **Smoothness Level** option to the middle and click the **Next Stage** button to continue.
3. Check the results in real-time by using the **Smoothness Level** option.

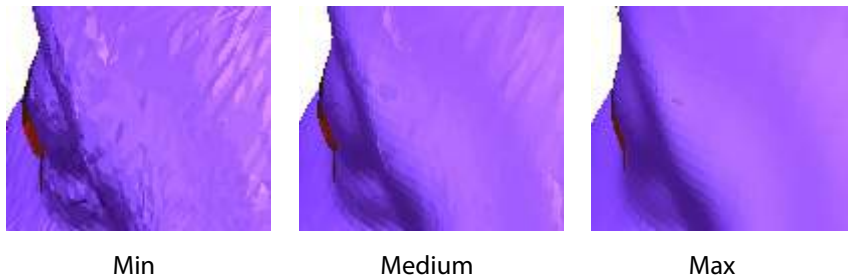


4. Click the **OK** button to complete the command.



Options in Detail

Smoothness Level – Adjust the iteration count of smoothness by moving the slider between **Min** and **Max**.



More Options

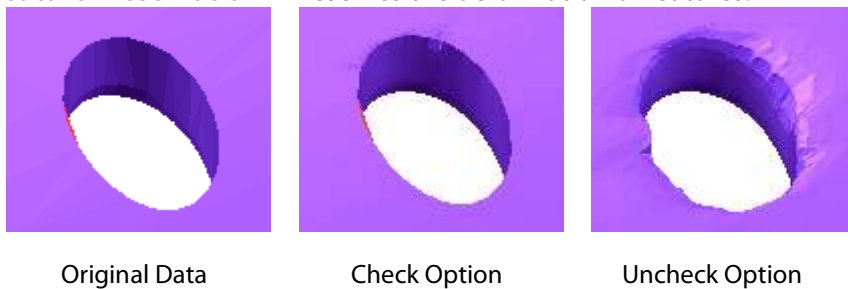
Allowable Deviation – Specifies an allowable deviation range for the deformation of poly-vertices during smoothing.



Do Not Move Boundaries – Preserves the movement of poly-vertices in boundaries.



Feature Preservation – Preserves the deformation of features.



See Also

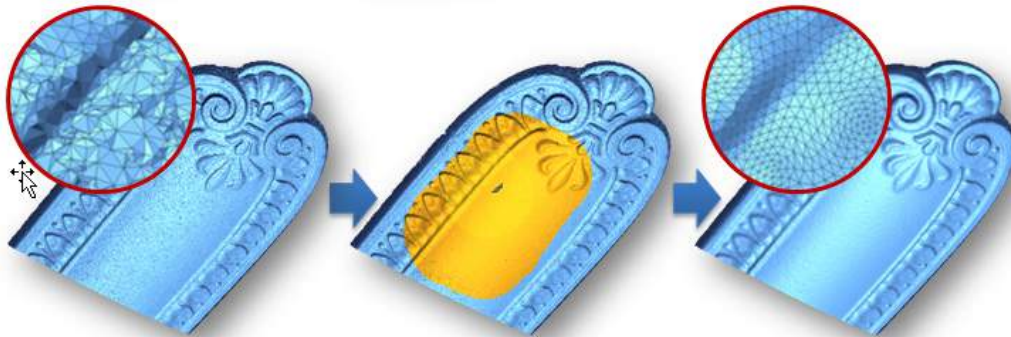
[Tools > Mesh Tools > Smart Brush](#)

[Tools > Mesh Tools > Enhance Shape](#)

Tools > Mesh Tools > Smart Brush

What

The **Smart Brush** command partially optimizes a target mesh and improves the quality of the mesh. This command uses advanced methods to interactively edit partial areas of mesh such as **Smooth**, **Decimate**, **Clean**, **Enhance** and **Deform**.



Why

The **Smart Brush** command is useful for:

- Partially optimizing and improving the quality of mesh
- Reducing the number of poly-faces in a specify area of a mesh
- Partially deforming mesh

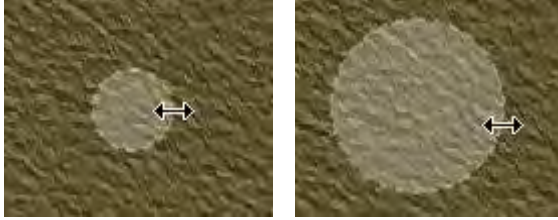
How to Use

A. How to interactively apply a smoothing effect to a partial area of a mesh by using the paint brush selection tool

1. Go to **Tools > Mesh Tools > Smart Brush** or click the icon in the toolbar.



2. Choose the **Smooth** method. Hold down the **Alt** key and drag the mouse cursor with the left mouse button to adjust the range of the editing zone.



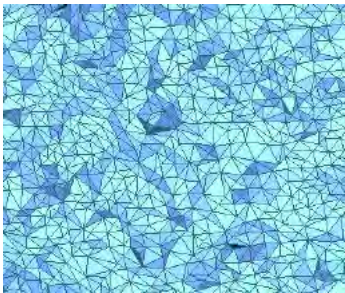
3. Select an area on the mesh and check the results.



4. Click the **OK** button to complete the command.

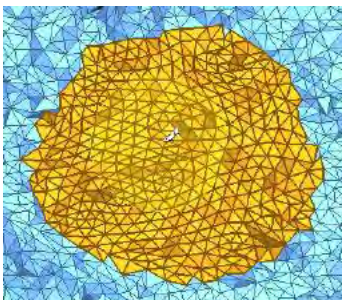
B. How to interactively equalize poly-vertex balance

1. Go to **Tools > Mesh Tools > Smart Brush** or click the icon in the toolbar.



2. Choose the **Clean** method. Hold down the **Alt** key and drag the mouse cursor with the left mouse button to adjust the range of the editing zone.

3. Select an area on the mesh and check the results.



4. Click the **OK** button to complete the command.

Options in Detail

Methods

This command features 5 methods:

- **Smooth**
Reduces the effect of noise and roughness in partial areas of mesh
 - **Decimate**
Reduces the number of poly-faces in partial areas of mesh
 - **Clean**
Equalizes poly-vertex balance
 - **Enhance**
Sharpen sharp areas and smooths rounded corners
 - **Deform**
Deforms feature shapes on a mesh
-

Smooth – Interactively smooths partial areas of a mesh. For more information, see [Tools > Mesh Tools > Smooth](#).



Strength – Adjusts the weight of smoothness by moving the slider between **Min** and **Max**.



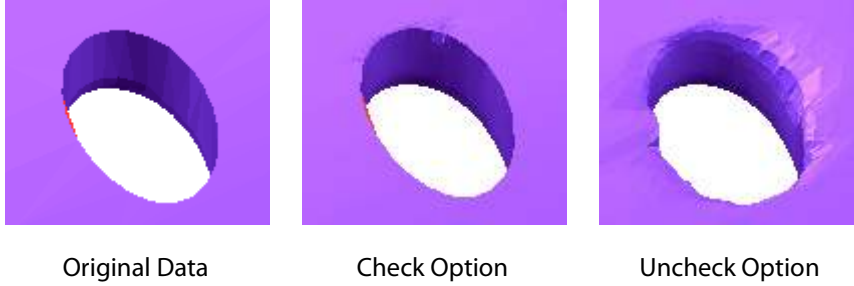
Allowable Deviation – Specifies an allowable deviation range and limits the deformation of poly-vertices within the allowable deviation during smoothing.



Do Not Move Boundaries – Preserve the movement of poly-vertices in boundaries.



Do Not Modify Sharp Edges – Preserve the deformation of poly-vertices in sharp edges.



Decimate – Interactively reduces the number of poly-faces in a partial area of mesh. For more information, see [Tools > Mesh Tools > Decimate](#).



Reduction Level – Adjusts the strength of reduction by moving the slider between **Less** and **More**.

Do Not Move Boundaries – Preserves the movement of poly-vertices in boundaries.

Clean – Interactively equalizes poly-vertex balance by flipping poly-edges. For more information, see [Tools > Mesh Tools > Optimize Mesh](#).



Smooth Cleaned Poly-Faces – Applies smoothing to cleaned poly-faces.

Enhance – Makes sharp areas sharper and rounded corner areas smoother. For more information, see [Tools > Mesh Tools > Enhance](#).



Sharpness – Adjusts the sharpness of selected poly-faces by moving the slider between **Min** and **Max**.

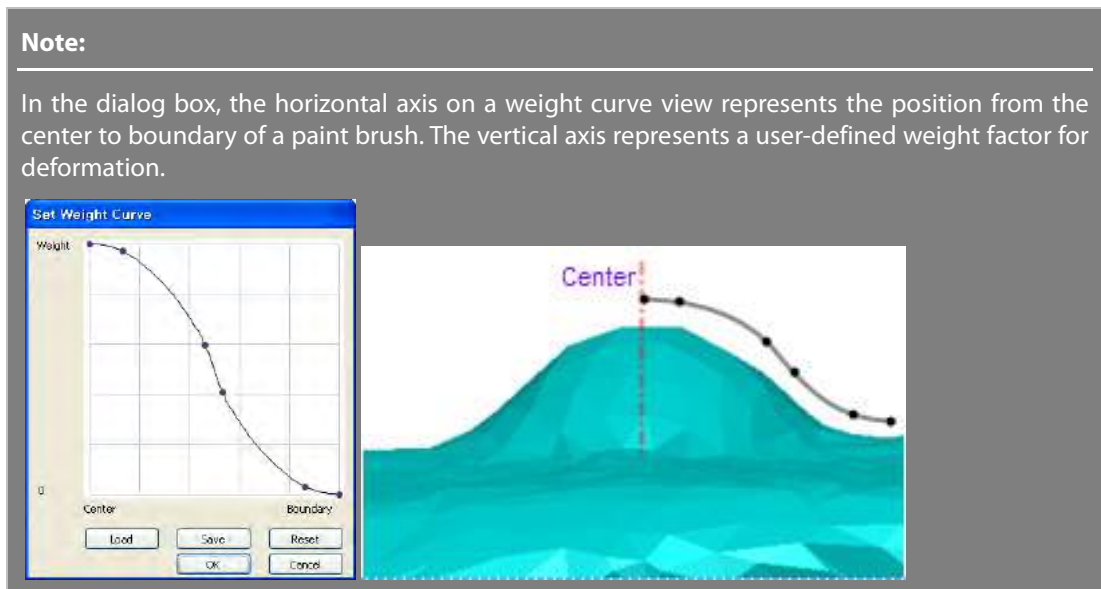
Overall Smoothness – Adjusts the smoothness of selected poly-faces by moving the slider between **Min** and **Max**.

Deform – Deforms feature shapes in a convex or concave shape on a mesh. To create a convex shape, select a partial area on the mesh. To create a concave shape, select a partial area on the mesh while holding down the **Ctrl** key.



Weight – Adjust the weight of deformation by moving the slider between **Min** and **Max**.

Set Weight Curve – Click the **Set Weight Curve** button to modify weight factor in each position of a paint brush.



Smooth Deformed Poly-Faces – Applies smoothing to deformed poly-faces.

Volume of Deformed Poly-faces – Shows the volume of deformed poly-faces.

See Also

[Tools > Mesh Tools > Smooth](#)

[Tools > Mesh Tools > Decimate](#)

[Tools > Mesh Tools > Global Remesh](#)

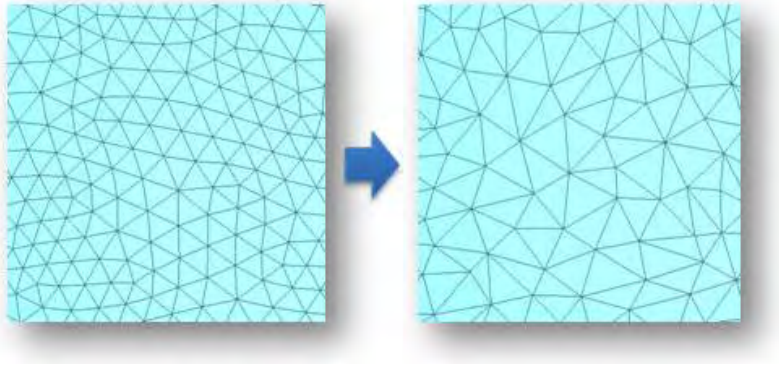
[Tools > Mesh Tools > Optimize Mesh](#)

[Tools > Mesh Tools > Enhance Shape](#)

Tools > Mesh Tools > Decimate

What

The **Decimate** command reduces the number of poly-faces in a mesh or selected areas by merging poly-vertices while preserving geometric feature shapes.



Why

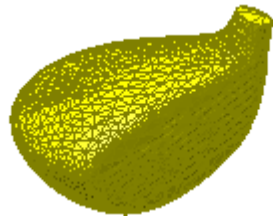
The **Decimate** command is useful for:

- Reducing the number and size of poly-faces on a mesh
- Maintaining the original shape of large meshes in the follow-up modeling process

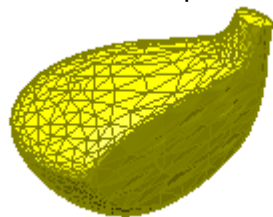
How to Use

How to reduce the number of poly-faces on a mesh

1. Go to **Tools > Scan Tools > Decimate Meshes** or click the icon in the toolbar.



2. Check the **Poly-Face Count** option and set the **Reduction Ratio** to **70 %**.
3. Click the **OK** button to complete the command.



Options in Detail

Methods

This command features 2 methods:

- **Poly-Face Count**

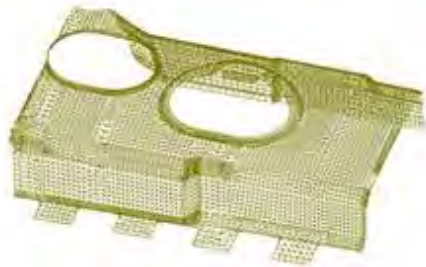
Reduces the number of poly-faces by controlling the target poly-face count

- **Allowable Deviation**

Reduces the number of poly-faces within an allowable deviation

Poly-Face Count – Reduces the number of poly-faces by using parameters specified by the **Target Poly-Face Count** or **Reduction Ratio** options.

Target Poly-Face Count – Determines a target number of poly-faces. This option can only be used with a single mesh.

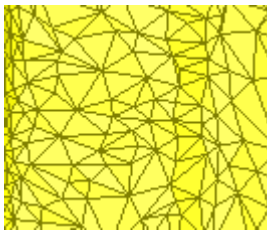


Original Face Count: 14,703

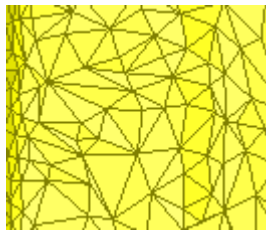


Target Face Count: 5,228

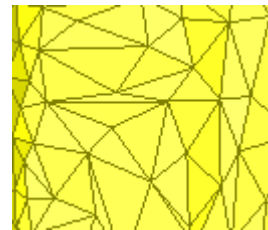
Reduction Ratio – Determines reduction ratio.



Original Data

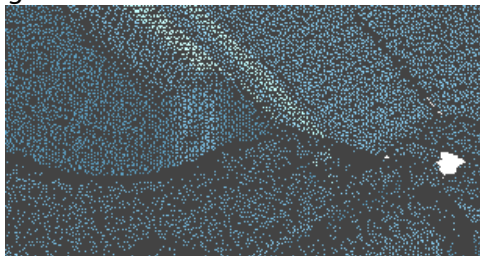


Reduction Ratio: 70%

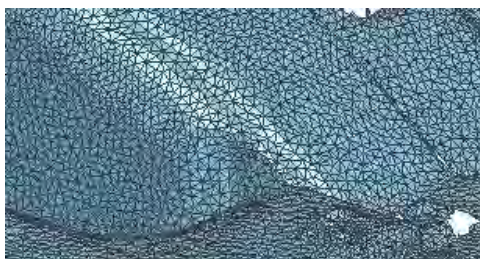


Reduction Ratio: 30%

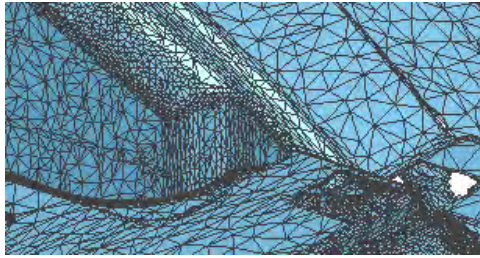
High Curvature Area Resolution – Determines resolution in high curvature areas of a target mesh.



Original Data



High Curvature Area Resolution: Sparse



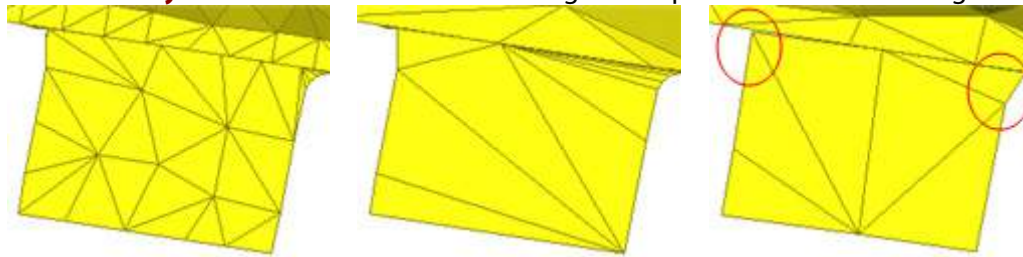
High Curvature Area Resolution: Dense

Allowable Deviation – Reduces the number of poly-faces within an allowable deviation between an original shape and decimated results.

More Options

Do Not Move Poly-Vertices – Preserves the original position of poly-vertices during decimation.

Do Not Modify Boundaries – Preserves the original shape of boundaries during decimation.



Original Data

Check Option

Uncheck Option

Regularize Poly-Face – Regenerates poly-faces with uniform poly-edge lengths.

Note:

The calculation time for this command may be highly increased with this option.

See Also

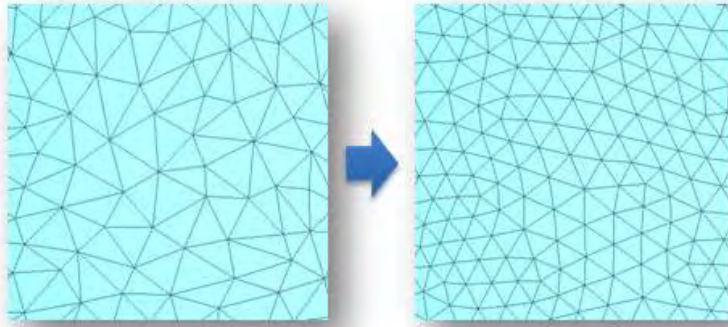
[Tools > Mesh Tools > Smart Brush](#)

[Tools > Mesh Tools > Subdivide](#)

Tools > Mesh Tools > Subdivide

What

The **Subdivide** command subdivides poly-faces to increase the number of poly-faces on a mesh or selected areas while preserving feature shapes. This command also improves the curvature flow between adjacent poly-faces.



Why

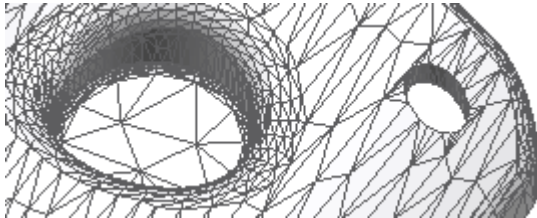
The **Subdivide** command is useful for:

- Increasing the number of poly-faces on a mesh or selected areas
- Reducing the size of poly-faces on a mesh or selected areas
- Characterizing feature shapes
- Working with mesh models that cannot be represented in detail with low poly-face count

How to Use

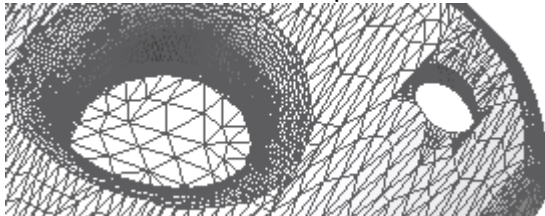
How to divide and increase the number of poly-faces

1. Go to **Tools > Mesh Tools > Subdivide** or click the icon in the toolbar.



2. Choose the **4x** method.

3. Click the **OK** button to complete the command.



Options in Detail

Methods

This command features 3 different methods:

- **4x**
Divides poly-faces into quarters
 - **3x**
Divides poly-faces into three equal parts
 - **Edge Length**
Regenerates poly-faces on a target mesh or selected areas by using specified parameters
-

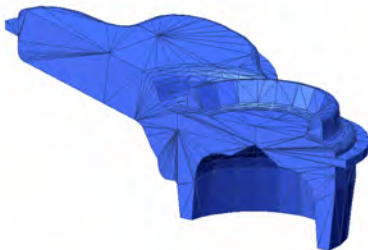
4x – Divides poly-faces into quarters.



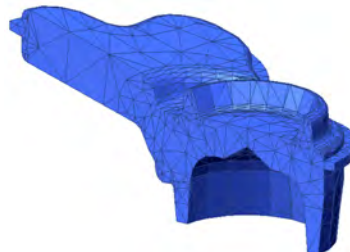
3x – Divides poly-faces into three equal parts.



Edge Length – Regenerates poly-faces on a target mesh or selected areas by using specified parameters. This method manually controls edge lengths and is useful for refining mesh that has been converted from CAD data.



Original Data



Min: 1.0x, Max: 5.0x

Current Average Edge Length – Shows the current average edge length of poly-edges calculated from a target mesh or selected areas.

Min. Edge Length Multiplier – Specifies a minimum edge length multiplier based on the current average edge length.

Min. Edge Length – Specifies a minimum edge length based on the current average edge length.

Max. Edge Length Multiplier – Specifies a maximum edge length multiplier based on the current average edge length.

Max. Edge Length – Specifies a maximum edge length based on the current average edge length.

Note:

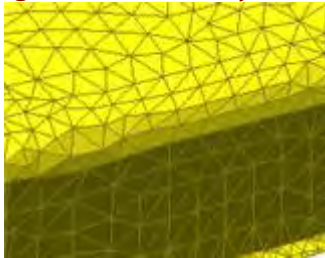
The size of resulting poly-faces will be determined within specified minimum and maximum edge lengths. The edge length of poly-faces can be control by either the **Min./Max. Edge Length Multiplier** or **Min./Max. Edge Length** options.

More Options

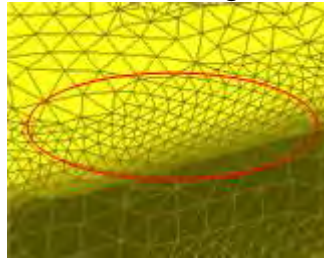
If Edge Length Is Longer Than – Divides poly-faces that have longest edge lengths that are longer than a specified value and can be used when the method is set to **4x**.

Restrict Target Poly-Faces – Restricts subdivision to either high or low curvature areas.

High Curvature Only – Restrict subdivision to high curvature areas.

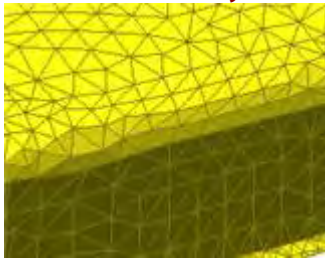


Original Data

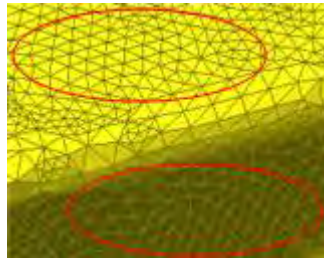


High Curvature Only

Low Curvature Only – Restricts subdivision to low curvature areas.



Original Data

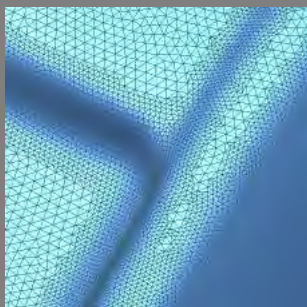


Low Curvature Only

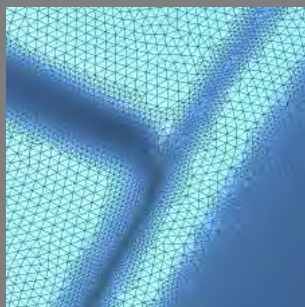
Curvature Searching Sensitivity – Specifies the sensitivity of curvature searching.

Note:

When the slider is moved towards Low, curvature areas will be detected with a wide searching range. When the slider is moved towards High, it will be detected with a narrow searching range.



Low



High

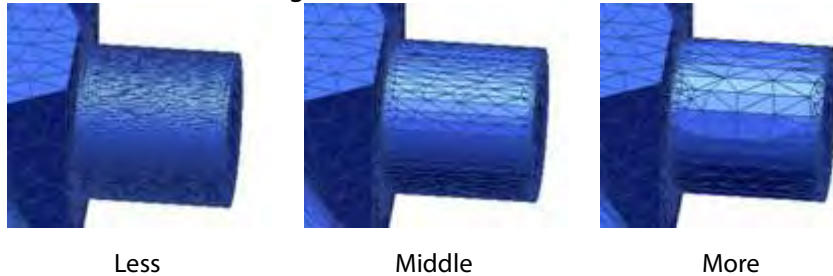
Do Not Move Poly-Vertices – Preserves the original position of poly-vertices during subdivision.

Do Not Modify Boundaries – Preserves the original shape of boundaries during subdivision.

Do Not Modify Sharp Edges – Preserves original sharp edges during subdivision.

Defeaturing Level – Adjust the level of defeaturing by moving the slider between **Less** and **More**.

Feature regions will be classified from a target mesh or selected areas and applied starting from low curvature areas to high curvature areas.



Fill Defective Holes – Refines defective holes and fills holes with poly-faces.

See Also

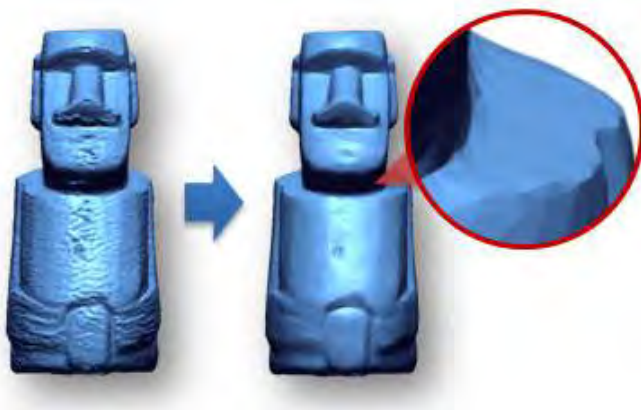
[Tools > Mesh Tools > Smart Brush](#)

[Tools > Mesh Tools > Decimate](#)

Tools > Mesh Tools > Enhance Shape

What

The **Enhance Shape** command enhances feature shapes by characterizing sharp areas and smoothing rounded corner areas.



Why

The **Enhance Shape** command is useful for:

- Enhancing feature shapes by characterizing feature areas
- Improving the quality of a mesh by applying smoothing and sharpening at the same time

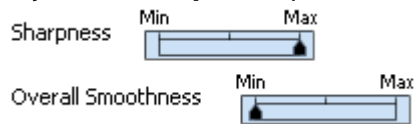
How to Use

How to enhance feature shapes in a mesh

1. Go to **Tools > Mesh Tools > Enhance Shape** or click the icon in the toolbar.



2. Adjust the **Sharpness** option to **Max** and the **Overall Smoothness** option to **Min**.



3. Click the **Next Stage** button to continue.
4. Adjust the **Enhance Level** option. Use the **Enable Realtime Preview** button to check results in real-time.



4. Click the **OK** button to complete the command.

Options in Detail

Sharpness – Adjusts the sharpness of a target mesh or selected poly-faces.



Original Data



Sharpness: Min



Sharpness: Max

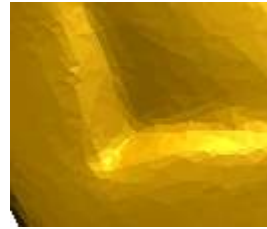
Overall Smoothness – Adjusts the smoothness of a target mesh or selected poly-faces.



Original Data



Overall Smoothness: Min



Overall Smoothness:
Max

Enhance Level – Adjusts the strength of enhancement.



Original Data



Level: Min



Level: Max

See Also

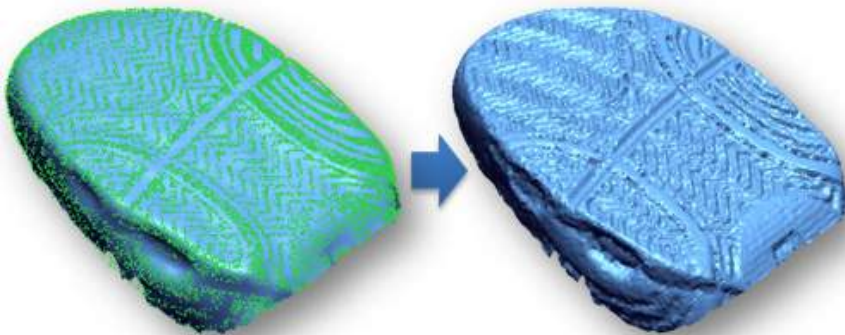
[Tools > Mesh Tools > Smooth](#)

[Tools > Mesh Tools > Smart Brush](#)

Tools > Mesh Tools > Refit

What

The **Refit** command regenerates a mesh using a reference entity. A mesh, a surface body or an analytic shape such as a plane, cylinder, cone, sphere, torus or box, can be a reference entity.



Why

The **Refit** command is useful for:

- Refitting specific areas to an analytic feature shape

- Creating an optimized mesh from irregular point clouds that are too complex to be triangulated
- Refitting a mesh that has been distorted by mesh operations to the original mesh or point cloud
- Refitting a mesh to a modeled surface body

How to Use

A. How to refit distorted mesh to an original mesh

1. Go to **Tools > Mesh Tools > Refit** or click the icon in the toolbar.



2. Select an original mesh as a **Target Entity**.



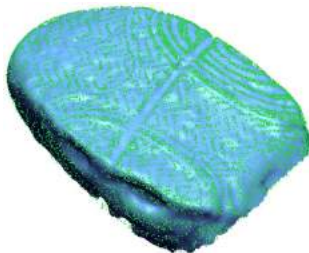
3. Click the **Preview** button to check the results.



4. Click the **OK** button to complete the command.

B. How to refit a mesh to an original point cloud

1. Go to **Tools > Mesh Tools > Refit** or click the icon in the toolbar.



Note:

When a point cloud is too complex to be triangulated, the **Refit** command can be used to create an optimized mesh.

When using the Refit command, prepare a base mesh by following the basic guidelines below:

Clean noise from a point cloud by filling holes and refining poly-faces.

Filter redundant points and sample noise from an original point cloud to create a mesh by using

the Triangulate command with the Mesh Construction method. For more information, see [Tools > Scan Tools > Triangulate](#).

Increase the number of poly-faces to fit to the original point cloud.

After preparing a base mesh, refit the mesh to the original point cloud.

2. Select an original point cloud as the **Target Entity**.



3. Set the Distance to **3mm**.
4. Click the **Preview** button to check the results.



5. Click the **OK** button to complete the command.

Note:

The **Tools > Mesh Tools > Smooth** command can be applied to the refitted mesh to get better results.



Options in Detail

Methods

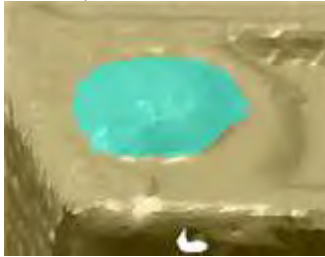
This command features 3 methods:

- **To Analytic Shape**
Refits specific areas to an analytic feature shape
- **To Mesh/Point Cloud**
Refits a mesh to a reference mesh or point cloud

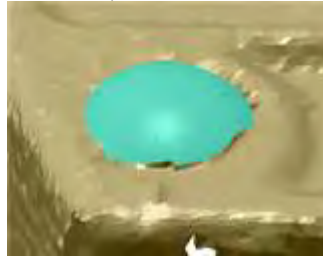
- **To Surface**

Refits a mesh to a reference surface body

To Analytic Shape – Refits selected poly-faces to a specified geometric feature shape.



Original Data



Fitting Shape: Sphere

Poly-Faces To Be Fitted – Selects poly-faces to be fitted.

Fitting Shape – Specifies a geometric feature fitting shape.

Projection On Plane – Specifies a planar entity to be projected onto.

Note:

A planar region, Ref.Plane, 2D sketch chain, curve, 3D sketch, sketch edge, loop, or planar face can be defined as a target plane.

Fix Radius – Specifies the value of the radius of a cylinder or a sphere if it is already known.

User Defined – Specifies a radius to be fitted.

Note:

A user defined radius can be specified when a circular entity is not defined.

Axis Constraint – Constrains the direction of an axis.

Use Specified Axis As Initial Guess – Defines an initial axis that can be used for finding the direction of a fitting shape.

Use Specified Axis As It Is – Manually defines the direction of an axis by selecting an entity that has a vector property or by manually typing the values.

To Mesh/Point Cloud – Refits a mesh to a reference mesh or point cloud.



Before Refit

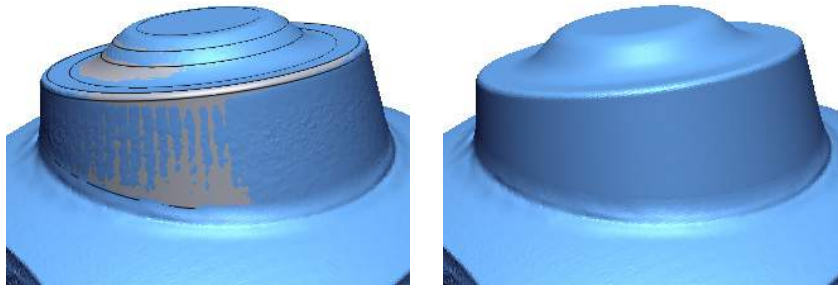


After Refit

Distance Criterion – Specifies the distance criteria used to define the influence range of a

fitting.

To Surface – Refits a mesh to a reference surface body.



Before Refit

After Refit

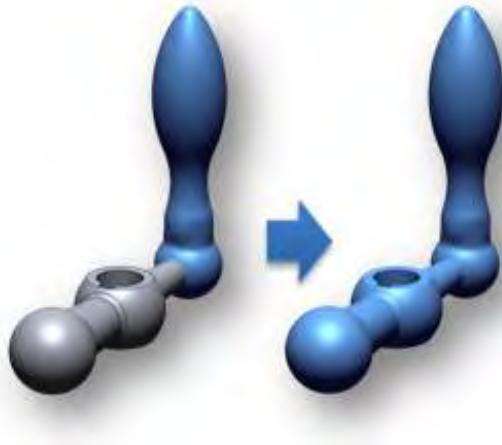
See Also

[Tools > Mesh Tools > Retriangulate](#)
[Tools > Mesh Tools > Edit Poly-Edges](#)

Tools > Mesh Tools > Insert

What

The **Insert** command inserts a mesh converted from body faces, edges or curve entities.



Why

The **Insert** command is useful for:

- Inserting a mesh converted from modeled entities into an existing mesh

How to Use

How to convert other entities and insert into mesh

1. Go to **Tools > Mesh Tools > Insert** or click the icon in the toolbar.
2. Select a solid body for target **Entities**.



3. Click the **Preview** button to check the results.



4. Click **OK** to complete the command.

Options in Detail

Target Entities

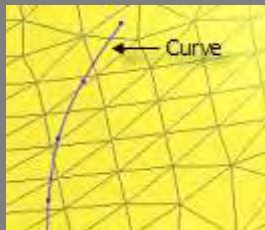
Entities – Selects body faces, edges, or curves to use as target entities to insert into existing mesh.

Note:

If a curve is selected, poly-vertices will be relocated closer to the curve.



Original Data



Insert Curve



Rearrange of poly-vertices

See Also

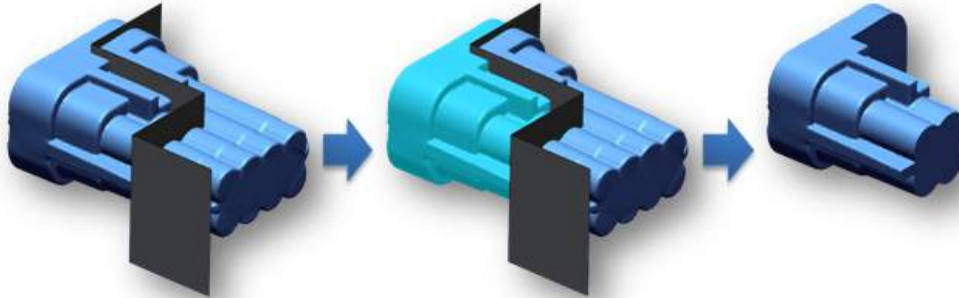
[Tools > Scan Tools > Boolean](#)

[Tools > Mesh Tools > Convert Body](#)

Tools > Mesh Tools > Split

What

The **Split** command divides a mesh into multiple parts by using various tool entities such as a Ref.Plane, curve, face, surface Body, solid Body, or polyline.



Why

The **Split** command is useful for:

- Splitting a mesh into multiple parts

How to Use

How to split a mesh with a tool entity

1. Go to **Tools > Mesh Tools > Split** or click the icon in the toolbar.
2. Choose the **By Entity** method and select a Ref. Plane.



3. Click the **Next Stage** button to continue and select a remaining area.



4. Click **OK** to complete the command.



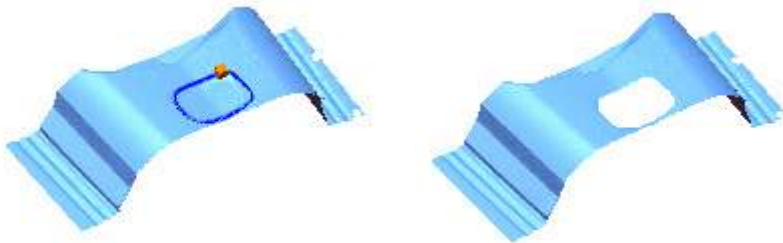
Options in Detail

Methods

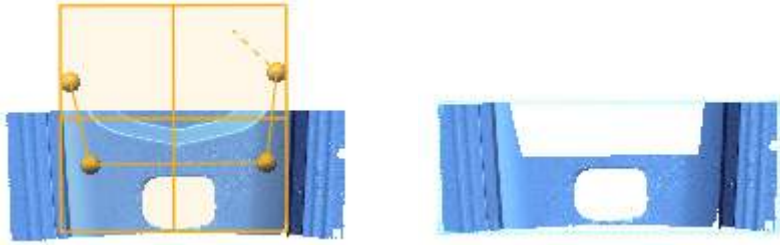
This command features 3 methods:

- **By Entity**
Splits a mesh by using a tool entity
 - **By Drawing Polyline On Screen**
Splits a mesh by drawing a polyline on the screen
 - **By User Defined Plane**
Splits a mesh by a user defined plane
-

By Entity – Splits a mesh by using a plane, curve, or a body face.



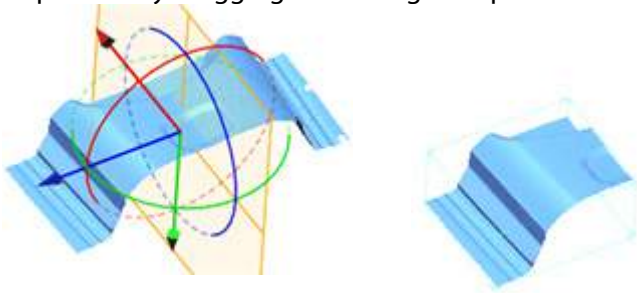
By Drawing Poly-Line On Screen – Splits a mesh by drawing a polyline on the screen.



Note:

Hold the Shift key while drawing to create a horizontal or vertical polyline.

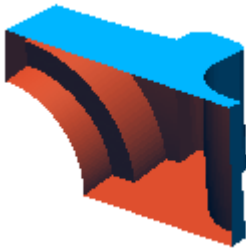
By User Defined Plane – Splits a mesh by using user-defined virtual plane that can be manipulated by dragging or rotating manipulator handles.



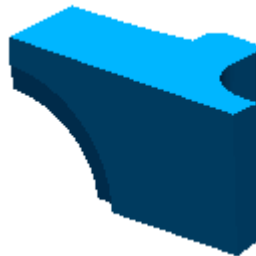
Base Plane – Splits a mesh by a selected base plane.

More Options

Cap Section Ends – Fills an open boundary to create a closed mesh.



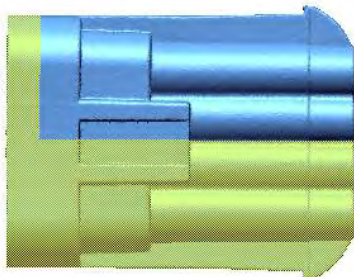
Disable Capping



Enable Capping

Result

Remaining Region – Selects remaining regions.

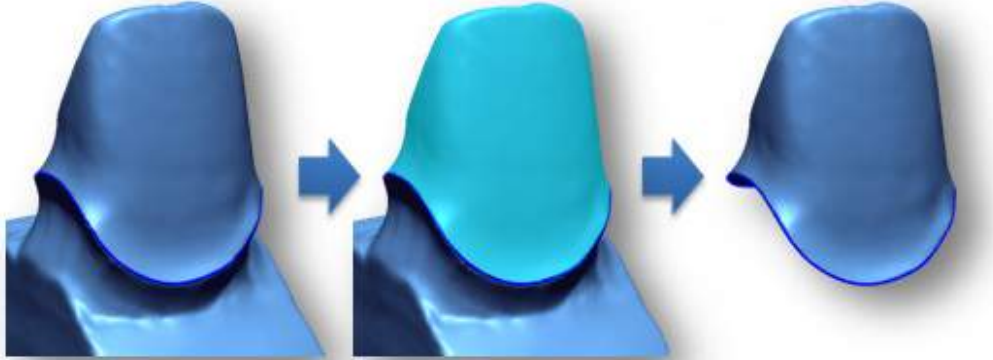


See Also

Tools > Mesh Tools > Trim

What

The **Trim** command trims poly-faces on a target mesh and cuts off trimmed inner or outer regions.



Why

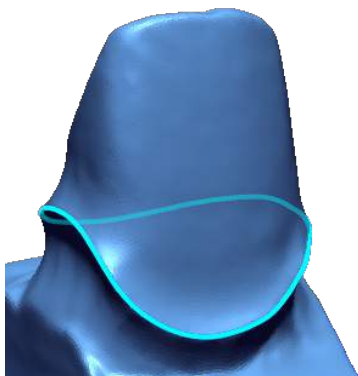
The **Trim** command is useful for:

- Trimming poly-faces out of a mesh
- Rearranging poly-edges to fit a reference curve

How to Use

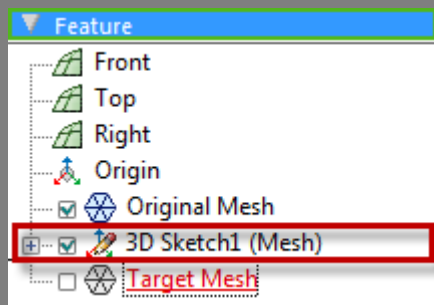
A. How to trim poly-faces on a mesh

1. Go to **Tools > Mesh Tools > Trim** or click the icon in the toolbar.
2. Select the **By Curve** method and select a predefined curve as a tool **Curves**.

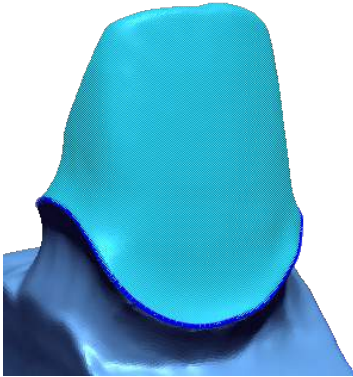


Note:

To use a curve as a tool curve it should be located in the higher level of a feature structure in the Feature Tree.



3. Choose the **Rearrange Poly-Vertices** sub-method.
4. Click the **Next Stage** button.
5. Select the region to remain.



6. Click **OK** to complete the command.

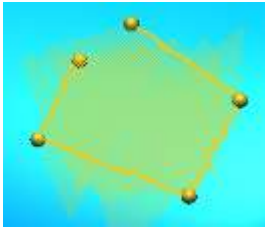


Options in Detail

Methods

By Custom Path – Trims a mesh by a custom path created by picking points on a mesh.

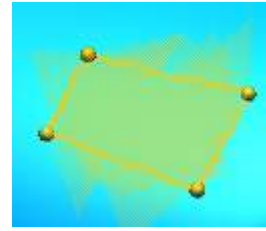
Trim Path – Trims a path using the **Undo Last Path (Ctrl + Z)**, **Close Path**, and **Reset Path** buttons.



Drawing Custom Path



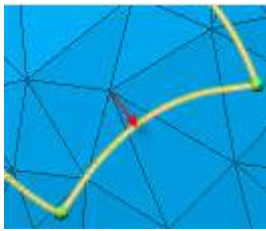
Undo Last Path



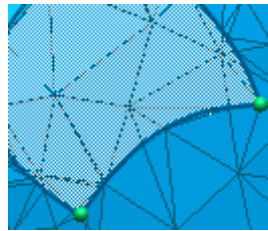
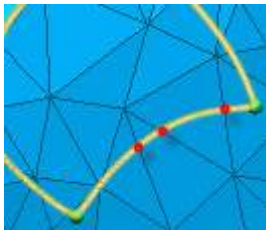
Close Path

By Curve – Trims a mesh using existing closed curves.

Rearrange Poly-Vertices – Trims a mesh by projecting nearby poly-vertices onto selected curves.

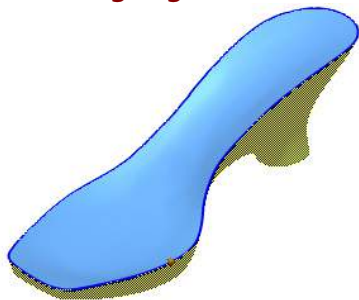


Rearrange Poly-Faces – The trimming will be done by dividing the poly-faces at the intersecting point between the curve and poly-edges.



Result

Remaining Region – Selects remaining regions.



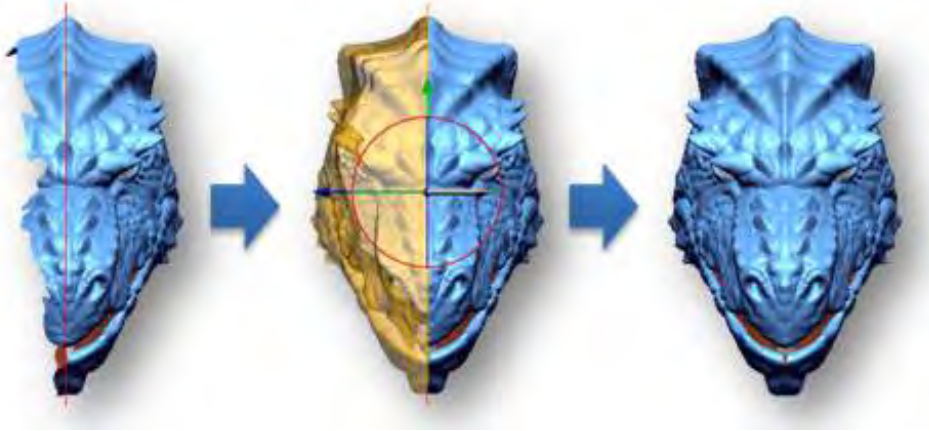
See Also

[Tools > Mesh Tools > Split](#)

Tools > Mesh Tools > Mirror

What

The **Mirror** command creates a new mesh by mirroring an existing mesh using a cutting plane.



Why

The **Mirror** command is useful for:

- Completing a mesh by mirroring one half of a mesh

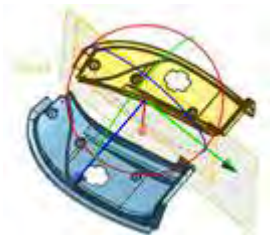
How to Use

A. How to mirror a mesh

1. Go to **Tools > Mesh Tools > Mirror** or click the icon in the toolbar.



2. Select a plane for a mirror plane.



3. Click the **OK** button to complete the command.



Options in Detail

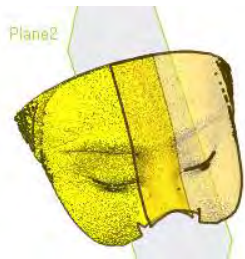
Plane Options

Position – Selects an entity or an x, y and z coordinate set as a position for a mirror plane.

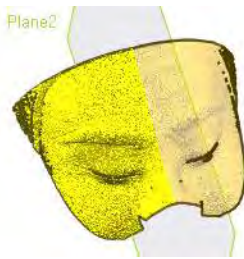
Normal – Specifies the normal direction of a mirror plane.

Remove Original Mesh – Removes original mesh after mirroring.

Trim And Merge – Removes overlapping areas between an original mesh and a mirrored mesh.



Disable Trim And Merge



Enable Trim And Merge

Remesh Poly-Faces Around Mirror Plane – Regenerates poly-faces around a mirror plane to smoothly connect two merged meshes.

See Also

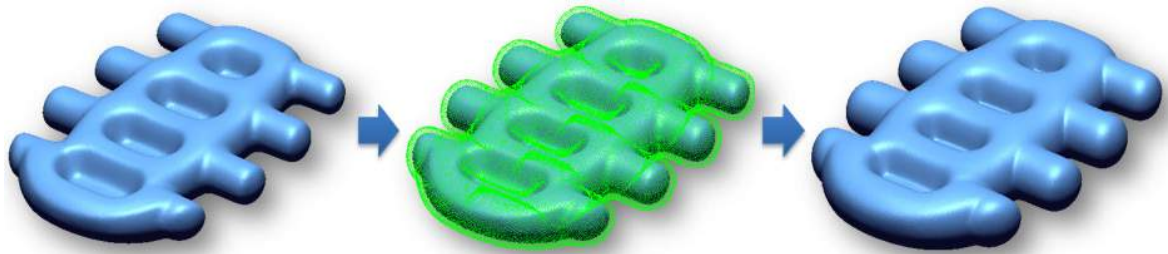
[Tools > Scan Tools > Transform Scan Data](#)

[Tools > Mesh Tools > Split](#)

Tools > Mesh Tools > Offset

What

The **Offset** command creates a new mesh that has a constant distance from an original mesh using either the **Surface** or **Volume** method.



Why

The **Offset** command is useful for:

- Increasing or decreasing the size of a mesh

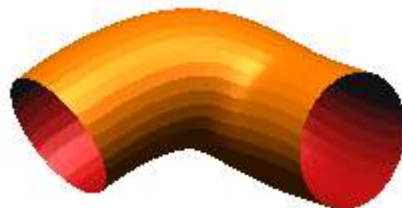
How to Use

A. How to increase the size of mesh by using distance

1. Go to **Tools > Mesh Tools > Offset** or click the icon in the toolbar.



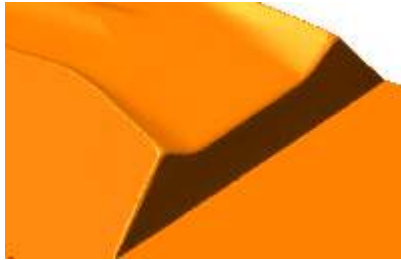
2. Set the **Distance** to **3 mm**.
3. Click **OK** to complete the command.



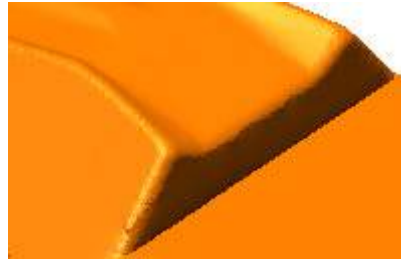
Options in Detail

Methods

Surface – Creates a new mesh that is offset from the original mesh. Although the operation appears to expand or contract the faces, it actually defines new offset faces at a specific distance from the original mesh.



Original Data



Surface Offset

Distance – Specifies an offset distance.

Volume – Calculates the distance from a voxel node to a poly-vertex and makes a new offset polygon using the distance.



Original Data



Volume Offset

Output Resolution Control – Determines the resolution of poly-faces after an offset by specifying a multiplier of the current poly-edge length. This option is available for the **Volume** method.

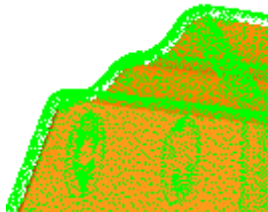
Current Average Edge Length – Shows the resolution of the current average edge length.

Edge Length Multiplier – Multiplies the average edge length by a specified value.

Target Average Edge Length – Defines average edge length by a specified value.

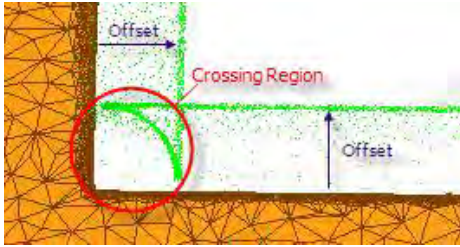
More Options

Preview Offset Result – Displays an offset result as a point cloud.



Preview

Remove Crossing Region – Automatically verifies and cleans crossing regions. This option is available for the **Surface** method.



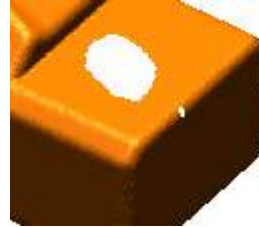
Make Closed Mesh – Closes and offset an open mesh. This option is available for the **Volume** method.



Original Data



With Closed Mesh Option



Without Closed Mesh Option

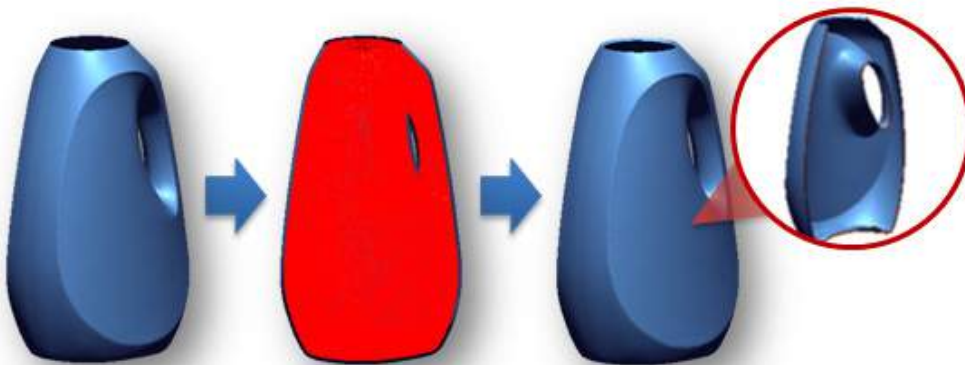
See Also

[Tools > Scan Tools > Transform Scan Data](#)
[Tools > Mesh Tools > Thicken](#)

Tools > Mesh Tools > Thicken

What

The **Thicken** command changes the volume of a mesh by giving a constant thickness to each face to create a completely closed mesh with no boundaries.



Why

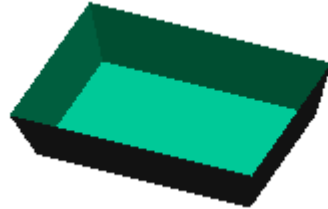
The **Thicken** command is useful for:

- Giving faces with no volume thickness
- Creating constant thickness parts

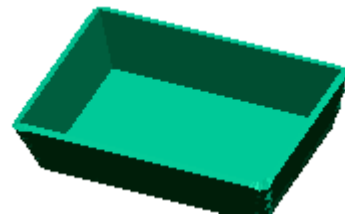
How to Use

A. How to create thickness

1. Go to **Tools > Mesh Tools > Thicken** or click the icon in the toolbar.



2. Set the **Thickness** to **3 mm**.
3. Click **OK** to complete the command.

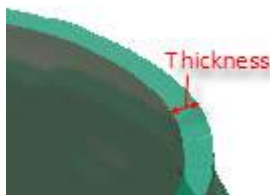


Options in Detail

Methods

Surface – Defines new offset poly-faces that are a specific distance from an original mesh.

Thickness – Specifies an offset distance. To change an offset direction, use the **Direction1** or **Direction2** options. To offset both directions at the same time, use **Both**.



Direction 1 – Thickens in the default positive direction.

Direction 2 – Thickens in the opposite, negative direction from **Direction1**.

Both – Thickens using both **Direction 1** and **Direction 2** at the same time.

Volume – Calculates the distance from a voxel node to a poly-vertex and makes a new

thickened polygon using the distance.

Output Resolution Control – Determines the resolution of poly-faces after an offset by specifying a multiplier of the current poly-edge length. This option is available for the **Volume** method.

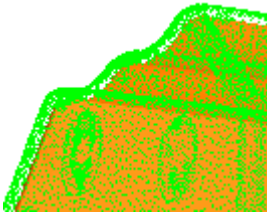
Current Average Edge Length – Shows the resolution of the current average edge length.

Edge Length Multiplier – Multiplies the average edge length by a specified value.

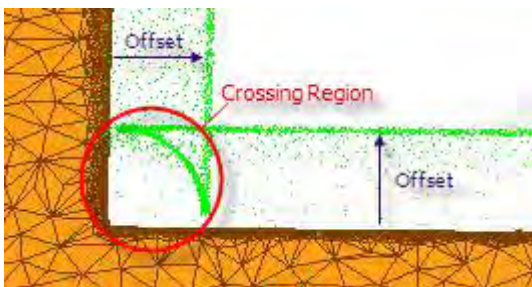
Target Average Edge Length – Defines average edge length by a specified value.

More Options

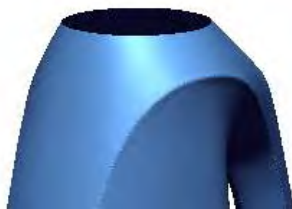
Preview Thicken Result – Displays a thickened result as a point cloud.



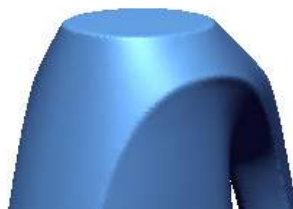
Remove Crossing Region – Automatically verifies and cleans crossing regions. This option is available for the **Surface** method.



Make Closed Mesh – This option is available when you select the **Volume** method. If a mesh is not closed, the application will close and thicken a mesh.



Original Data



Check the Option



Uncheck the Option

See Also

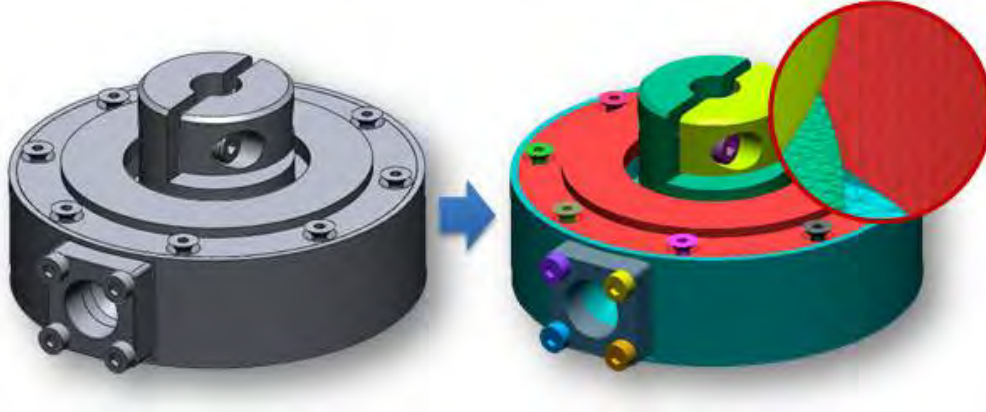
[Tools > Mesh Tools > Fill Holes](#)

[Tools > Mesh Tools > Offset](#)

Tools > Mesh Tools > Convert Body

What

The **Convert Body** command converts a selected solid or surface body to a mesh.



Why

The **Convert Body** command is useful for:

- Converting a solid body into mesh

How to Use

A. How to convert bodies into mesh

1. Go to **Insert > Mesh** or click the icon in the toolbar to create a mesh feature.



2. Click **Tools > Mesh Tools > Convert Body** or click the icon in the toolbar.
3. Select a body to convert.



4. Click **OK** to complete the command.



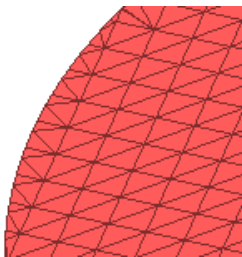
Options in Detail

Target Entities

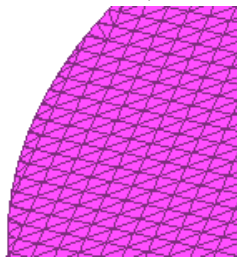
Bodies – Selects solid or surface bodies to convert into meshes.

Options

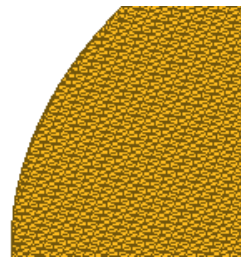
Quality – Determines the accuracy of a mesh. Moving the slider towards **High**, will produce a more accurate mesh that contains more poly-faces.



Quality: Low



Quality: Medium



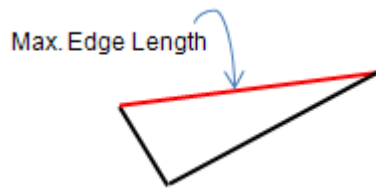
Quality: High

Advanced – Displays sub-options used to precisely limit the size of poly-faces.

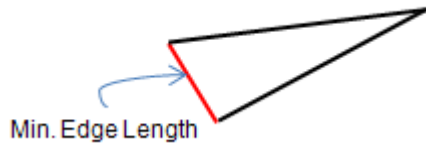
Max. Chord Height – Specifies how accurately a mesh represents a surface by setting the distance between the mesh and the part of the surface it represents.



Max. Edge Length – Specifies the value of the maximum edge length of a poly-face.



Min. Edge Length – Specifies the value of the minimum edge length of a poly-face.



More Options

Sew Boundary – Removes small gaps along boundaries.

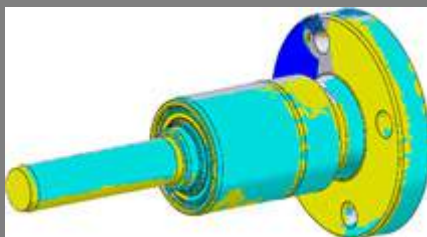


Include Accuracy Analyzer Color – Generates colored mesh from bodies. This option is available when selected bodies have **Accuracy Analyzer™** information.

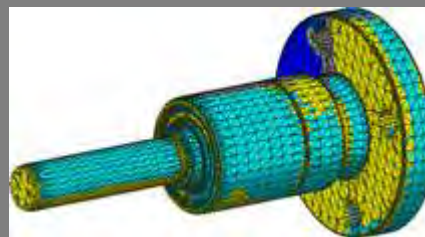
Note:

To use this option:

1. Run **Deviation** in the **Accuracy Analyzer** panel in the **Default** mode.
2. Go to **Insert > Mesh**.
3. Execute the **Tools > Mesh Tools > Convert Body** command.
4. Select a body.
5. Check the **Include Accuracy Analyzer Color** option.
6. Click **OK** to complete the command.



Deviation Result



Colored Mesh

See Also

[Tools > Point Cloud Tools > Convert Entities](#)

7.5.3. Tools > Region Tools

The **Region Tools** menu features various commands used to generate and edit feature regions on a mesh. Geometry feature regions can be classified with region tools and created feature regions can be used for creating feature shapes.

The **Region Tools** are divided into 3 tool groups used to:

Creating Feature Regions

- **Auto Segment**
Automatically segments feature regions on a mesh
- **Resegment**
Resegments existing feature regions by using different classification criteria

Editing Feature Regions

- **Merge**
Merges multiple feature regions into a single feature region
- **Split**
Divides feature regions into multiple parts
- **Insert**
Inserts a new feature region

Managing Feature Regions

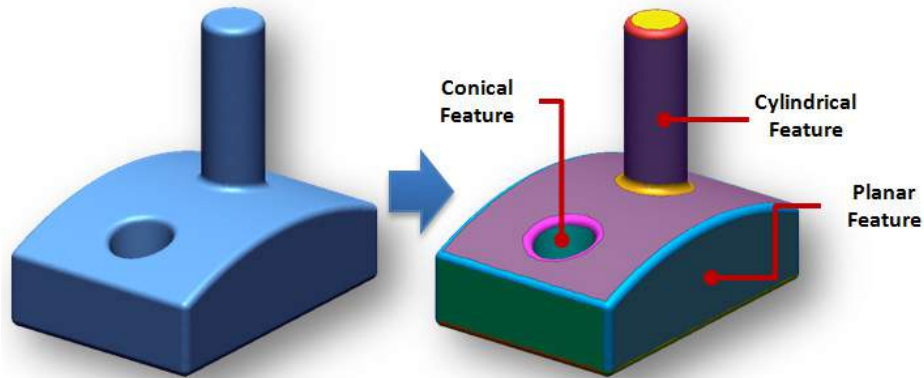
- **Separate**
Separates a feature region into multiple feature regions
- **Enlarge**
Enlarges feature regions
- **Shrink**
Shrinks feature regions

Tools > Region Tools > Auto Segment

What

The **Auto Segment** command automatically classifies feature regions by recognizing 3D features

from raw scan data. Classified feature regions have geometric feature information that can be used for creating features quickly.



Why

The **Auto Segment** command is useful for:

- Classify feature regions on a mesh
- Using geometric feature information for easily and quickly creating features

How to Use

How to automatically segment feature regions on a mesh

- 1 Go to **Insert > Region Group** or click the icon in the toolbar to enter **Region Group** mode.

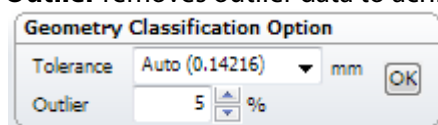


- 2 The **Auto Segment** command will automatically initiate.

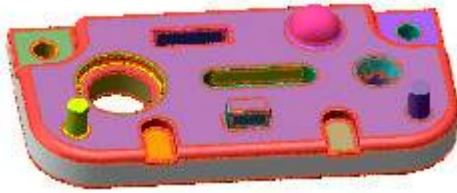
Note:

If feature regions have been classified, manually go to **Tools > Region Tools > Auto Segment** or click the icon in the toolbar to execute the Auto Segment command.

- 3 While the Auto Segment command is executed, specify **Geometry Classification Option**.
- **Tolerance** is used to define the geometry type.
- **Outlier** removes outlier data to achieve more accurate fitting results.

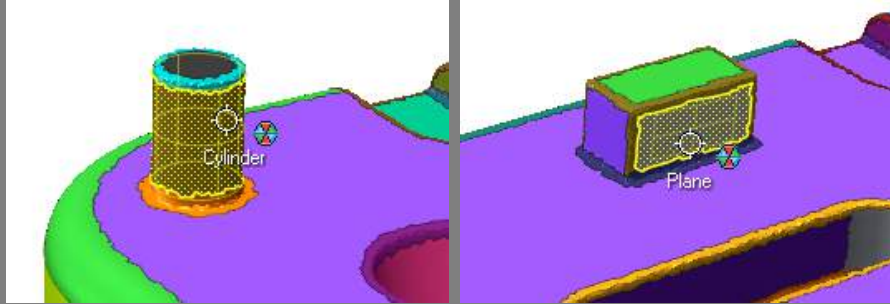


- 4 Click the **OK** button to classify the feature regions on the mesh.



Note:

After classifying regions, check the geometry types on the regions by hovering the mouse cursor over the regions and the geometry type will be displayed. To select regions only, toggle the Filter Regions button in the Select Filter toolbar.

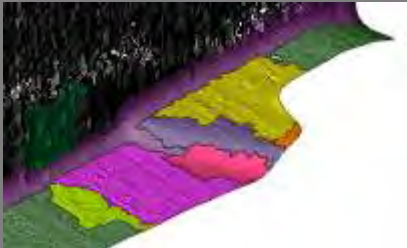


Options in Detail

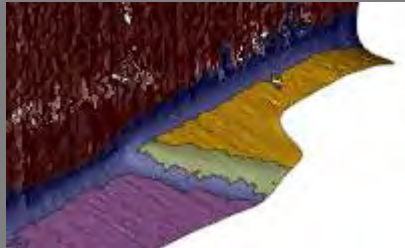
Sensitivity – Specifies search sensitivity for feature regions.

Note:


When the slider is moved towards **More**, search sensitivity will increase. A higher search sensitivity may generate many particle regions if a mesh is rough.

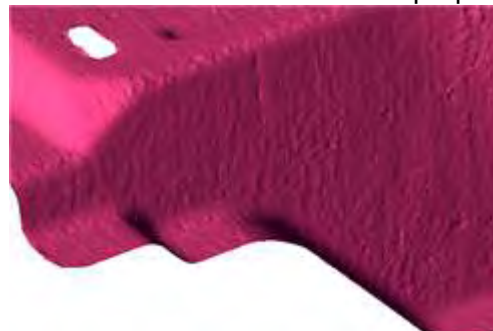


Sensitivity: 100



Sensitivity: 20

Mesh Roughness – Adjusts the roughness of a mesh depending on the noise level. If you click the **Estimate** button , the application analyzes the mesh and recommends a proper value.



Preserve Existing Region – Preserves existing regions.

Note:

If feature regions have been classified, the existing regions will be preserved using this option. This option is useful for managing the freeform and feature parts of a mesh.

Select poly-faces on freeform areas by using a selection mode and manually insert new feature regions by going to Tools > Region Tools > Insert or click the icon in the toolbar. Apply the Auto Segment command to the mesh and the pre-classified freeform regions will be preserved and feature regions will be classified on feature parts only.

See Also

[Tools > Region Tools > Resegment](#)

[Tools > Region Tools > Merge](#)

[Tools > Region Tools > Split](#)

[Tools > Region Tools > Insert](#)

[Tools > Region Tools > Separate](#)

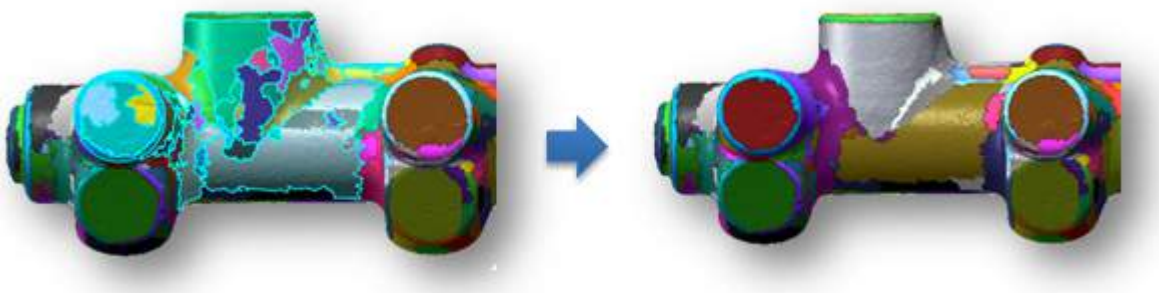
[Tools > Region Tools > Enlarge](#)

[Tools > Region Tools > Shrink](#)

Tools > Region Tools > Resegment

What

The **Resegment** command removes selected regions then classifies them again. Sometimes the region segmentation criteria should have a different value from the size or detail of features in a model. Generally the Resegment command applies to partial areas after Auto Segmenting, which classifies overall areas.



Why

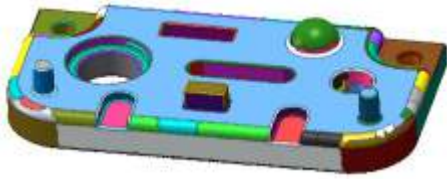
The **Resegment** command is useful for:

- Accurately classifying geometries on a region

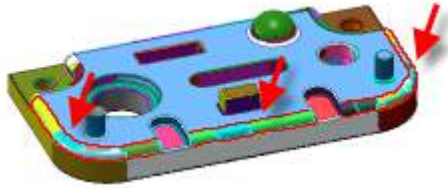
How to Use

How to resegment feature regions on a mesh

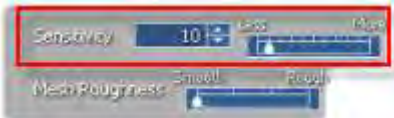
- 1 Go to **Tools > Region Tools > Resegment** or click the icon in the toolbar.



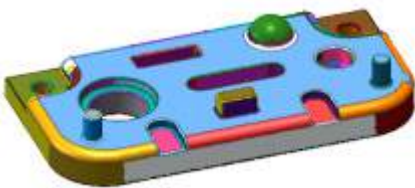
- 2 Select regions to resegment.



- 3 Set the **Sensitivity** and **Mesh Roughness** options. In this example, regions should be simplified. So move the Sensitivity slider to Less.



- 4 Click **OK** to complete the command.

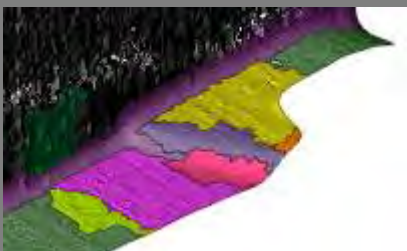


Options in Detail

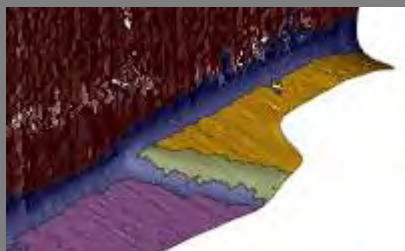
Sensitivity – Specifies the search sensitivity of feature regions.

Note:

When the slider is moved towards **More**, search sensitivity will be increase. Higher search sensitivity may generate many particle regions if a mesh is rough.



Sensitivity: 100

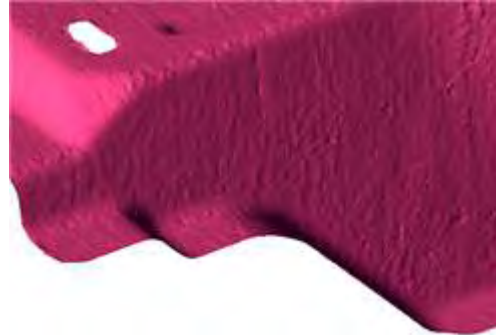


Sensitivity: 20

Mesh Roughness – Adjusts the roughness of a mesh depending on the noise level.



Rough Mesh



Smooth Mesh

See Also

[Tools > Region Tools > Merge](#)

[Tools > Region Tools > Split](#)

[Tools > Region Tools > Insert](#)

[Tools > Region Tools > Separate](#)

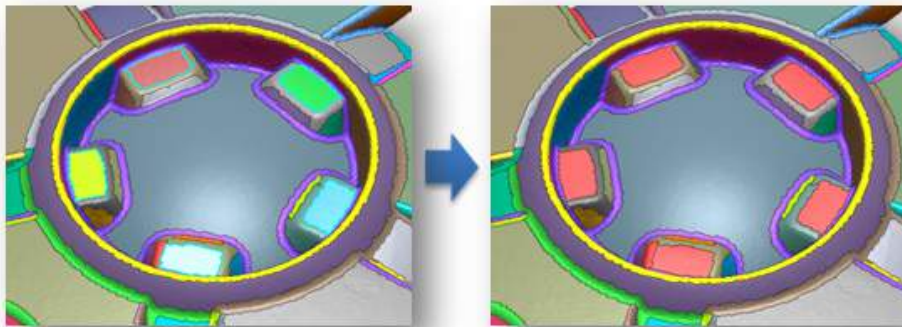
[Tools > Region Tools > Enlarge](#)

[Tools > Region Tools > Shrink](#)

Tools > Region Tools > Merge

What

The **Merge** command merges multiple feature regions into a single feature region. This command performs the opposite function of the **Separate** command.



Why

The **Merge** command is useful for:

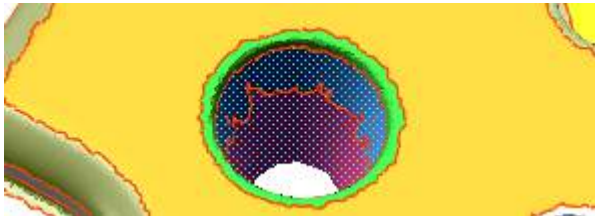
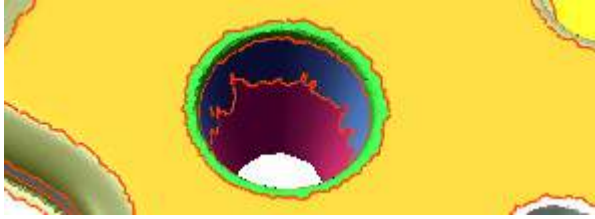
Merging multiple feature regions into a single feature region

- Managing separated feature regions as a single feature region

How to Use

How to merge multiple feature regions into a single feature region

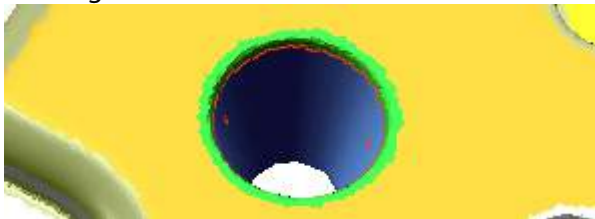
1. Select two regions to merge.



Note:

Poly-faces of regions can be selected and merged with regions.

2. Go to **Tools > Region Tools > Merge** or click the icon in the toolbar. The regions will be merged.



See Also

[Tools > Region Tools > Resegment](#)

[Tools > Region Tools > Split](#)

[Tools > Region Tools > Insert](#)

[Tools > Region Tools > Separate](#)

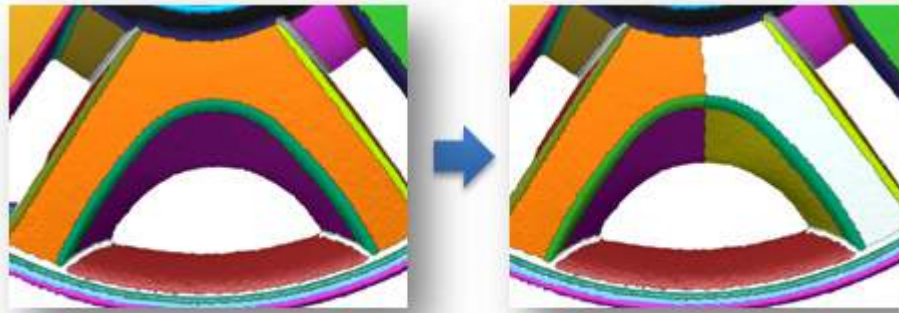
[Tools > Region Tools > Enlarge](#)

[Tools > Region Tools > Shrink](#)

Tools > Region Tools > Split

What

The **Split** command divides a feature region into multiple parts.



Why

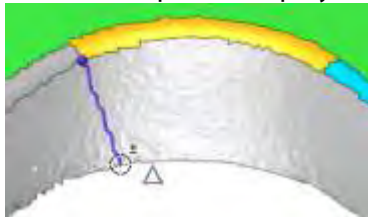
The **Split** command is useful for:

- Manually dividing feature regions
- Separating multiple parts of feature region

How to Use

How to divide feature regions into multiple parts

1. Go to **Tools > Region Tools > Split** or click the icon in the toolbar.
2. Pick the first point of a polyline.



Note:

To exit the selection status, use the **Esc** key on the keyboard.

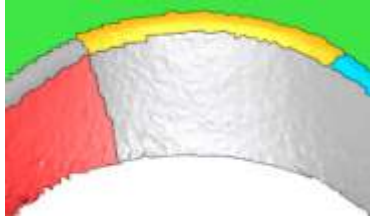
Note:

When splitting regions there are two selection methods; picking and dragging. If a point on a region a polyline is selected a polyline will be used to divide regions. However if regions are selected by dragging, the current selection option method will be used. For example, if the current selection method is the Paint Brush, and regions are selected by dragging, the paint brush tool will be used.





3. Pick the second point of a polyline.
If regions can be split, the polyline color will change from yellow to violet.

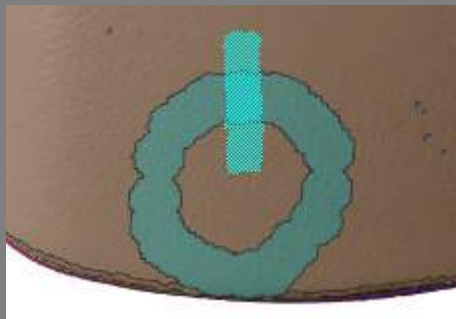


4. Continue by dividing other regions.

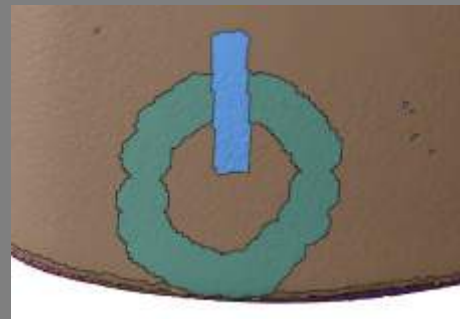


Note:

When a selected area cannot be completely divided into feature regions, the selected area regions will be removed.



Before Split



After Split

Options in Detail

Limit to Region – Splitting is applied to only a selected single region.

See Also

[Tools > Region Tools > Resegment](#)

[Tools > Region Tools > Merge](#)

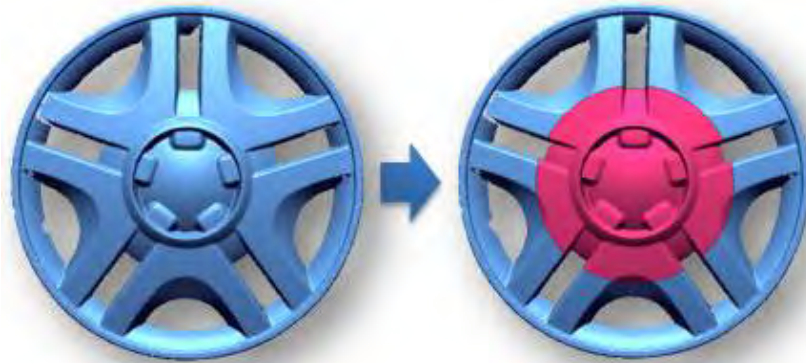
[Tools > Region Tools > Insert](#)

[Tools > Region Tools > Separate](#)

Tools > Region Tools > Insert

What

The **Insert** command manually adds new feature regions.



Why

The **Insert** command is useful for:

- Manually adding new feature regions
- Making interesting areas by grouping feature regions

How to Use

How to insert a new feature region

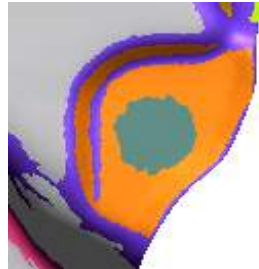
1. Select areas by using the **Paint Brush Selection Mode**. Insert a new region.



2. Go to **Tools > Region Tools > Insert** or click the icon in the toolbar. A new feature region will be inserted on the existing feature region.



Before Insert



After Insert

See Also

[Tools > Region Tools > Resegment](#)

[Tools > Region Tools > Merge](#)

[Tools > Region Tools > Split](#)

[Tools > Region Tools > Separate](#)

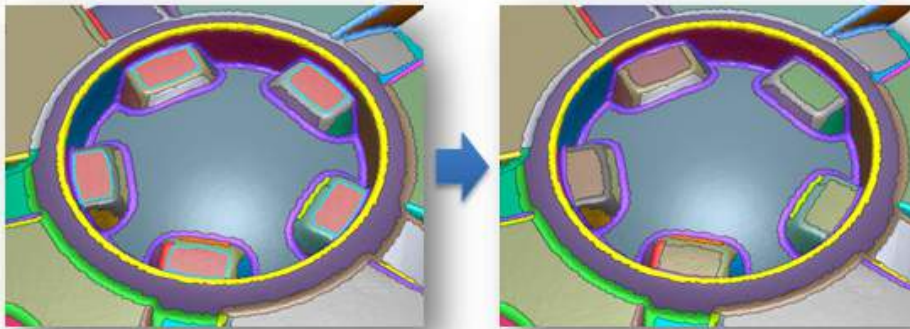
[Tools > Region Tools > Enlarge](#)

[Tools > Region Tools > Shrink](#)

Tools > Region Tools > Separate

What

The **Separate** command creates independent feature regions. This command performs the opposite function of the **Merge** command.



Why

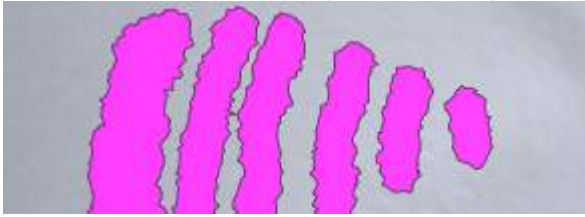
The **Separate** command is useful for:

- Separating a feature region into multiple feature regions

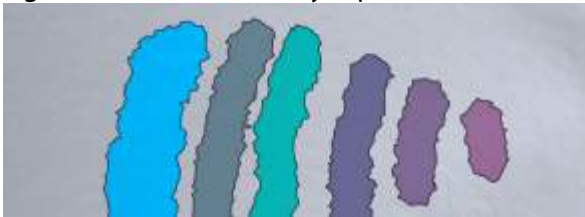
How to Use

How to separate a feature region into multiple feature regions

1. Select a feature region to separate. Feature regions that have been distributed in separate areas can be a target feature region for this command.



2. Go to **Tools > Region Tools > Separate** or click the icon in the toolbar. The feature regions will be individually separated.



See Also

[Tools > Region Tools > Resegment](#)

[Tools > Region Tools > Merge](#)

[Tools > Region Tools > Split](#)

[Tools > Region Tools > Insert](#)

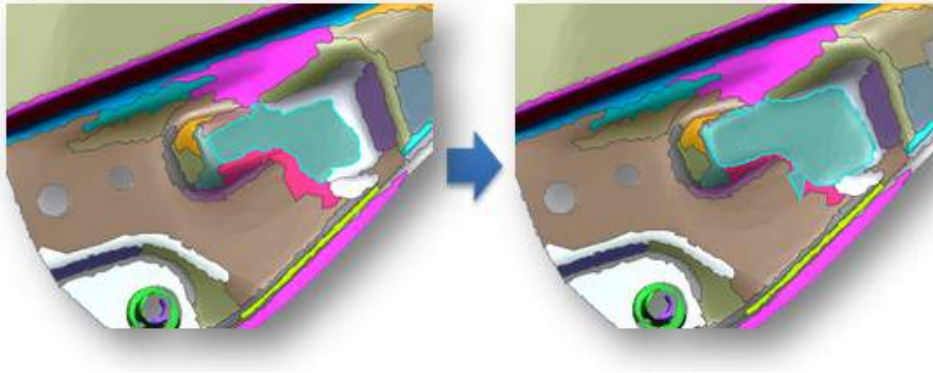
[Tools > Region Tools > Enlarge](#)

[Tools > Region Tools > Shrink](#)

Tools > Region Tools > Enlarge

What

The **Enlarge** command expands a feature region. This command performs the opposite function of the **Shrink** command.



Why

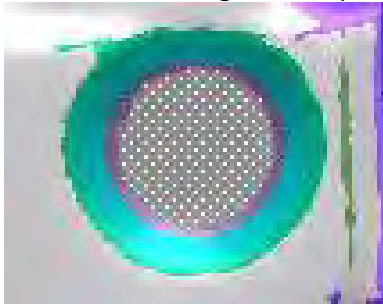
The **Enlarge** command is useful for:

- Expanding an existing feature region

How to Use

How to enlarge the size of feature regions

1. Select a feature region to expand.



2. Go to **Tools > Region Tools > Enlarge** or click the icon in the toolbar. The feature region will be enlarged.



See Also

[Tools > Region Tools > Resegment](#)

[Tools > Region Tools > Merge](#)

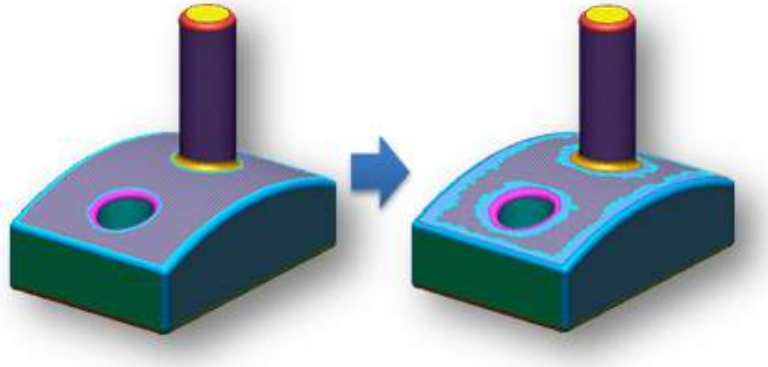
[Tools > Region Tools > Split](#)

[Tools > Region Tools > Insert](#)

Tools > Region Tools > Shrink

What

The **Shrink** command shrinks a feature region. This command performs the opposite function of the **Enlarge** command.



Why

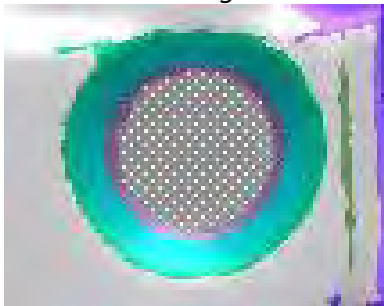
The **Shrink** command is useful for:

- Shrinking a feature region
- Reducing the roughness of feature regions around boundaries
- Correcting geometry properties of automatically classified feature regions

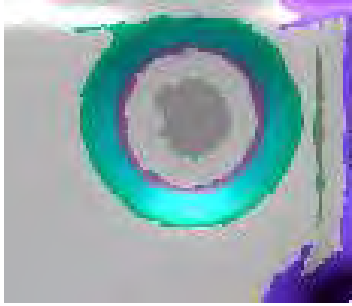
How to Use

How to shrink the size of feature regions

1. Select a feature region to shrink.



2. Go to **Tools > Region Tools > Shrink** or click the icon in the toolbar. The feature region will be shrunk.



See Also

[Tools > Region Tools > Resegment](#)

[Tools > Region Tools > Merge](#)

[Tools > Region Tools > Split](#)

[Tools > Region Tools > Insert](#)

[Tools > Region Tools > Separate](#)

[Tools > Region Tools > Enlarge](#)

7.5.4. Tools > Point Cloud Tools

The **Point Cloud Tools** menu features various commands used to edit point clouds.

The **Point Cloud Tools** are divided into 2 tool groups used to:

Editing Point Clouds

- **Filter Noise**
Filters noisy clusters from point clouds
- **Sampling**
Samples point clouds
- **Smooth**
Reduces the effects of roughness in a point cloud
- **Offset**
Offsets point clouds by a certain distance

Converting Point Clouds From/To Other Entities

- **Construct Mesh**
Constructs poly-faces from a partial point cloud
- **Convert Entities**

Converts other entities into point clouds

- **Separate Point Cloud**

Separates point clouds from original point cloud

Tools > Point Cloud Tools > Filter Noise

What

The **Filter Noise** command cleans noisy clusters from point clouds.



Why

The **Filter Noise** command is useful for:

- Cleaning noisy clusters from point clouds
- Removing unnecessary clusters of points

How to Use

How to filter noisy clusters from point clouds

1. Go to **Insert > Point Cloud**, click the icon in the toolbar, or double-click a point cloud to enter the **Point Cloud** mode.



2. Go to **Tools > Point Cloud Tools > Filter Noise** or click the icon in the toolbar. Check

the **Filter Noise Clusters** option and set the value to **100**.

3. Click the **OK** button to complete the command. Small noisy clusters will be removed from the point cloud.



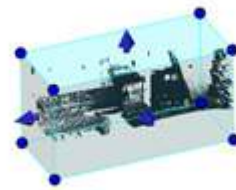
Options in Detail

Filter Outlier Regions – Defines an area of interest and removes all points outside of the defined area.

Filter With Box Volume – Defines an area of interest by the volume of a box.

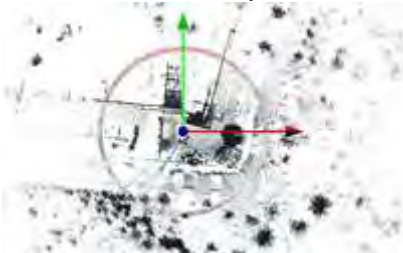


Define Box Volume



After Filter Noise

Filter With Scanner Range – Defines an area of interest by the volume of a cylindrical. If this option is used, the viewpoint will be automatically aligned to the top.



Define Scanner Range



After Filter Noise

Scanner Center – Specifies the center of a cylindrical volume.

Scanner Range – Specifies the radius of a cylindrical volume.

Note:

This option adjusts the radius of cylindrical volume by using the dark red-colored manipulator circle.

Filter Noise Clusters – Specifies the maximum poly-vertices number per noisy cluster.



Max. Poly-Vertex Count Per Noisy Cluster – Specifies the maximum poly-vertices number per noisy cluster that can be removed.

Note:

Noisy clusters are determined by the average distance between poly-vertices and will be removed when the number of poly-vertices contained in noisy clusters is smaller than a specified value.

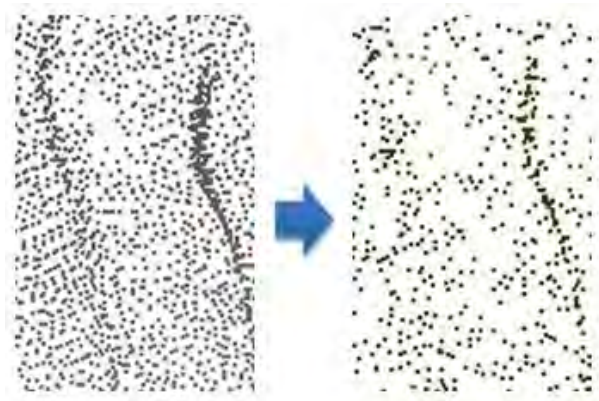
See Also

[Tools > Point Cloud Tools > Sampling](#)

Tools > Point Cloud Tools > Sampling

What

The **Sampling** command reduces the number of poly-vertices in a model based on curvature, ratio or distance.



Why

The **Sampling** command is useful for:

- Reducing the number of poly-vertices based on curvature, ratio, or distance
- Reducing the number of poly-vertices based on an allowable tolerance for dealing with large point clouds

- Removing redundant poly-vertices from points clouds

How to Use

How to sample point clouds

1. Go to **Insert > Point Cloud**, click the icon in the toolbar, or double-click a point cloud to enter the **Point Cloud** mode.



Note:

This command can be applied to the partial selected poly-vertices.

2. Go to **Tools > Point Cloud Tools > Sampling** or click the icon in the toolbar.
3. Choose the **Uniform Ratio** method and check the **Consider Curvature** option to sample point cloud based on its curvature flow. Set the **Sampling Ratio** to **40 %**.
4. Click the **OK** button to complete the command.



Options in Detail

Methods

Uniform Ratio – Reduces the number of poly-vertices uniformly by the ratio of poly-vertices.



Consider Curvature – Samples a point cloud based on its curvature flow. With this option, poly-vertices in a high curvature area are sampled less than poly-vertices in a low curvature area in order to maintain the accuracy of a curvature flow.



Target Poly-Vertex Count – Specifies a target number of poly-vertices to remain after sampling.

Sampling Ratio – Samples data points by a specified value. When the ratio is 100%, all selected data will be used. When the ratio is 50%, half of the selected data will be used.

Uniform Distance – Reduces the number of poly-vertices so that poly-vertices are uniformly arranged by average distance. This method removes redundant poly-vertices and constructs a grid-type point cloud.



Average Distance Between Poly-Vertices – Specifies an average distance between poly-vertices. To estimate an average distance, click the **Estimate** button.

To manually set the distance or radius, use the **Measure Distance** or **Measure Radius** buttons.

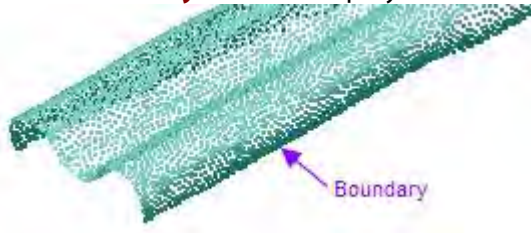
To measure distance between poly-vertices, click the **Measure Distance** button and pick two poly-vertices.

To measure a radius of a hole, click the **Measure Radius** button and pick three poly-vertices.

Sampling Ratio – Refer to the **Sampling Ratio** option in the **Uniform Ratio** method.

More Options

Preserve Boundary – Preserves poly-vertices around boundaries.



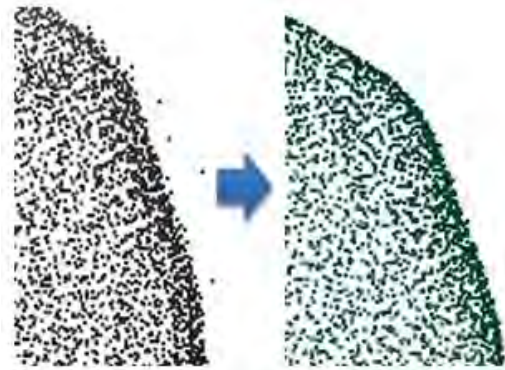
See Also

[Tools > Point Cloud Tools > Filter Noise](#)

Tools > Point Cloud Tools > Smooth

What

The **Smooth** command reduces the roughness in the outer shape of point clouds.



Why

The **Smooth** command is useful for:

- Reducing the effects of roughness in a point cloud

How to Use

How to reduce the effects of roughness in a point cloud

1. Go to **Insert > Point Cloud**, click the icon in the toolbar, or double-click a point cloud to enter the **Point Cloud** mode.



Note:

This command can be applied to partially selected poly-vertices. If 3D scan data is scanned by a long-range 3D laser scanner using a spherical measurement type, this information will be used for smoothing.

2. Go to **Tools > Point Cloud Tools > Smooth** or click the icon in the toolbar.
3. Adjust the **Strength** to **Max** and the **Smoothness Level** to middle.
4. Click the **OK** button to complete the command.



Options in Detail

Options

Strength – Adjusts the weight of smoothness by moving the slider between **Min** and **Max**.

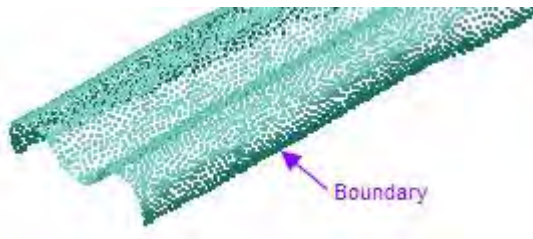
Smoothness Level – Specifies the iteration of smoothness by moving the slider between **Min** and **Max**.

More Options

Allowable Deviation – Specifies an allowable deviation range and limits the deformation of poly-vertices within the allowable deviation during smoothing.



Do Not Move Boundaries – Preserves poly-vertices around boundaries.



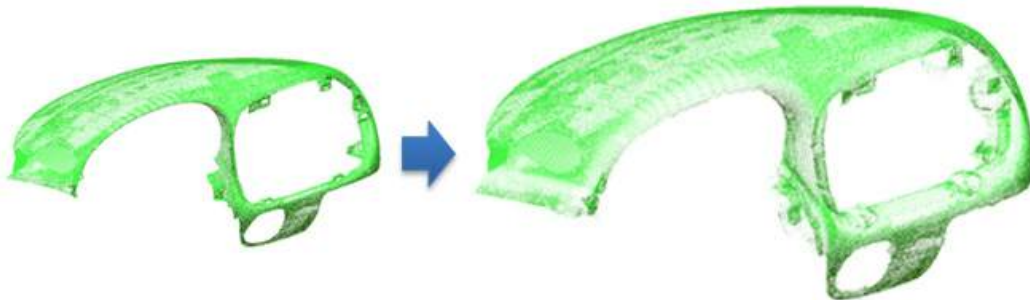
See Also

[Tools > Mesh Tools > Smooth](#)

Tools > Point Cloud Tools > Offset

What

The **Offset** command offsets a point cloud by a specified distance.



Why

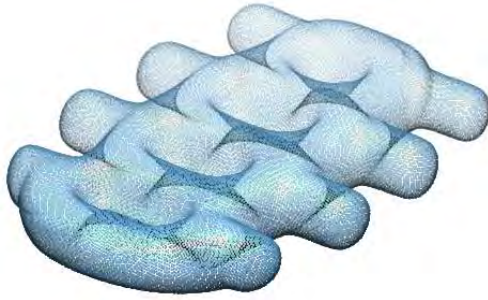
The **Offset** command is useful for:

- Offsetting a point cloud by a specific distance
- Creating a new point cloud that has the same shape but a different scale from an original point cloud
- Creating the opposite side of a point cloud from a partial scan of an object

How to Use

How to offset point clouds by a certain distance

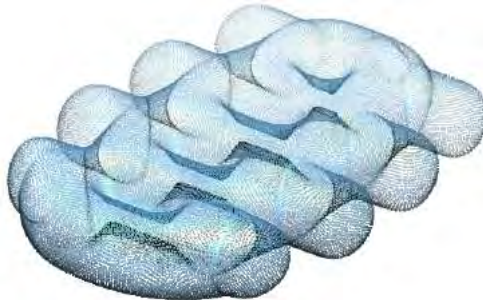
1. Go to **Insert > Point Cloud**, click the icon in the toolbar, or double-click a point cloud to enter the **Point Cloud** mode.



Note:

This command can be applied to partial selected poly-vertices.

2. Go to **Tools > Point Cloud Tools > Offset** or click the icon in the toolbar.
3. Set the **Offset Distance** to **3mm**.
4. Click the **OK** button to complete the command.



Options in Detail

Distance – Specifies an offset distance. To change the offset direction, click the **Reverse Direction** button.

To manually set the distance or radius, use the **Measure Distance** or **Measure Radius** buttons.

To measure distance between poly-vertices, click the **Measure Distance** button and pick two poly-vertices.

To measure a radius of a hole, click the **Measure Radius** button and pick three poly-vertices.



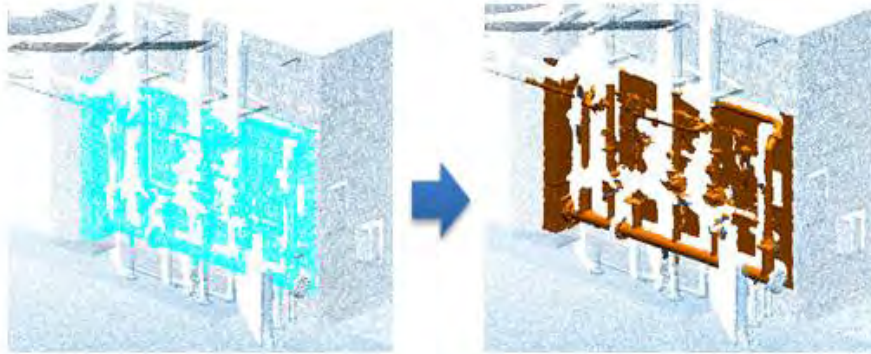
See Also

[Tools > Mesh Tools > Offset](#)

Tools > Point Cloud Tools > Construct Mesh

What

The **Construct Mesh** command generates a mesh from selected poly-vertices.



Why

The **Construct Mesh** command is useful for:

- Creating a mesh from partial areas of a point cloud

How to Use

How to construct poly-faces from a part of point clouds

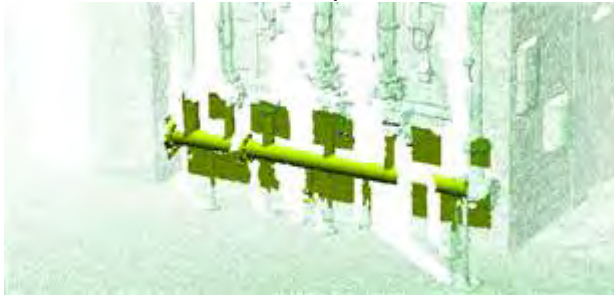
1. Go to **Insert > Point Cloud**, click the icon in the toolbar, or double-click a point cloud to enter the **Point Cloud** mode.



2. Go to **Tools > Point Cloud Tools > Construct Mesh** or click the icon in the toolbar.
3. Select poly-vertices in a point cloud.



4. Click the **OK** button to complete the command.



Options in Detail

Select Entities

Point Clouds – Selects point clouds or partial poly-vertices as a target entity.

Options

Methods

This command features 2 methods:

- **Mesh Construction**

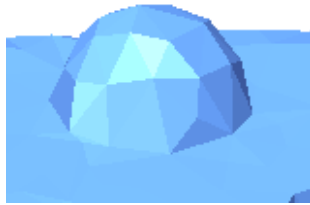
Generates a polygonal mesh by following the geometric shape of a point cloud

- **New Mesh Construction**

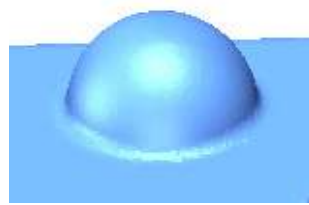
Generates a polygonal mesh by using voxel data structures

Mesh Construction – Generates poly-faces by following the geometric shape of a point cloud. This method is useful for generating polygonal mesh from a point cloud that has multiple combined directional scans that cannot be projected onto a virtual plane or sphere or when results are required to be constructed by poly-faces with uniform poly-edge lengths.

Geometry Capture Accuracy – Defines geometry capture accuracy. When the slider is moved towards **Loose**, less poly-faces will be used to create a shape. When the slider is moved towards **Tight**, more poly-faces will be used to create a shape.



Loose



Tight

Scanner Accuracy – Specifies the accuracy of a scanner. Specify a value by referring to the specification of a particular scanner. For example, if target 3D scan data is scanned by a small/medium object scanner, the default value of 0.05 mm is suitable. If target 3D scan data is scanned by a long-range laser scanner, increase the value based on the precision of the scanner.

Note:

In general, if scan data is clean with a low level of noise, has a regular density, and also has satisfactory align accuracy, then using the maximum value of the 'Scanner Accuracy' slider will preserve and use most points just as they are.

Each decreased step in the slider will decrease the creation of meshes by approximately a half of the previous step. The result will not be a uniform reduction and high curvature regions will be preserved, which is similar to applying a 50% decimation.

For example, the following combinations create almost the same results:

- 'Scanner Accuracy' is 0.05mm and 'Geometry Capture Accuracy' is at the 4th step
- 'Scanner Accuracy' is 0.0333mm ($0.05 / 1.5$) and 'Geometry Capture Accuracy' is at the 3th step
- 'Scanner Accuracy' is 0.075mm (0.05×1.5) and 'Geometry Capture Accuracy' is at the 5th step

New Mesh Construction – Generates poly-faces by using voxel data structures. This method is useful for generating polygonal mesh from a point cloud that does not have many open boundaries. This method is also quicker and can produce better results than the **Mesh Construction** method.

Note:

To get satisfactory results from the New Mesh Construction method, scan data must have normal information and using a mesh that has few open boundaries is ideal because the algorithm extends boundaries beyond their initial settings. The New Mesh Construction method keeps normal information consistent on connected triangles and can be useful for long range scan data and scan data with no boundaries.

Global Shape Fitting Resolution – Controls how many voxel structures are used to create a resulting mesh. The higher the voxel structure count, the more control points can be used to represent the original shape of the scan data.

Shape Fitting To Source Data – Defines geometry capture accuracy. When the slider is moved towards **Loose**, less poly-faces will be used to create a shape. When the slider is moved towards **Tight**, more poly-faces will be used to create a shape.

More Options

Remove Original Data – Removes original 3D scan data after triangulation.

Reduce Noise – Reduces noisy poly-faces.



Without Reduce Noise option

With Reduce Noise option

Remove Redundant Points – Removes redundant poly-vertices by a specified distance for better triangulation results.

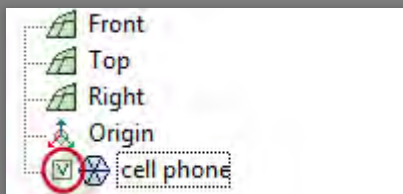
Remove Original Data – Removes original point cloud after triangulation.

Suppress Result In Feature Tree – Suppresses results from the **Feature Tree**.

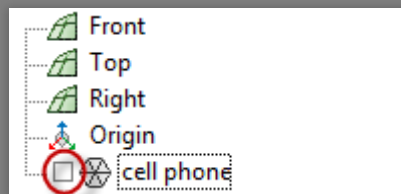
Note:

The Suppress and Unsuppress mechanisms manage data memory by swapping data dynamically between physical memory and a hard disk. This function is useful for effectively managing and handling a large 3D scan data file.

Created Features can be suppressed or unsuppressed by checking the box next to the feature name in the Feature Tree.



Unsuppressed



Suppressed

Sampling Ratio – Samples data points by a specified value. When the ratio is 100%, all selected data will be used. When the ratio is 50%, half of the selected data will be used.

Assume Vertices Are Densely Placed Along A Direction – Samples redundant poly-vertices closed by a certain distance for better triangulation results. Check this option when target 3D scan data is scanned by a line laser with a CMM/Arm device.

Crease Angle – Specifies a maximum angle between normal vectors of adjacent poly-faces to avoid generating abnormal poly-faces.

Noise Reduction Sampling – Reduces noise using a sampling algorithm.

Create Watertight Mesh – Closes boundaries using the curvature of scan data.

Curved – Creates a closed curved boundary at open boundaries.

Flat – Creates a closed flat boundary at open boundaries.

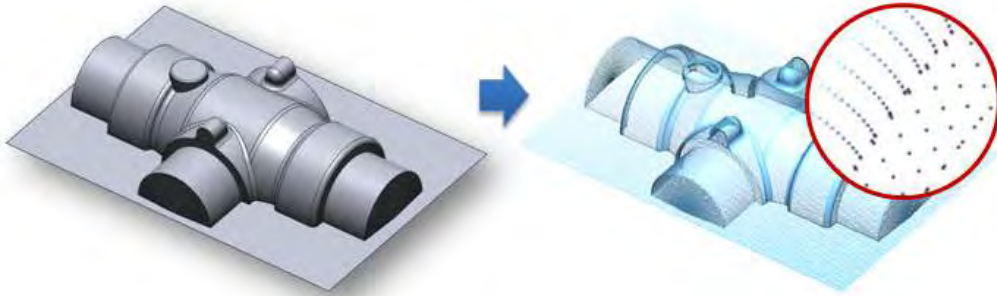
See Also

[Tools > Scan Tools > Triangulate](#)

Tools > Point Cloud Tools > Convert Entities

What

The **Convert Entities** command converts bodies, faces, and curve into point clouds.



Why

The **Convert Entities** command is useful for:

- Converting other entities into a point cloud

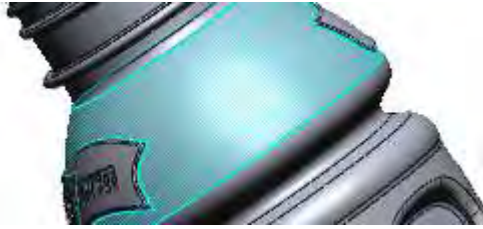
How to Use

How to convert an entity into a point cloud

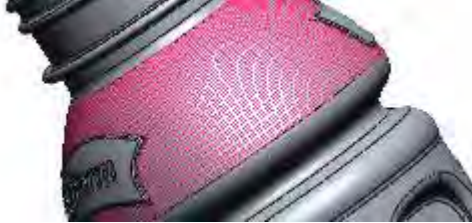
1. Go to **Insert > Point Cloud**, click the icon in the toolbar, or double-click a point cloud to enter the **Point Cloud** mode.



2. Go to **Tools > Point Cloud Tools > Convert Entities** or click the icon in the toolbar.
3. Select a face on a body to convert it into a point cloud.



4. Click the **OK** to complete the command.



5. Hide the body in the **Model View** to see only the point cloud.



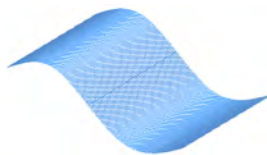
Options in Detail

Target Entities

Entities – Selects curves, 3D sketches, sketches, faces, and surface and solid bodies as target entities.



Body



Face



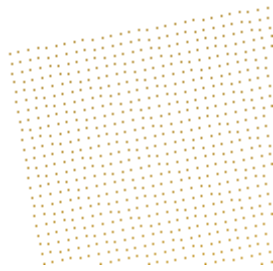
Curve

Options

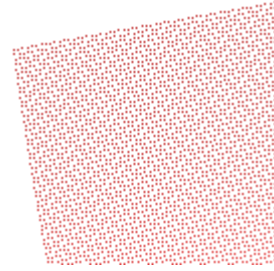
Quality – Specifies the resolution of a point cloud by moving the slider between **Low** and **High**.



Quality: Low



Quality: Medium



Quality: High

Advanced – Adjusts the distance between poly-vertices in detail.

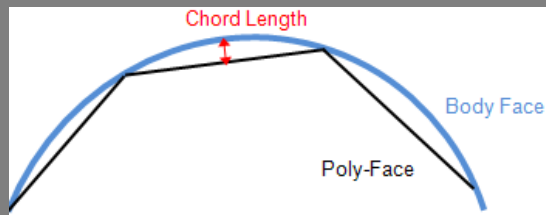
Max. Chord Height – Specifies the maximum chord height of poly-faces to be converted. To manually set the distance or radius, use the **Measure Distance** or **Measure Radius** buttons.

To measure distance between poly-vertices, click the **Measure Distance** button and pick two poly-vertices.

To measure a radius of a hole, click the **Measure Radius** button and pick three poly-vertices.

Note:

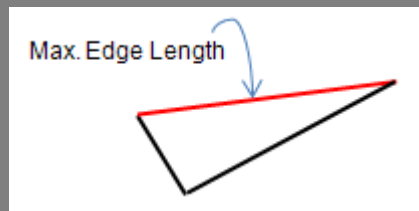
The chord height is the distance between a mesh and a face of a target body to be converted.



Max. Edge Length – Specifies the maximum edge length of poly-faces to be converted.

Note:

This option limits the maximum edge length of poly-faces to be converted.



Min. Edge Length – Specifies the minimum edge length of poly-faces to be converted.

Note:

This option limits the minimum edge length of poly-faces to be converted.



More Options

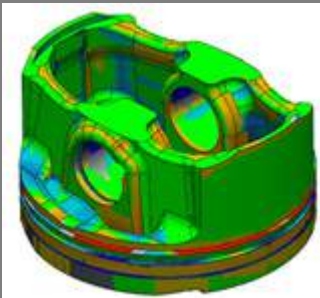
Include Accuracy Analyzer Color – Adds a deviation color map onto a converted point cloud as a vertex color.

Note:

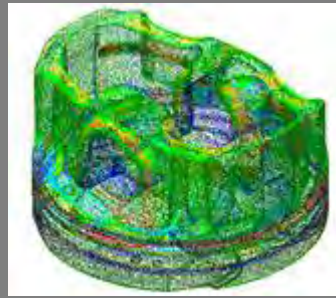
This option is useful for creating a point cloud with a deviation color map from a modeled solid or surface body. It is available when scan data and a modeled solid or surface bodies exist and have been compared using Accuracy Analyzer™.

To use this option:

1. Choose the **Deviation** option in the **Accuracy Analyzer** panel in the **Default** mode.
2. Go to **Insert > Point Cloud** or click the icon in the toolbar.
3. Go to **Tools > Point Cloud Tools > Convert Entities** or click the icon in the toolbar.
4. Select modeled bodies.
5. Check the **Include Accuracy Analyzer Color** option.
6. Click the **OK** button.



Deviation Result



Colored Point Cloud

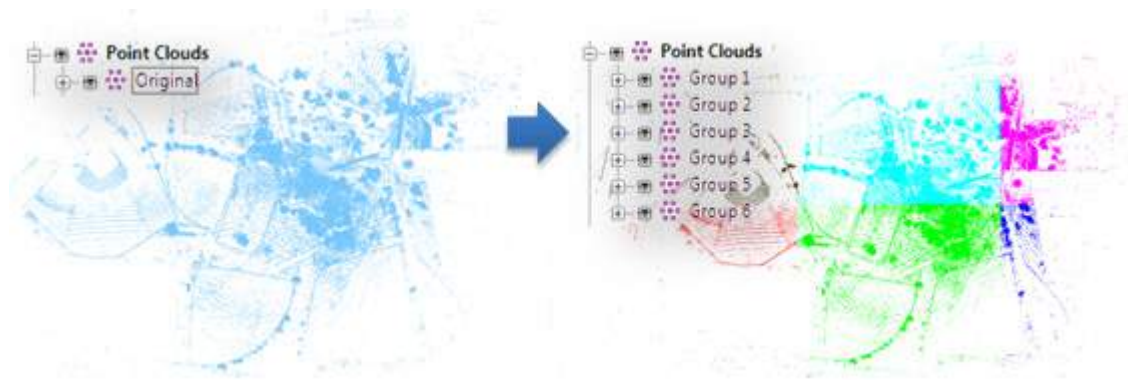
See Also

[Tools > Mesh Tools > Convert Body](#)

Tools > Point Cloud Tools > Separate Point Cloud

What

The **Separate Point Cloud** command takes a single point cloud and divides it into several groups.



Why

The **Separate Point Cloud** command is useful for:

- Separating areas of interest from a large point clouds
- Handling with large-sized point clouds
- Modeling with district units

How to Use

How to separate a point cloud from an original point cloud

- 1 Go to **Insert > Point Cloud**, click the icon in the toolbar, or double-click a point cloud to enter the **Point Cloud** mode.



- 2 Go to **Tools > Point Cloud Tools > Separate Point Cloud** or click the icon in the toolbar.

- 3 Select target point clouds, then click the **Next Stage** button .

- 4 Choose the **Grid** method, and then draw a rectangle by dragging to define the separating areas.



- 5 Set the grid size by inputting the X and Y values of cells.



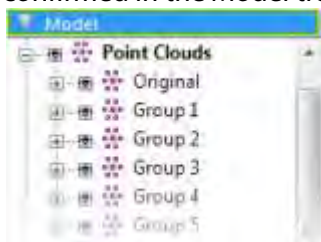
6 To manually adjust the distance between cells, drag the border of a cell.

•



7 Click the **OK** button to complete the command. Separated point clouds can be confirmed in the Model tree.

•



Options in Detail

Select Entities

Point Clouds – Selects a point cloud or multiple point clouds as target entities.

Methods

Grid – Divides a point cloud by a grid.

Draw Grid – Draws a rectangular grid by dragging on the point cloud.

No. Of Cells – Defines the number of cells in the grid. Cell size can be adjusted by dragging the border of a cell.




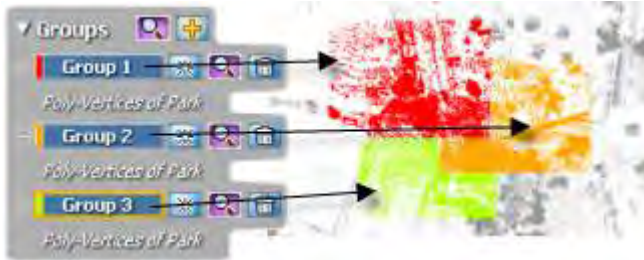
Overlap Range – Defines the overlapping area between neighboring point clouds.

Ratio – Defines the overlapping area by the ratio of the cell size.

Distance – Defines the overlapping area by the distance from border of cells.

Selection – Divides a point cloud by the manual selection of points.

Groups – Includes selected points in a current group. More groups can be added using the Add Group button .



More Options

Remove Original Data – Removes separated point clouds from the original data after the command is finished.

Show Target Point Could Only – Hides remaining point clouds in Stage2 if point clouds are selected as a target from multiple point clouds in Stage1. If this option is unchecked all point clouds will be shown.

See Also

[Insert > Point Cloud](#)

7.5.5. Tools > Texture Tools

The **Texture Tools** menu features various commands used to edit and manage colors and textures on a mesh or point cloud.

The **Texture Tools** are divided into 3 tool groups used to:

Editing Texture

- **Add Texture**
Attaches texture onto a mesh
- **Edit Texture**
Edits texture on mesh
- **Remove Texture**
Removes texture mapped on a mesh
- **Export Texture**

Exports texture to a 2D image file format

Managing Texture And Color

- **Convert Texture To Color**
Converts a texture to a vertex color
- **Convert Color To Texture**
Converts a vertex color to a texture
- **Copy Texture/Vertex Color**
Copies a texture or a vertex color

Matching Colors

- **Match Color**
Matches colors

Tools > Texture Tools > Add Texture

What

The **Add Texture** command attaches 2D texture, also known as a bitmap image, onto a mesh.



Why

The **Add Texture** command is useful for:

- Attaching 2D textures onto a mesh
- Attaching a high quality 2D texture image onto 3D mesh model
- Creating a 3D texture-mapped model from a mesh model scanned by a 3D scanner that cannot capture color information

How to Use

How to attach texture onto a mesh

1. Select a mesh. Go to **Insert > Mesh**, click the icon in the toolbar, or double-click the mesh to enter the **Mesh** mode.

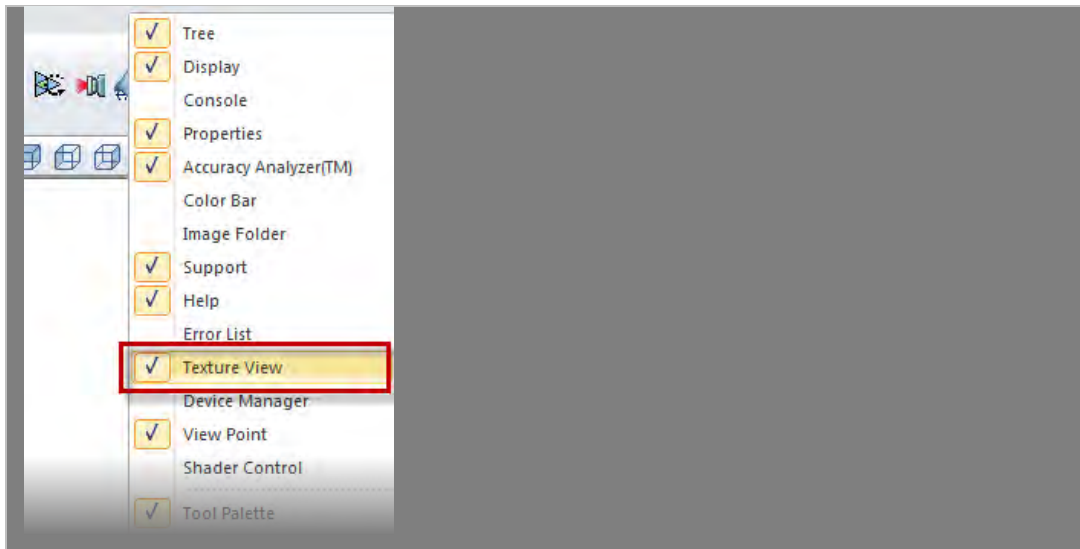


2. Go to **Tools > Texture Tools > Add Texture** or click the icon in the toolbar.
3. Click the **Import Texture** button and import an image file. The imported image will be shown in the **Texture View**.

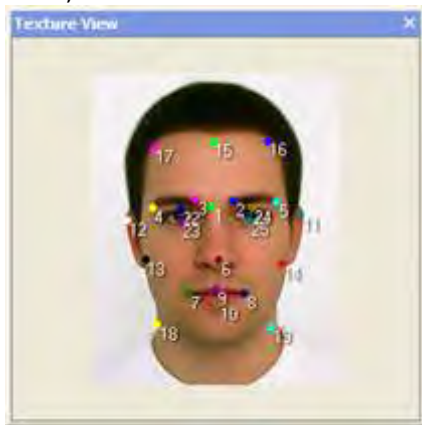


Note:

When Texture View is not visible, right-click on an empty area in the toolbar and choose Texture View in the pop-up list.

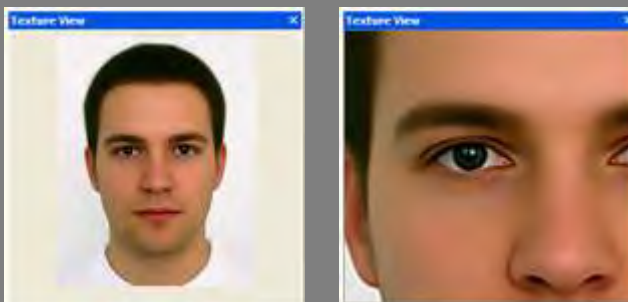


4. Pick points on the image and pick the corresponding points on the model. To get better results,



Note:

Zooming and panning an image in Texture View is the same as manipulating a model in the Model View.



5. Click the **OK** button to complete the command.



Note:

After adding a texture it can be finely adjusted by using the Smudge method in the Edit Texture command.



Options in Detail

Method

Entire – Applies image mapping to an entire area of a mesh.

Region – Applies image mapping to the selected areas of a mesh.

Load Texture

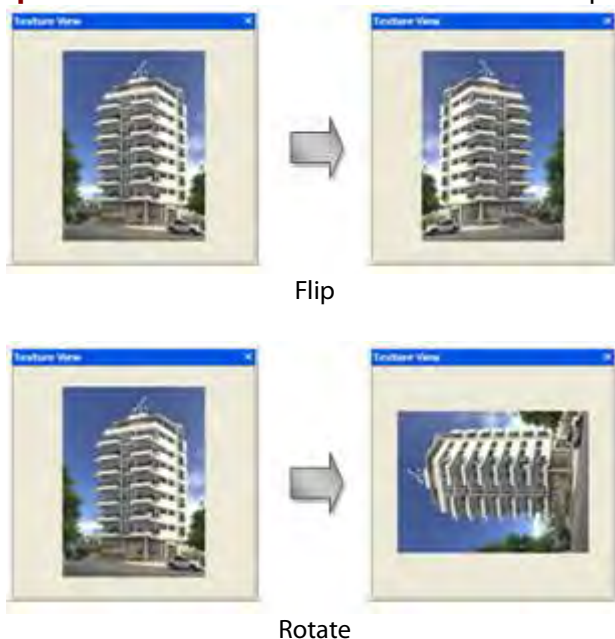
Load Texture – Imports a texture image to be mapped on a mesh.

Add Texture Option

Brightness – Specifies the brightness of a texture image between **-255** and **255**, or sets the brightness by moving the slider between **Dark** and **Bright**.



Flip & Rotate – Uses transformation buttons to flip or rotate an imported texture image.



Create Corresponding Points – Picks corresponding points on an image and mesh.

Note:

To reset all selections, click the **Select None** button. To remove only the last picked point, click the **Deselect Last** button.

Delete Corresponding Points – Select the **Select** button and defined corresponding points to remove them. To remove selected points, click the **Delete** button or press the **Delete** key.

See Also

[Tools > Texture Tools > Remove Texture](#)

Tools > Texture Tools > Edit Texture

What

The **Edit Texture** command edits brightness, contrast, and U/V coordinate color balance of mapped texture on mesh data. This command is available with color or texture mapped mesh and point clouds.



Why

The **Edit Texture** command is useful for:

- Editing textures on a mesh

How to Use

How to edit texture on mesh

1. Select a mesh. Go to **Insert > Mesh**, click the icon in the toolbar, or double-click a mesh to enter the **Mesh** mode.
2. Go to **Tools > Texture Tools > Edit Texture** or click the icon in the toolbar.



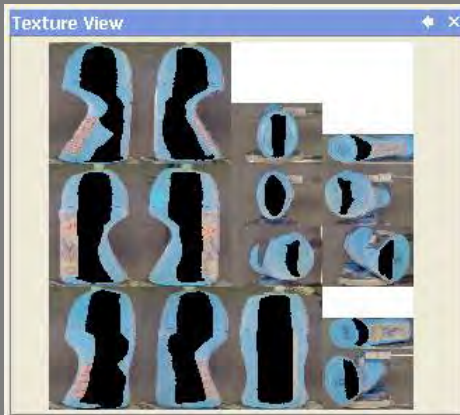
3. Choose the **Auto Level** method.



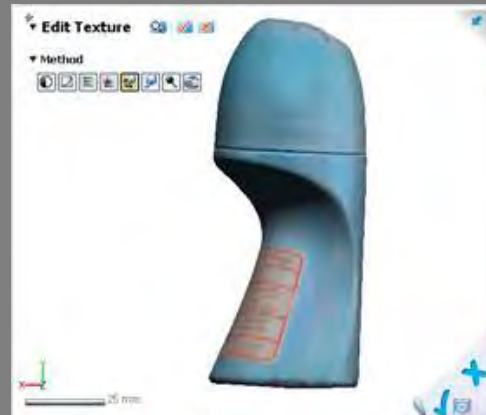
Note:

The **Texture View** will be displayed when a target mesh has texture. The **Texture View** shows all textures in mesh data and enables the interactive editing of texture.

Zooming or panning a textured image is the same as manipulating a model in the **Model View**. To adjust the editing zone, drag the mouse cursor using the Left Mouse Button, or **LMB**, while holding the **Ctrl** key.



In Texture View



In Model View

4. Click the **OK** button to complete the command. The overall color will be brighter.



Options in Detail

Methods

This command features 8 methods:

- **Brightness And Contrast**
Adjusts brightness and contrast
- **Curves**
Edits the tonality of colors by a curve
- **Balance**
Edits color balance
- **Auto Level**

Automatically adjusts color level

- **UV Coordinate**

Edits U/V coordinates on a texture

- **Smudge**

Adds smudges onto a texture

- **Dodge**

Lightens a texture

- **Burn**

Darkens a texture

Brightness And Contrast – Adjusts **Brightness** and **Contrast**.

Brightness – Sets the brightness of a texture by specifying a value between **-255** and **255** or moving the slider between **Min.** and **Max.**



Brightness: -100



Brightness: 0



Brightness: 100

Contrast – Sets the contrast of a texture by specifying a value between **-100** and **100** or moving the slider between **Min.** and **Max.**



Contrast: -50



Contrast: 0

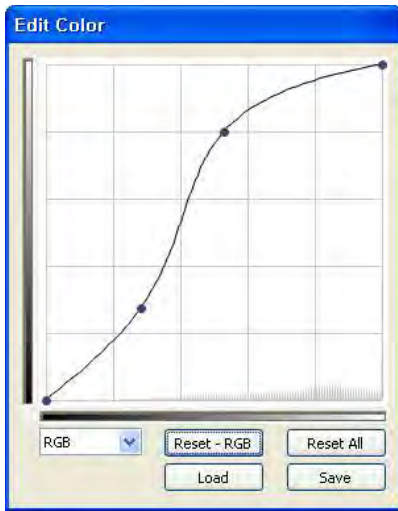


Contrast: 50

Curves – Edits the tonality of colors by a curve. The shape of the curve can be modified in the **Curves** dialog box to adjust the tonality and color of textures.

Bowing the curve upward lightens textures, and bowing the curve downward darkens them. The steeper sections of the curve represent portions of an image with more contrast. Conversely, flatter sections of the curve represent areas of lower contrast in a texture. Changes can be applied to **RGB** at the same time or for each value independently.

To add control points, click on a curve. To remove control points, select a point and drag it outside of the graph area.



Balance – Edits color balance.

Select Range To Modify – Selects **Shadows**, **Midtones**, or **Highlights** to change its tonal range.

Modify Selected Range's Color Levels – Sets the color balance for a range by specifying a value between **-100** and **100** or by moving the sliders.



Balanced



Emphasizing Red Color

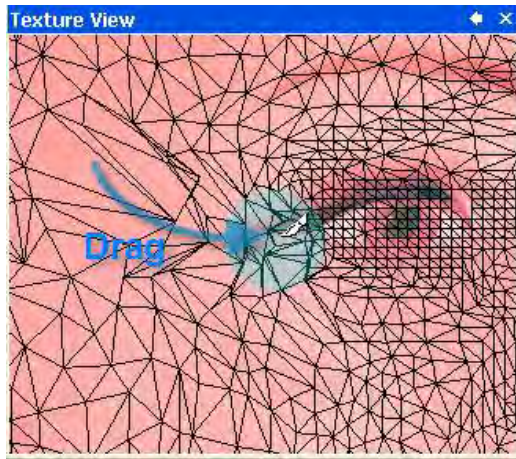


Emphasizing Blue Color

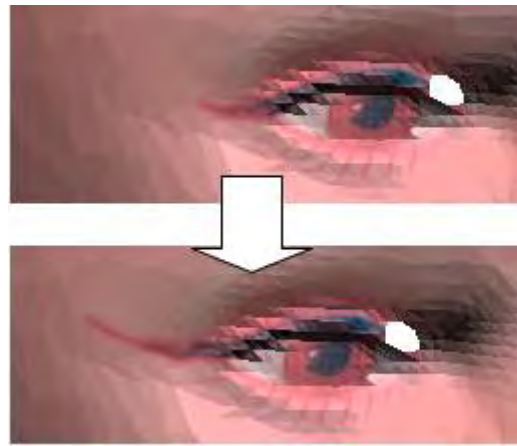
Preserve Luminosity – Preserve the luminosity values in a texture while changing its color. This option maintains the tonal balance in a texture.

Auto Level – Automatically adjusts the color level of a texture. This method clips a portion of the shadows and highlights in each channel and maps the lightest and darkest pixels in each color channel to pure white (level 255) and pure black (level 0). Using **Auto Levels** increases the contrast in a texture.

UV Coordinate (Texture Only) – Edits U/V coordinates on a texture by using a brush in the Texture View. The editing zone can be adjusted by using **Alt + LMB** and dragging the U/V texture map in the **Texture View**.

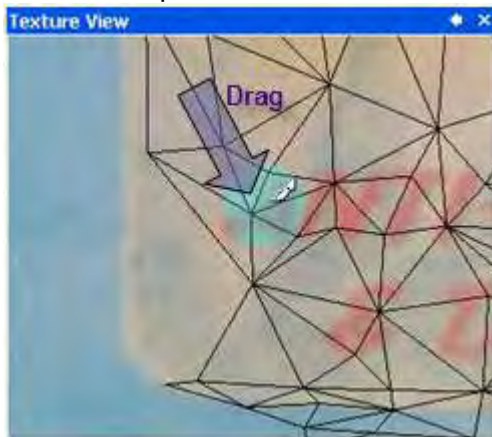


In Texture View

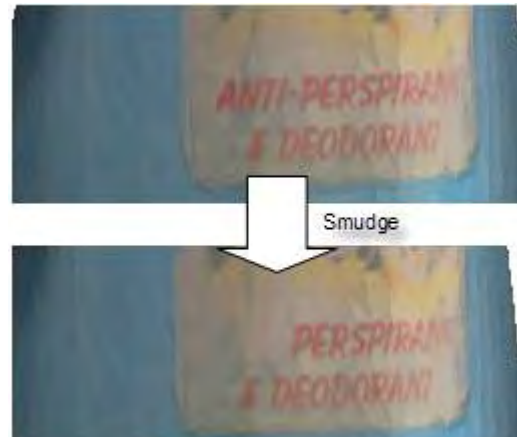


In Model View

Smudge (Texture Only) – Adds smudging on textures by simulating the action of dragging a finger through wet paint. The command picks up color where the stroke begins and pushes it in the dragging direction. The editing zone can be adjusted by using **Alt + LMB** and dragging the U/V texture map in the **Texture View**.



In Texture View



In Model View

Strength – Adjusts the weight of smudging by moving the slider between **Min** and **Max**.

Dodge and Burn (Texture Only) – Lightens or darkens a texture. The Dodge and Burn methods are based on traditional photography techniques for regulating exposure on specific areas of a print. The **Shadows**, **Midtones**, or **Highlights** options specify the tonal range to be changed.

Exposure – Specifies the strength of a light by moving the slider between **Less** and **More**.

See Also

[Tools > Texture Tools > Add Texture](#)

[Tools > Texture Tools > Remove Texture](#)

Tools > Texture Tools > Remove Texture

What

The **Remove Texture** command removes textures mapped on a mesh.



Why

The **Remove Texture** command is useful for:

- Removing texture from a mesh

How to Use

How to remove textures from mesh

1. Select a mesh. Go to **Insert > Mesh**, click the icon in the toolbar, or double-click the mesh to enter the **Mesh** mode.



2. Go to **Tools > Texture Tools > Remove Texture** or click the icon in the toolbar. Check the results.



See Also

[Tools > Texture Tools > Add Texture](#)

Tools > Texture Tools > Export Texture

What

The **Export Texture** command exports mapping source textures to an image file such as a jpg, bmp, tiff, tga, gif or png.



Why

The **Export Texture** command is useful for:

- Exporting a texture into an image file

How to Use

How to export a texture into a 2D image file format

1. Select a mesh. Go to **Insert > Mesh**, click the icon in the toolbar, or double-click the mesh to enter the **Mesh** mode.
2. Go to **Tools > Texture Tools > Export Texture** or click the icon in the toolbar.
3. The **Export Image** dialog box appears. Select a file format and click the **Save** button.

See Also

[Tools > Texture Tools > Add Texture](#)

Tools > Texture Tools > Convert Texture To Color

What

The **Convert Texture To Color** command converts a texture to a vertex or mesh color.



Why

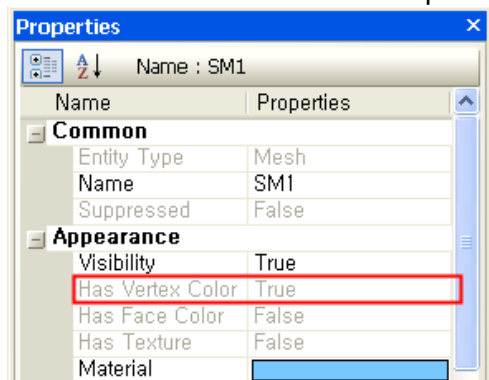
The **Convert Texture To Color** command is useful for:

- Converting a texture to a vertex or mesh color

How to Use

How to convert a texture to a vertex color

1. Select a mesh. Go to **Insert > Mesh**, click the icon in the toolbar, or double-click the mesh to enter the **Mesh** mode.
2. Go to **Tools > Texture Tools > Convert Texture To Color** or click the icon in the toolbar. Check that the **Has Vertex Color** option in the **Property** panel is set to **True**.



See Also

[Tools > Texture Tools > Convert Color to Texture](#)

Tools > Texture Tools > Convert Color To Texture

What

The **Convert Color To Texture** command converts a vertex or mesh color to a texture.



Why

The **Convert Color To Texture** command is useful for:

- Converting a vertex or mesh color to a texture

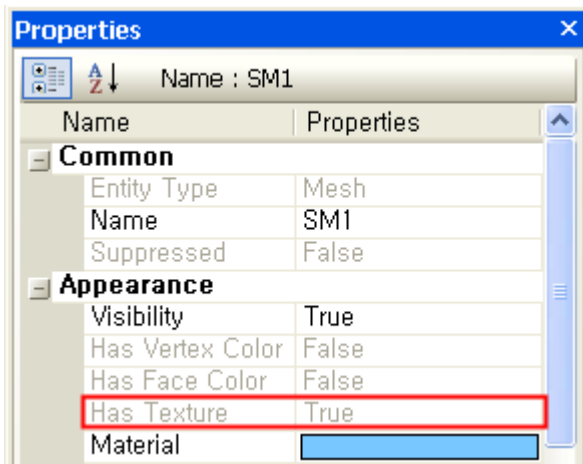
How to Use

How to convert vertex color to texture

1. Select a mesh. Go to **Insert > Mesh**, click the icon in the toolbar, or double-click the mesh to enter the **Mesh** mode.



2. Go to **Tools > Texture Tools > Convert Color To Texture** or click the icon in the toolbar.
Check that the **Has Texture** option in the **Property** panel is set to **True**.



See Also

[Tools > Texture Tools > Convert Texture To Color](#)

Tools > Texture Tools > Copy Texture/Vertex Color

What

The **Copy Texture/Vertex Color** command copies a texture or vertex color from an existing mesh or point cloud and pastes it into another mesh or point cloud.



Why

The **Copy Texture/Vertex Color** command is useful for:

- Copying a texture or vertex color from an existing mesh or point cloud and pasting it into another mesh or point cloud

How to Use

How to copy a texture or vertex color and paste it into another mesh

1. Select a mesh. Go to **Insert > Mesh**, click the icon in the toolbar, or double-click the mesh to enter the **Mesh** mode.



2. Go to **Tools > Texture Tools > Copy Texture/Vertex Color** or click the icon in the toolbar.
3. Select a mesh that has a texture.



4. Click the **OK** button to complete the command. Check that the texture is copied and pasted into the other mesh.



See Also

[Tools > Texture Tools > Convert Texture To Color](#)
[Tools > Texture Tools > Convert Color To Texture](#)

Tools > Texture Tools > Match Color

What

The **Match Color** command synchronizes the color temperature between two meshes or point clouds. This tool is available when point clouds or meshes have color information.



Why

The **Match Color** command is useful for:

- Matching color temperatures between two meshes or point clouds

How to Use

How to match colors

1. Go to **Tools > Texture Tools > Match Color** or click the icon in the toolbar.



2. Choose the **Match Color Based On Reference** option.
3. Select a reference and target meshes.



4. Click the **Next Stage** button to continue.



5. Click the **OK** button to complete the command.

Options in Detail

Choose Among Candidates – Automatically synchronizes the color temperatures of two meshes or point clouds and presents candidates to choose from in the second stage.

Match Color Based On Reference – Manually defines the **Reference** and **Targets**. Targets will be compensated based on the reference color by choosing a result from the candidates in the second stage.

See Also

[Tools > Texture Tools > Copy Texture/Vertex Color](#)

7.5.6. Tools > Sketch Entities

The **Sketch Entities** menu features various commands used to make sketches such as the **Line**, **Rectangle**, **Circle**, **Arc** commands.

Create sketch entities on a base sketch plane by using the various sketch drawing tools to create feature shapes.

The **Sketch Entities** are divided into 6 tool groups used to create:

Automatic Sketching Tool

- **Auto Sketch**

Automatically creates sketch entities based on analyzed geometric features

Line Sketch Type

- **Lines**

Draws lines

- **Rectangles**

Draws rectangles

- **Parallelograms**
Draws parallelograms
- **Polygons**
Draws polygons
- **Slots**
Draws slots

Arc Sketch Type

- **Circles**
Draws circles by defining a center and radius
- **Perimeter Circles**
Draws circles by selecting 3 points
- **3 Tangent Circles**
Draws inscribed circles that touch the edges of 3 other sketch entities
- **Centerpoint Arcs**
Draws arcs by defining a center position and radius
- **Tangent Arcs**
Draws arcs continually connected to another sketch entity by a tangent constraint
- **3 Point Arcs**
Draws arcs by selecting 3 points
- **3 Tangent Arcs**
Draws inscribed arcs that touch the edges of 3 other sketch entities

Spline Sketch Type

- **Ellipses**
Draws ellipses
- **Partial Ellipses**
Draws partial ellipses
- **Parabolas**
Draws parabolas
- **Splines**
Draws splines

See Also Sketch Type

- **Centerlines**

Adds centerlines as a reference for drawing other sketch entities or an axis for constructing geometry features

- **Points**

Adds points as a reference for drawing other sketch entities or a reference point for constructing geometry features

- **Texts**

Add text sketch entities constructed by lines, circles and splines

Useful Sketch Tools

- **By Fitting Polyline**

Creates circle or arc sketch entities by fitting a section polyline

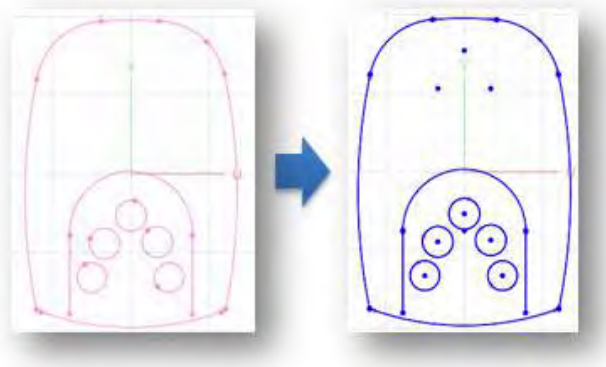
- **By Import**

Imports 2D CAD drawing files as sketch entities onto a base sketch plane

Tools > Sketch Entities > Auto Sketch

What

The **Auto Sketch** command automatically creates sketch entities such as lines, circles, and arcs on section polylines based on analyzed section polylines. This command is available in the **Mesh Sketch** mode only.



Why

The **Auto Sketch** command is useful for:

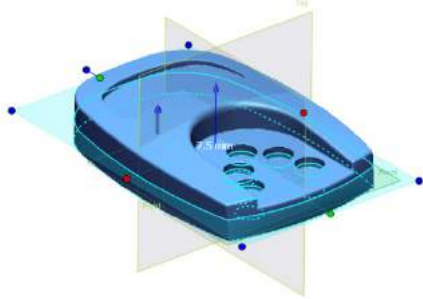
- Quickly creating sketch entities based on scan sections.

How to Use

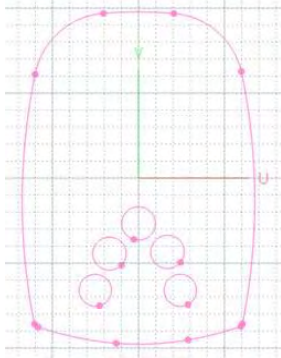
How to automatically create sketch entities on section polyline in the Mesh Sketch mode

1. Go to **Insert > Mesh Sketch** or click the icon in the toolbar to enter the **Mesh Sketch** mode.

Create section polylines. For more information, see [Insert > Mesh Sketch](#).



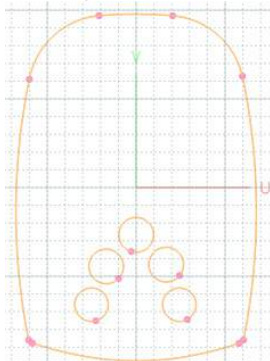
2. **Show** section polylines only.



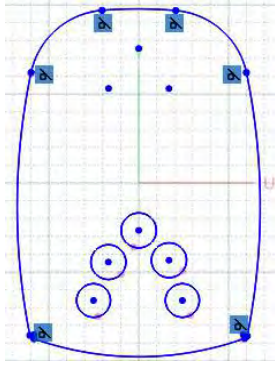
3. Go to **Tools > Sketch Entities > Auto Sketch** or click the icon in the toolbar.
4. Set the resolution of section polylines by adjusting the slider in the **Segment Section Polyline** dialogue box.



5. The segments of the section polylines will be classified.

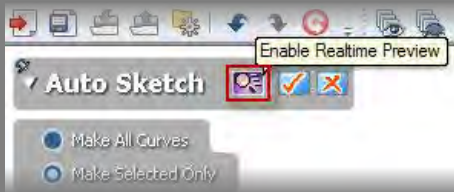


6. Choose the **Make All Curves** option and then click the **OK** button to complete the command.



Note:

The **Enable Realtime Preview** button displays results in real-time without leaving the command.



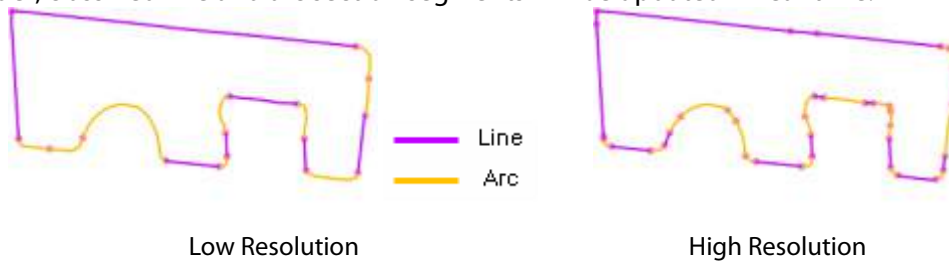
Options in Detail

Methods

Make All Curves – Creates a sketch to whole polylines.

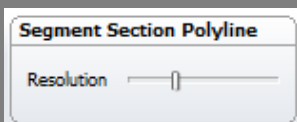
Make Selected Only – Creates a sketch only to selected polylines. To cancel the selection, push the **Cancel** button on the keyboard.

Resolution – Specifies the resolution of section segments by adjusting the slider between Min. and Max. Adjusting the slider towards **Max.**, will construct more segments. While adjusting the slider, classified line and arc section segments will be updated in real-time.



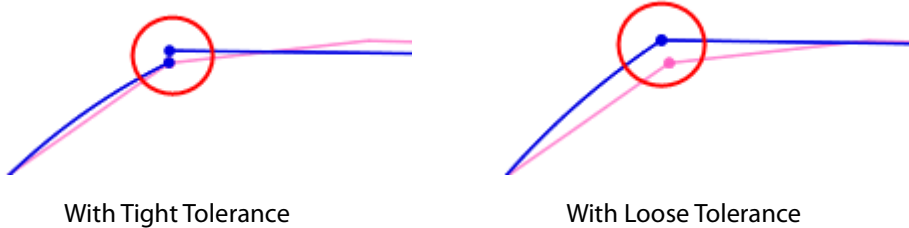
Note:

The **Resolution** option is synchronized with the value of the Resolution option in the **Segment Section Polyline** tool palette. While adjusting the resolution of section polylines, the resolution in the **Segment Section Polyline** tool palette will also be changed.



Geometry Capture Accuracy – Specifies geometry capture accuracy by adjusting the slider between **Loose** and **Tight**. When moving the slider toward **Tight**, sketches will be extracted from section segments with a tight tolerance.

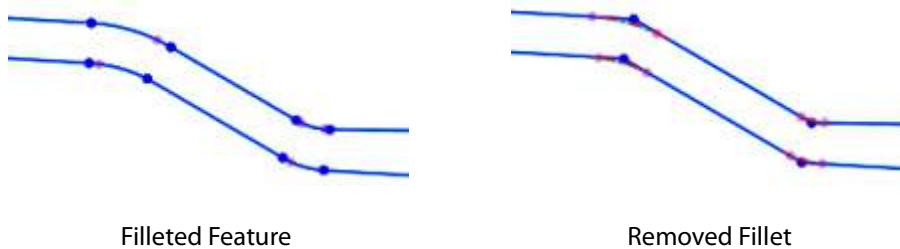
Sewing Tolerance – Specifies a tolerance range for sewing sketches by adjusting the slider between **Min.** and **Max.** When adjusting the slider toward **Max.**, sketches will be joined with coincidence constraint with a loose tolerance.



Note:

When moving the slider toward **Min.**, sketches may not be joined with coincidence constraint because of use of tight tolerance.

Remove Small Fillet – Removes automatically generated small fillets by using a radius value.



See Also

[Tools > Sketch Entities > Line](#)

[Tools > Sketch Entities > Circle](#)

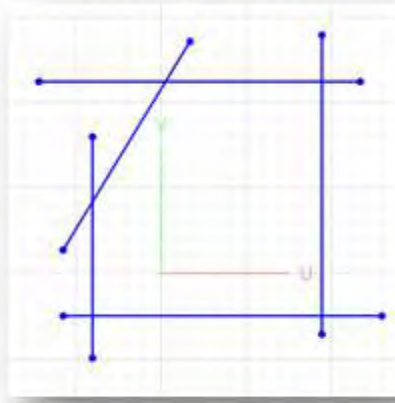
[Tools > Sketch Entities > Centerpoint Arc](#)

[Tools > Sketch Entities > Tangent Arc](#)

Tools > Sketch Entities > Line

What

The **Line** command draws lines on a base sketch plane.



Why

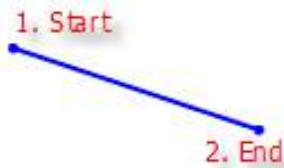
The **Line** command is useful for:

- Drawing lines on a base sketch plane
- Creating a sketch profile to be used for creating planar features

How to Use

A. How to draw lines in Sketch mode

- 1 Go to **Insert > Sketch** or click the icon in the toolbar. Select a base sketch plane to draw sketches on in the **Sketch** mode. For more information, see [Insert > Sketch](#).
- 2 Go to **Tools > Sketch Entities > Line** or click the icon in the toolbar.
- 3 Pick a starting point and draw a line to an ending point.



- 4 Click the right mouse button to complete the construction of the line. Click the **OK** button to complete the command.

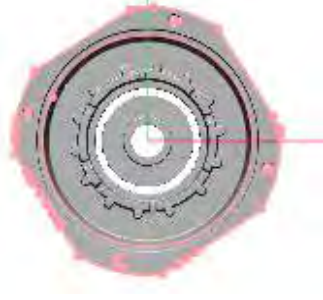
Note:

To finish drawing a line, double-click the left mouse button or single-click the right mouse button. If more lines are needed, continually draw lines without leaving the command. After drawing lines, the command can be finished by clicking the **Cancel** button.

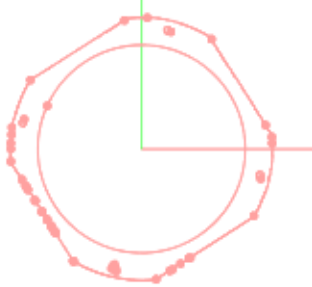
B. How to draw lines on section polylines in Mesh Sketch mode

1. Go to **Insert > Mesh Sketch** or click the icon in the toolbar to enter the **Mesh Sketch** mode.

Create section polylines. For more information, see [Insert > Mesh Sketch](#).

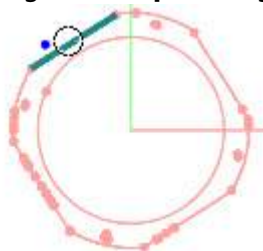


2. Hide mesh to make only the section polylines visible.

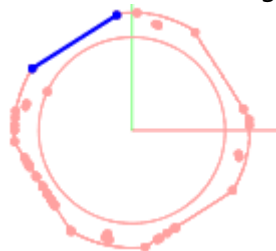


3. Go to **Tools > Sketch Entities > Line** or click the icon in the toolbar.

4. Check the **Fit Polyline** option, select section polylines, and confirm the preview line by clicking the **Accept Fitting** button or double-clicking the left mouse button.

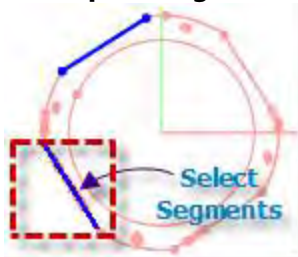


Select Segment

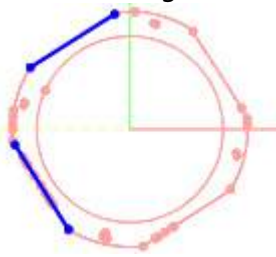


Apply

5. Drag and select segments on the section polyline. Confirm the preview line by clicking the **Accept Fitting** button or double-clicking the left mouse button.

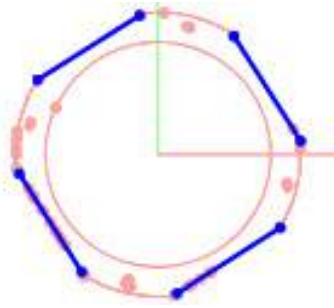


Select Segments



Apply

6. Create more lines as they are needed. Click the **OK** button to complete the command.



Options in Detail

Start – Shows the starting position of a line to be generated.

End – Shows the ending position of a line to be generated.

For Construction – Creates a line for construction. The construction line can be used as a reference for drawing other sketch entities or an axis for constructing revolving features.



Normal Line






For Construction Line

Fit Polyline – Creates a line on a section polyline from a mesh or point cloud.



Note:

To accept a fitting, click the **Accept Fitting** button  or double-click the left mouse button. To cancel a fitting, click the **Cancel Fitting** button . To clear a selection, click the **Clear Selection** button .

See Also

[Tools > Sketch Entities > Rectangle](#)

[Tools > Sketch Entities > Parallelogram](#)

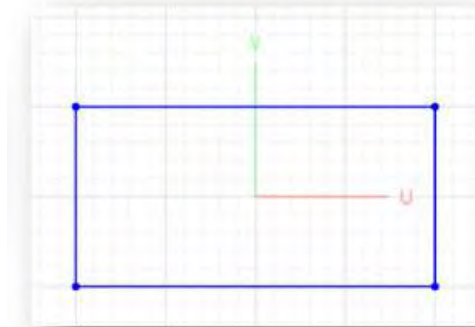
[Tools > Sketch Entities > Polygon](#)

[Tools > Sketch Entities > Slot](#)

Tools > Sketch Entities > Rectangle

What

The **Rectangle** command draws rectangles on a base sketch plane.



Why

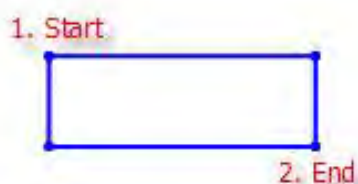
The **Rectangle** command is useful for:

- Drawing rectangles on a base sketch plane
- Creating 4-sided closed outline sketches constructed out of lines and direction constraints
- Creating sketch profiles to be used for creating planar features

How to Use

How to draw Rectangles

1. Go to **Insert > Sketch** or click the icon in the toolbar. Select a base sketch plane to draw sketches on in the **Sketch** mode. For more information, see [Insert > Sketch](#).
2. Go to **Tools > Sketch Entities > Rectangle** or click the icon in the toolbar.
3. Pick a starting point and draw a rectangle to an ending point.



4. Click the **OK** button to complete the command.

Options in Detail

Start – Shows the starting top-left position of a rectangle to be generated.

End – Shows the ending bottom-right position of a rectangle to be generated.

See Also

[Tools > Sketch Entities > Line](#)

[Tools > Sketch Entities > Parallelogram](#)

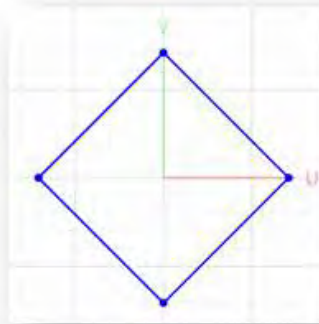
[Tools > Sketch Entities > Polygon](#)

[Tools > Sketch Entities > Slot](#)

Tools > Sketch Entities > Parallelogram

What

The **Parallelogram** command draws parallelograms on a base sketch plane.



Why

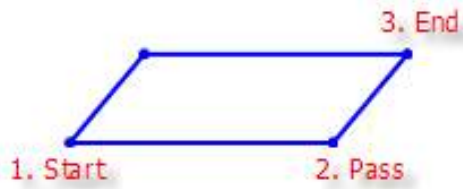
The **Parallelogram** command is useful for:

- Drawing parallelograms on a base sketch plane
- Creating 4-sided closed outline sketches constructed by lines
- Creating a sketch profile used for creating planar features

How to Use

How to draw parallelograms

1. Go to **Insert > Sketch** or click the icon in the toolbar. Select a base sketch plane to draw sketches on in the **Sketch** mode. For more information, see [Insert > Sketch](#).
2. Go to **Tools > Sketch Entities > Parallelogram** or click the icon in the toolbar.
3. Pick a starting point and draw a parallelogram to an ending point.



4. Click the **OK** button to complete the command.

Options in Detail

Start – Shows the starting corner position of a parallelogram to be generated.

Pass – Shows the passing corner position of a parallelogram to be generated.

End – Shows the ending corner position of a parallelogram to be generated.

See Also

[Tools > Sketch Entities > Line](#)

[Tools > Sketch Entities > Rectangle](#)

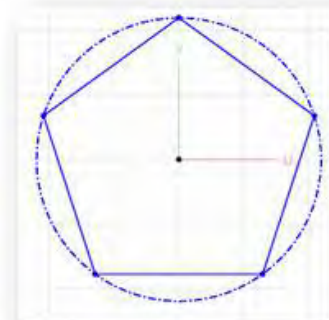
[Tools > Sketch Entities > Polygon](#)

[Tools > Sketch Entities > Slot](#)

Tools > Sketch Entities > Polygon

What

The **Polygon** command draws polygons on a base sketch plane.



Why

The **Polygon** command is useful for:

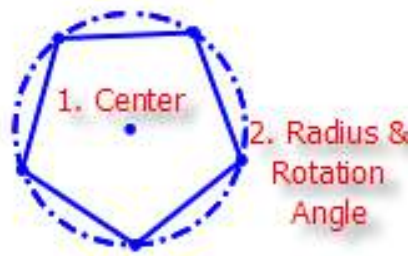
- Drawing polygons on a base sketch plane

- Creating multi-sided closed outline sketches constructed by lines
- Creating a sketch profile to be used for creating planar features

How to Use

How to draw Polygons

1. Go to **Insert > Sketch** or click the icon in the toolbar. Select a base sketch plane to draw sketches on in the **Sketch** mode. For more information, see [Insert > Sketch](#).
2. Go to **Tools > Sketch Entities > Polygon** or click the icon in the toolbar.
3. Set the **Edge Count** to 5.
4. Pick a center point and define a radius and an angle by picking a second point on the base sketch plane.



5. Click the **OK** button to complete the command.

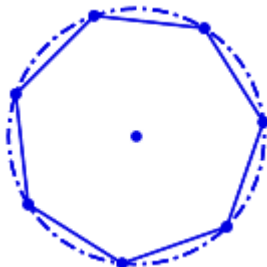
Options in Detail

Center – Shows the center position of a polygon to be generated.

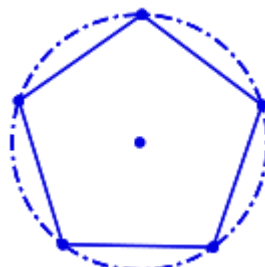
Radius – Shows the radius of a polygon to be generated.

Rotation Angle – Shows the rotation angle of a polygon with respect to the U axis of a base sketch plane.

Edge Count – Specifies the number of edges in a polygon.



Edge Count: 7

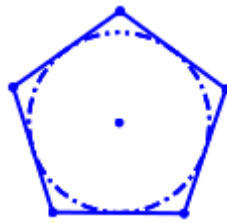


Edge Count: 5

Circumscribed Circle – Creates a polygon using a circumscribed circle.



Circumscribed Circle



Inscribed Circle

See Also

[Tools > Sketch Entities > Line](#)

[Tools > Sketch Entities > Rectangle](#)

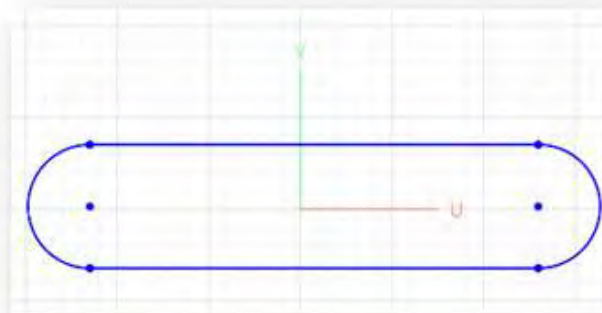
[Tools > Sketch Entities > Parallelogram](#)

[Tools > Sketch Entities > Slot](#)

Tools > Sketch Entities > Slot

What

The **Slot** command draws slots on a base sketch plane.



Why

The **Slot** command is useful for:

- Drawing slots on a base sketch plane
- Creating slot sketches constructed by lines and arcs with tangent constraints
- Creating a sketch profile used for creating a slot boss or hole features

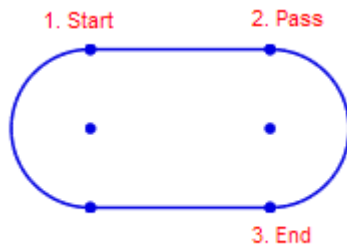
How to Use

How to draw Slots

1. Go to **Insert > Sketch** or click the icon in the toolbar. Select a base sketch plane to draw

sketches on in the **Sketch** mode. For more information, see [Insert > Sketch](#).

2. Go to **Tools > Sketch Entities > Slot** or click the icon in the toolbar.
3. Pick a starting point. Draw a line to a passing point and finish the drawing at the ending point.



4. Click the **OK** button to complete the command.

Options in Detail

Start – Shows the starting corner position of a slot to be generated.

Pass – Shows the passing corner position of a slot to be generated.

End – Shows the ending corner position of a slot to be generated.

Length – Shows the length between the center points of two arcs.

Radius – Shows the radius of an arc.

See Also

[Tools > Sketch Entities > Line](#)

[Tools > Sketch Entities > Rectangle](#)

[Tools > Sketch Entities > Parallelogram](#)

[Tools > Sketch Entities > Polygon](#)

Tools > Sketch Entities > Circle

What

The **Circle** command draws circles on a base sketch plane.



Why

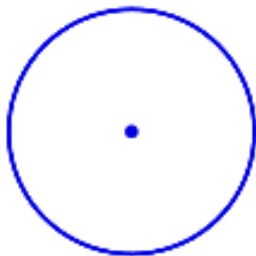
The **Circle** command is useful for:

- Drawing circles on a base sketch plane
- Creating a sketch profile to be used for creating circular or cylindrical features

How to Use

A. How to draw circles in Sketch mode

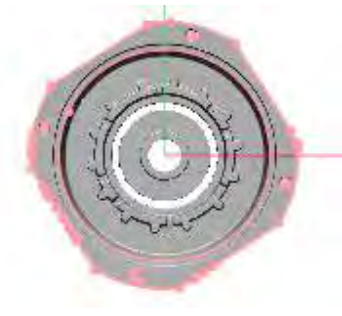
1. Go to **Insert > Sketch** or click the icon in the toolbar. Select a base sketch plane to draw sketches on in the **Sketch** mode. For more information, see [Insert > Sketch](#).
2. Go to **Tools > Sketch Entities > Circle** or click the icon in the toolbar.
3. Pick a center point.
4. Pick a point to define the radius of the circle to be generated.



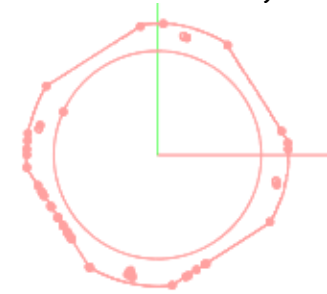
5. Click the **OK** button to complete the command.

B. How to draw circles on a section polyline in Mesh Sketch mode

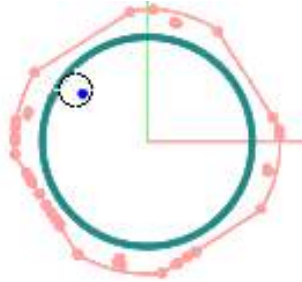
1. Go to **Insert > Mesh Sketch** or click the icon in the toolbar to enter the Mesh Sketch mode. Create section polylines using the **Mesh Sketch** mode. For more information, see [Insert > Mesh Sketch](#).



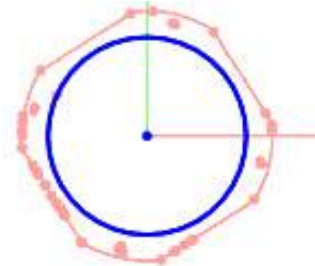
2. Hide mesh to make only section polylines visible in the **Model View**.



3. Go to **Tools > Sketch Entities > Circle** or click the icon in the toolbar.
4. Check the **Fit Polyline** option. Select section polylines and confirm the preview line by clicking the **Accept Fitting** button or double-clicking the left mouse button.



5. Click the **OK** button to complete the command.

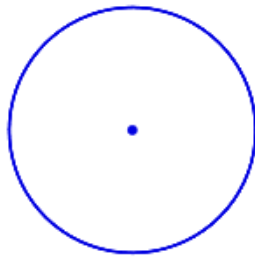


Options in Detail

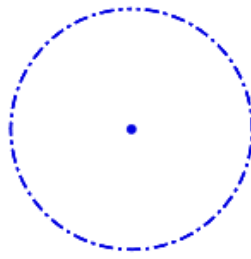
Center – Shows the center position of a circle to be generated.

Radius – Shows the radius of a circle to be generated.

For Construction – Creates a circle for construction. The construction circle can be used as a reference for drawing other sketch entities or a guide curve for constructing circular patterned features.

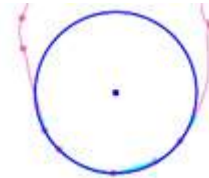
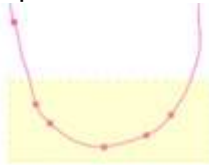


Normal Circle






For Construction Circle

Fit Polyline – Creates a circle by fitting to section polylines that are a section segment from a mesh or point cloud.



Note:

To accept a fitting, click the **Accept Fitting** button  or double-click the left mouse button. To cancel a fitting, click the **Cancel Fitting** button . To clear a selection, click the **Clear Selection** button .

See Also

[Tools > Sketch Entities > Perimeter Circle](#)

[Tools > Sketch Entities > 3 Tangent Circle](#)

[Tools > Sketch Entities > Centerpoint Arc](#)

[Tools > Sketch Entities > Tangent Arc](#)

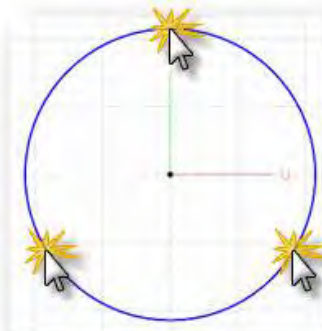
[Tools > Sketch Entities > 3 Point Arc](#)

[Tools > Sketch Entities > 3 Tangent Arc](#)

Tools > Sketch Entities > Perimeter Circle

What

The **Perimeter Circle** command draws circles constructed by 3 points on a base sketch plane.



Why

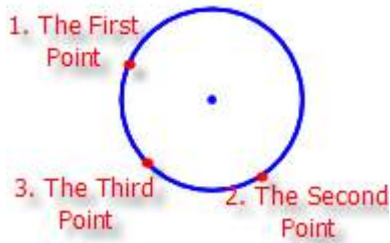
The **Perimeter Circle** command is useful for:

- Drawing circles by picking 3 points on a base sketch plane
- Creating a sketch profile to be used for creating circular or cylindrical features

How to Use

To draw circle by constructing with picking 3 points

1. Go to **Insert > Sketch** or click the icon in the toolbar . Select a base sketch plane to draw sketches in the **Sketch** mode. For more information, see [Insert > Sketch](#).
2. Go to **Tools > Sketch Entities > Perimeter Circle** or click the icon in the toolbar.
3. Pick a first point, a second point, and a third point to create a circle.



4. Click the **OK** button to complete the command.

Options in Detail

Center – Shows the center position of a circle to be generated.

Radius – Shows the radius of a circle to be generated.

See Also

[Tools > Sketch Entities > Circle](#)

[Tools > Sketch Entities > 3 Tangent Circle](#)

[Tools > Sketch Entities > Centerpoint Arc](#)

[Tools > Sketch Entities > Tangent Arc](#)

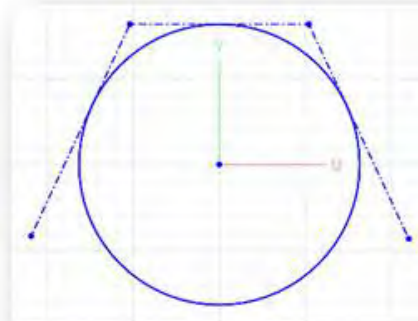
[Tools > Sketch Entities > 3 Point Arc](#)

[Tools > Sketch Entities > 3 Tangent Arc](#)

Tools > Sketch Entities > 3 Tangent Circle

What

The **3 Tangent Circle** command draws inscribed circles that touch the edges of 3 other sketch entities on a base sketch plane.



Why

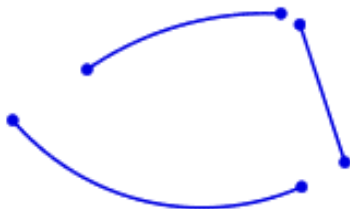
The **3 Tangent Circle** command is useful for:

- Drawing inscribed circles that touch the edges of 3 other sketch entities on a base sketch plane
- Creating a sketch profile to be used for creating circular or cylindrical features

How to Use

To draw inscribed circles that touch the edges of 3 other sketch entities

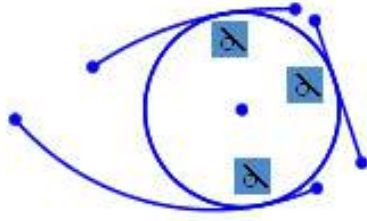
1. Go to **Insert > Sketch** or click the icon in the toolbar. Select a base sketch plane to draw sketches in the **Sketch** mode. For more information, see [Insert > Sketch](#).
2. Go to **Tools > Sketch Entities > 3 Tangent Circle** or click the icon in the toolbar.



Note:

To use this command, three sketch entities must exist.

3. Pick a line and two arcs.



4. Click the **OK** button to complete the command.

Options in Detail

Center – Shows the center position of a circle to be generated.

Radius – Shows the radius of a circle to be generated.

See Also

[Tools > Sketch Entities > Circle](#)

[Tools > Sketch Entities > Perimeter Circle](#)

[Tools > Sketch Entities > Centerpoint Arc](#)

[Tools > Sketch Entities > Tangent Arc](#)

[Tools > Sketch Entities > 3 Point Arc](#)

[Tools > Sketch Entities > 3 Tangent Arc](#)

Tools > Sketch Entities > Centerpoint Arc

What

The **Centerpoint Arc** command draws arcs on a base sketch plane.



Why

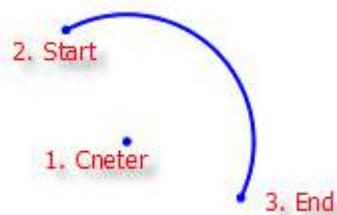
The **Centerpoint Arc** command is useful for:

- Drawing arcs on a base sketch plane
- Creating a sketch profile to be used for creating cylindrical features

How to Use

A. How to draw centerpoint arcs in Sketch mode

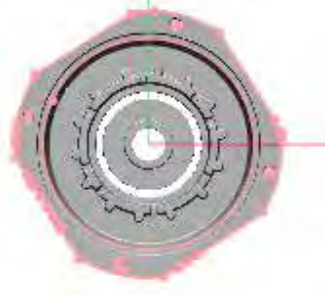
1. Go to **Insert > Sketch** or click the icon in the toolbar. Select a base sketch plane to draw sketches on in the **Sketch** mode. For more information, see [Insert > Sketch](#).
2. Go to **Tools > Sketch Entities > Centerpoint Arc** or click the icon in the toolbar.
3. Pick a center point.
4. Pick a starting and ending points for an arc.



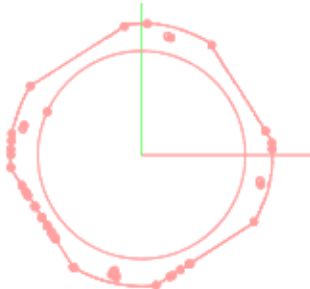
5. Click the **OK** button to complete the command.

B. How to draw centerpoint arcs on section polyline in Mesh Sketch mode

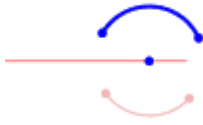
1. Go to **Insert > Mesh Sketch** or click the icon in the toolbar to enter the Mesh Sketch mode.
Create section polylines. For more information, see [Insert > Mesh Sketch](#).



2. Hide the mesh to make section polylines visible only in the Model View.



3. Go to **Tools > Sketch Entities > Centerpoint Arc** or click the icon in the toolbar.
4. Check the **Fit Polyline** option. Select section polylines and confirm the preview arc by clicking the **Accept Fitting** button or double-clicking the left mouse button.



Note:

Arcs can be created on partial section polylines by dragging and selecting segments with the **Fit Polyline** option.

5. Click the **OK** button to complete the command.

Options in Detail

Center – Shows the center position of an arc to be generated.

Radius – Shows the radius of an arc to be generated.

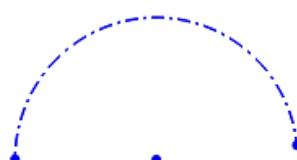
Start – Shows the starting position of an arc to be generated.

End – Shows the ending position of an arc to be generated.

For Construction – Creates an arc for construction. The construction arc can be used as a reference for drawing other sketch entities or a guide curve for constructing circular patterned features.



Normal Arc






For Construction Arc

Fit Polyline – Creates an arc by fitting to section polylines that are a section segment from a mesh or point cloud.



Note:

To accept a fitting, click the **Accept Fitting** button  or double-click the left mouse button. To cancel a fitting, click the **Cancel Fitting** button . To clear a selection, click the **Clear Selection** button .

See Also

[Tools > Sketch Entities > Circle](#)

[Tools > Sketch Entities > Perimeter Circle](#)

[Tools > Sketch Entities > 3 Tangent Circle](#)

[Tools > Sketch Entities > Tangent Arc](#)

[Tools > Sketch Entities > 3 Point Arc](#)

[Tools > Sketch Entities > 3 Tangent Arc](#)

Tools > Sketch Entities > Tangent Arc

What

The **Tangent Arc** command draws arc that can be continually connected to another sketch entity by a tangent constraint.



Why

The **Tangent Arc** command is useful for:

- Drawing arcs on a base sketch plane that are continually connected with other sketch entity by a tangent constraint
- Creating a sketch profile to be used for creating blended features with planar/cylindrical or cylindrical/cylindrical feature shapes

How to Use

How to draw a tangent arc that is continually connected to another sketch entity

1. Go to **Insert > Sketch** or click the icon in the toolbar. Select a base sketch plane to draw sketches on in the **Sketch** mode. For more information, see [Insert > Sketch](#).

2. Go to **Tools > Sketch Entities > Tangent Arc** or click the icon in the toolbar.

Note:

To use this command, sketch entities must exist.

3. Pick a point on another entity to set the selected point as a **Starting** point of an arc to be generated.



4. Pick an ending point and click the right mouse button to complete the construction of the arc.

Click the **OK** button to complete the command.



Options in Detail

Center – Shows the center position of an arc to be generated.

Radius – Shows the radius of an arc to be generated.

Start – Shows the starting position of an arc to be generated.

End – Shows the ending position of an arc to be generated.

See Also

[Tools > Sketch Entities > Circle](#)

[Tools > Sketch Entities > Perimeter Circle](#)

[Tools > Sketch Entities > 3 Tangent Circle](#)

[Tools > Sketch Entities > Centerpoint Arc](#)

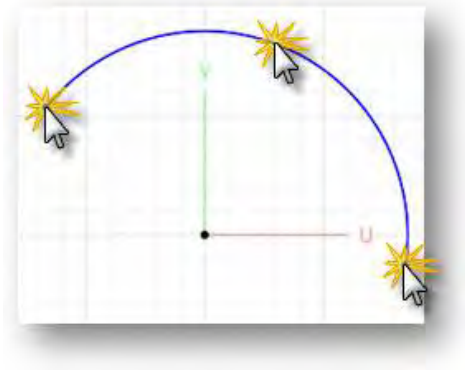
[Tools > Sketch Entities > 3 Point Arc](#)

[Tools > Sketch Entities > 3 Tangent Arc](#)

Tools > Sketch Entities > 3 Points Arc

What

The **3 Point Arc** command draws arcs constructed by three points on a base sketch plane.



Why

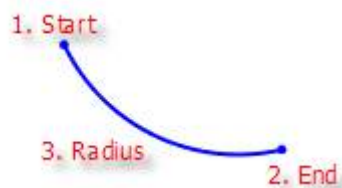
The **3 Point Arc** command is useful for:

- Drawing arcs on a base sketch plane
- Creating a sketch profile to be used for generating cylindrical features

How to Use

How to draw arcs by selecting 3 points

1. Go to **Insert > Sketch** or click the icon in the toolbar. Select a base sketch plane to draw sketches on in the **Sketch** mode. For more information, see [Insert > Sketch](#).
2. Go to **Tools > Sketch Entities > 3 Points Arc** or click the icon in the toolbar.
3. Pick a starting and ending point.
4. Pick a final point to set the radius of an arc.



Options in Detail

Center – Shows the center position of an arc to be generated.

Radius – Shows the radius of an arc to be generated.

Start – Shows the starting position of an arc to be generated.

End – Shows the ending position of an arc to be generated.

See Also

[Tools > Sketch Entities > Circle](#)

[Tools > Sketch Entities > Perimeter Circle](#)

[Tools > Sketch Entities > 3 Tangent Circle](#)

[Tools > Sketch Entities > Centerpoint Arc](#)

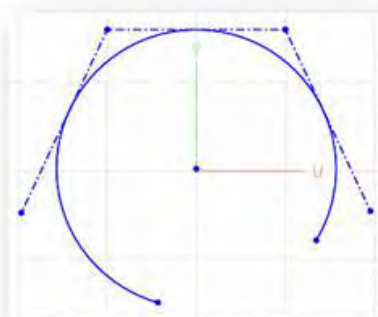
[Tools > Sketch Entities > Tangent Arc](#)

[Tools > Sketch Entities > 3 Tangent Arc](#)

Tools > Sketch Entities > 3 Tangent Arc

What

The **3 Tangent Arc** command draws arcs using an inscribed circle that touches the edges of 3 other sketch entities on a base sketch plane.



Why

The **3 Tangent Arc** command is useful for:

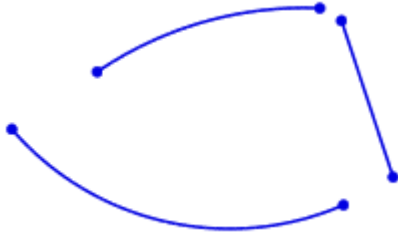
- Drawing inscribed arcs that touch the edges of 3 other sketch entities on a base sketch plane
- Creating a sketch profile to be used for generating cylindrical features

How to Use

How to draw inscribed arcs that touch the edges of 3 other sketch entities

1. Go to **Insert > Sketch** or click the icon in the toolbar. Select a base sketch plane to draw sketches on in the **Sketch** mode. For more information, see [Insert > Sketch](#).

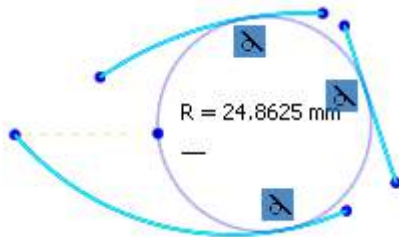
2. Go to **Tools > Sketch Entities > 3 Tangent Arc** or click the icon in the toolbar.



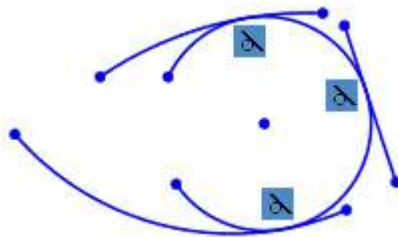
Note:

To use this command, sketch entities must exist.

3. Pick a line and two arcs.



4. Pick a starting and ending position on the preview of the inscribed circle.



5. Click the **OK** button to complete the command.

Options in Detail

Center – Shows the center position of an arc to be generated.

Radius – Shows the radius of an arc to be generated.

Start – Shows the starting position of an arc to be generated.

End – Shows the ending position of an arc to be generated.

See Also

[Tools > Sketch Entities > Circle](#)

[Tools > Sketch Entities > Perimeter Circle](#)

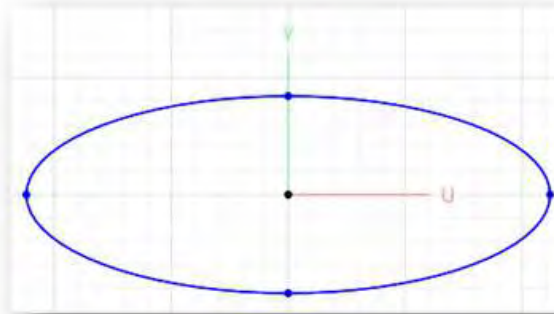
[Tools > Sketch Entities > 3 Tangent Circle](#)

[Tools > Sketch Entities > Centerpoint Arc](#)

Tools > Sketch Entities > Ellipse

What

The **Ellipse** command draws ellipses on a base sketch plane.



Why

The **Ellipse** command is useful for:

- Drawing ellipses on a base sketch plane

How to Use

How to draw ellipses

1. Go to **Insert > Sketch** or click the icon in the toolbar. Select a base sketch plane to draw sketches on in the **Sketch** mode. For more information, see [Insert > Sketch](#).
2. Go to **Tools > Sketch Entities > Ellipse** or click the icon in the toolbar.
3. Pick a center point of an ellipse to be generated.
4. Specify the radius of a major arc by picking a **Major** point and then specifying the radius of a minor arc by picking a **Minor** point.



5. Click the **OK** button to complete the command.

Options in Detail

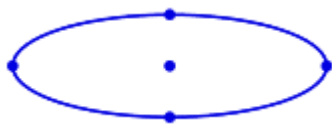
Center – Shows the center position of an ellipse to be generated.

Major Point – Shows the major arc position of an ellipse to be generated.

Minor Point – Shows the minor arc position of an ellipse to be generated.

Radius – Shows the major and minor arc radius of an ellipse to be generated.

For Construction – Creates an ellipse for construction. The construction ellipse can be used as a reference for drawing other sketch entities or a guide curve for constructing patterned features.

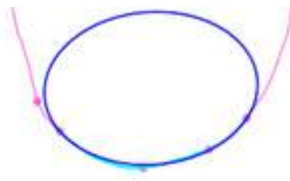


Normal Ellipse






For Construction Ellipse

Fit Polyline – Creates an ellipse by fitting to section polylines that are a section segment from a mesh or point cloud.



Note:

To accept a fitting, click the **Accept Fitting** button  or double-click the left mouse button. To cancel a fitting, click the **Cancel Fitting** button . To clear a selection, click the **Clear Selection** button .

See Also

[Tools > Sketch Entities > Partial Ellipse](#)

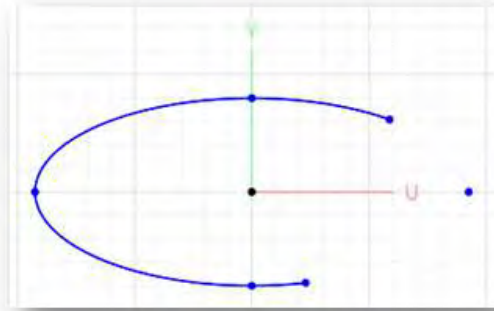
[Tools > Sketch Entities > Parabola](#)

[Tools > Sketch Entities > Spline](#)

Tools > Sketch Entities > Partial Ellipse

What

The **Partial Ellipse** command draws a partial ellipse on a base sketch plane.



Why

The **Partial Ellipse** command is useful for:

- Drawing a partial ellipses on a base sketch plane

How to Use

To draw partial ellipses

1. Go to **Insert > Sketch** or click the icon in the toolbar. Select a base sketch plane to draw sketches on in the **Sketch** mode. For more information, see [Insert > Sketch](#).
2. Go to **Tools > Sketch Entities > Partial Ellipse** or click the icon in the toolbar.
3. Pick a center point of an ellipse to be generated.
4. Specify the radius of a major arc by picking a **Major** point and then drag mouse cursor to specify the radius of a minor arc.
5. Pick a starting position and an ending position on the preview ellipse.



6. Click the **OK** button to complete the command.

Options in Detail

Center – Shows the center position of an ellipse to be generated.

Major Point – Shows the major arc position of an ellipse to be generated.

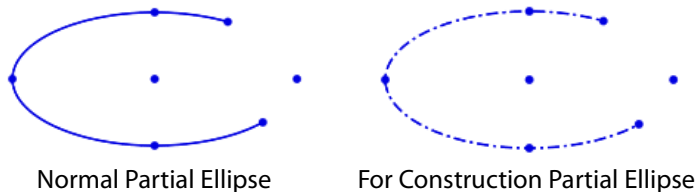
Minor Point – Shows the minor arc position of an ellipse to be generated.

Radius – Shows the major and minor arc radius of an ellipse to be generated.

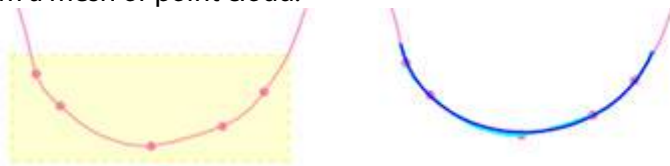
Start – Shows the starting position of a partial ellipse.

End – Shows the ending position of a partial ellipse.




For Construction – Creates an ellipse for construction. The construction ellipse can be used as a reference for drawing other sketch entities or as a guide curve for constructing patterned features.



Fit Polyline – Creates a partial ellipse by fitting to section polylines that are a section segment from a mesh or point cloud.



Note:

To accept a fitting, click the **Accept Fitting** button  or double-click the left mouse button. To cancel a fitting, click the **Cancel Fitting** button . To clear a selection, click the **Clear Selection** button .

See Also

[Tools > Sketch Entities > Ellipse](#)

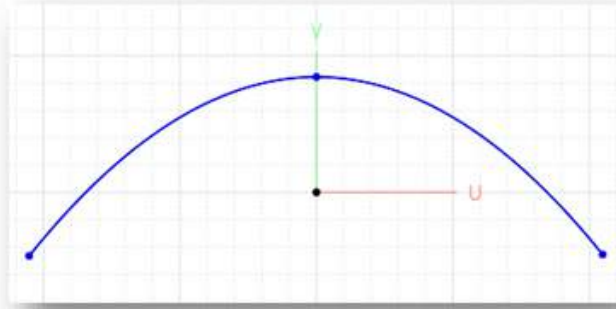
[Tools > Sketch Entities > Parabola](#)

[Tools > Sketch Entities > Spline](#)

Tools > Sketch Entities > Parabola

What

The **Parabola** command draws a parabola on a base sketch plane.



Why

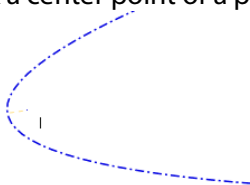
The **Parabola** command is useful for:

- Drawing parabolas on a base sketch plane

How to Use

How to draw parabolas

1. Go to **Insert > Sketch** or click the icon in the toolbar. Select a base sketch plane to draw sketches on in the **Sketch** mode. For more information, see [Insert > Sketch](#).
2. Go to **Tools > Sketch Entities > Parabola** or click the icon in the toolbar.
3. Pick a center point of a parabola to be generated.



4. Pick an apex position while previewing the parabola.



5. Pick a starting position and an ending position on the preview parabola.



6. Click the **OK** button to complete the command.

Options in Detail

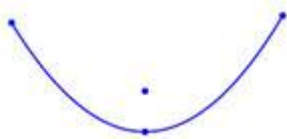
Center – Shows the center position of a parabola to be generated.

Apex – Shows the apex position of a parabola to be generated.

Start – Shows the starting position of a parabola to be generated.

End – Shows the ending position of a parabola to be generated.

For Construction – Creates a parabola for construction. The construction parabola can be used as a reference for drawing other sketch entities or as a guide curve for constructing patterned features.



Normal Parabola



For Construction Parabola

See Also

[Tools > Sketch Entities > Ellipse](#)

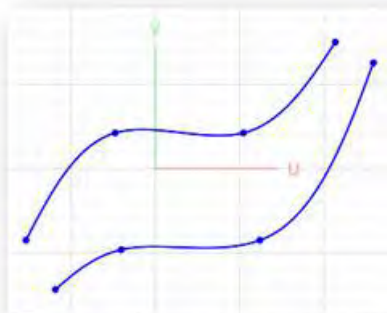
[Tools > Sketch Entities > Partial Ellipse](#)

[Tools > Sketch Entities > Spline](#)

Tools > Sketch Entities > Spline

What

The **Spline** command draws splines on a base sketch plane.



Why

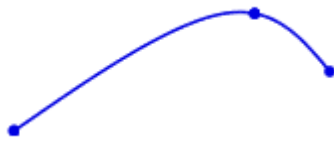
The **Spline** command is useful for:

- Drawing splines on a base sketch plane
- Creating a sketch profile to be used for creating freeform features

How to Use

A. How to draw splines in Sketch mode

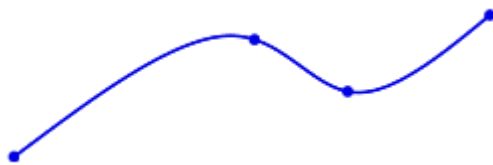
1. Go to **Insert > Sketch** or click the icon in the toolbar. Select a base sketch plane to draw sketches on in the **Sketch** mode. For more information, see [Insert > Sketch](#).
2. Go to **Tools > Sketch Entities > Spline** or click the icon in the toolbar.
3. Pick a starting point and draw a spline by picking more points.



Note:

Use the Ctrl + Z shortcut to cancel the last picked position.

4. Press the right mouse button to complete the construction of the spline. Click the **OK** button to complete the command.

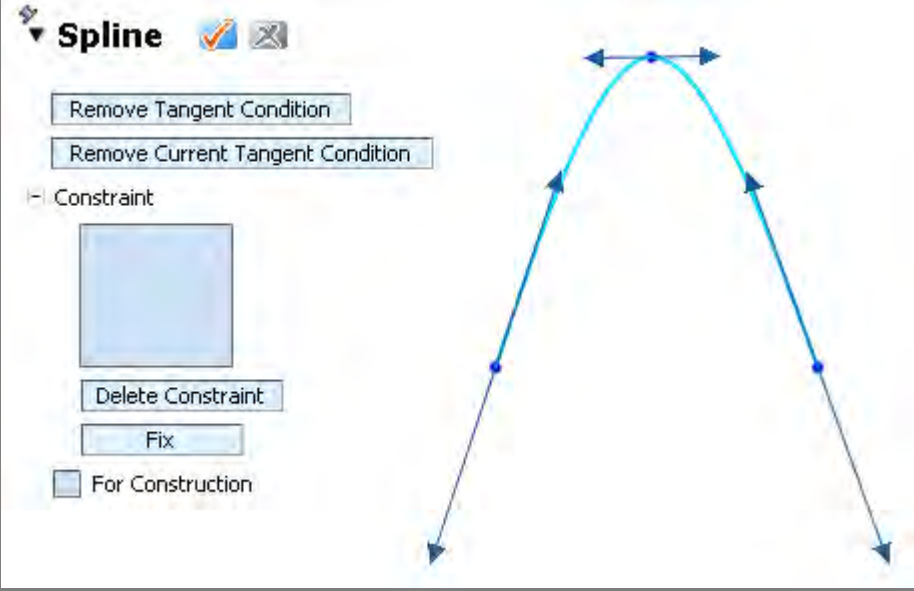


Note:

After drawing a spline, double-click the spline to manually edit it in the **Sketch** mode. Edit tangent conditions by dragging an arrow on the spline node points. The tangent constraint can be removed by clicking the **Remove Tangent Condition** in the spline edit dialog.

- To adjust the arrow direction while only changing the arrow length, or the tension of tangent condition, drag the arrow while holding down the **Shift** key.

- To adjust the arrow length while only changing its direction, drag the arrow while holding down the **Ctrl** key.



Remove Tangent Condition – Removes all edited tangent conditions.

Remove Current Tangent Condition – Removes selected tangent conditions.

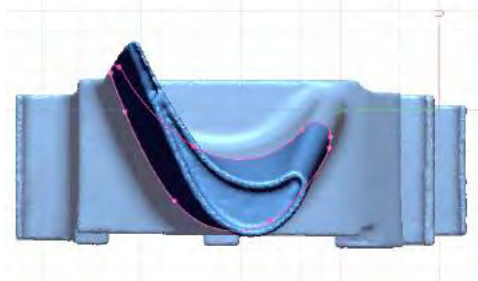
Delete Constraint – Removes selected constraints from the list.

Fix – Fixes a spline. This option does not edit tangent conditions. When the Fix constraint is given to a spline, the spline will get full constraints.

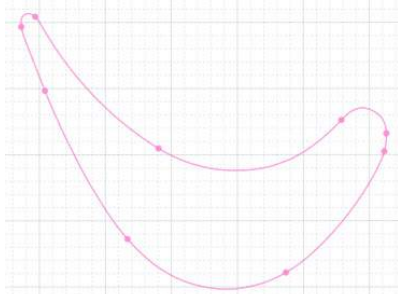
B. How to draw splines in Mesh Sketch mode

1. Go to **Insert > Mesh Sketch** or click the icon in the toolbar to enter the **Mesh Sketch** mode.

Create section polylines. For more information, see [Insert > Mesh Sketch](#).

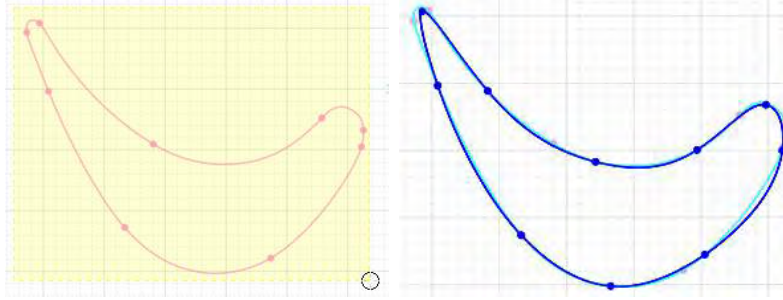


2. **Hide** the mesh to make only section polylines visible in the **Model View**.



3. Go to **Tools > Sketch Entities > Spline** or click the icon in the toolbar.

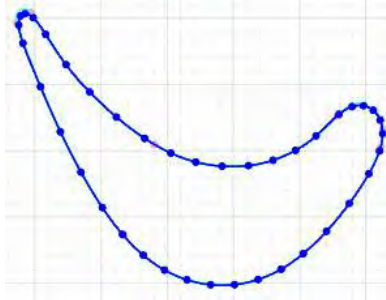
4. Check the **Fit Polyline** option and select section polylines.



Note:

The Fit Polyline option is only available in the **Mesh Sketch** mode.

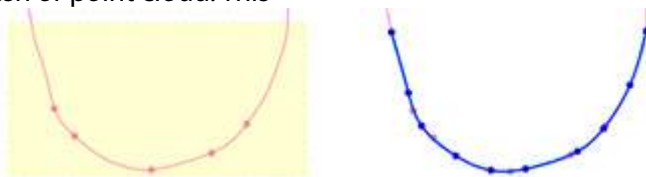
5. Increase No. Of Interpolation Points to **40**.






6. Confirm the preview spline by clicking the **Accept Fitting** button or by double-clicking the left mouse button.
7. Click the **OK** button to complete the command.

Options in Detail

Fit Polyline – Creates a spline by fitting to section polylines that are a section segment from a mesh or point cloud. This



Note:

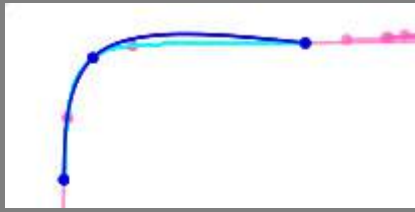
To accept a fitting, click the **Accept Fitting** button  or double-click the left mouse button. To cancel a fitting, click the **Cancel Fitting** button . To clear a selection, click the **Clear Selection** button .

Fitting Options – Specifies fitting options.

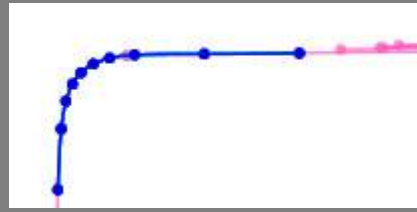
No. Of Interpolation Point – Specifies the number of interpolation points in a spline.

Note:

A smaller number of interpolation points may not accurately capture the shape of a section polyline, so an adaptable number of interpolation points can be used.



3 Points



10 Points

Smoothness – Specifies the overall smoothness level of a spline by moving the slider between **Min.** and **Max.** Smoothness is related to the curvature of a spline.

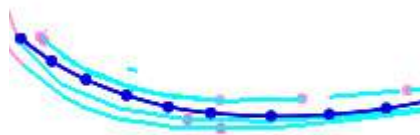


Smoothness: Min

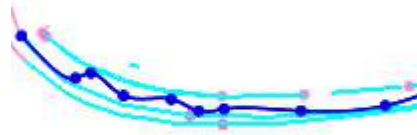


Smoothness: Max

Merge Range – Specifies the merging range of segment polylines. To make a smooth spline from overlapped segment polylines, set a larger value than the width of selected segment polylines.



Merge Range: 0.5



Merge Range: 0.05

Local Smoothness – Specifies a local smoothness level of a spline by moving the slider between **Min.** and **Max.**



Section Segment Polyline



Local Smoothness: Min

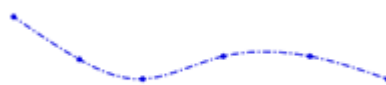


Local Smoothness: Max

For Construction – Creates a spline for construction. The construction spline can be used as a reference for drawing other sketch entities or as a guide curve for constructing loft or sweep features. For more information, see [Insert > Solid > Sweep](#) or [Insert > Solid > Loft](#).



Normal Spline



For Construction Spline

Dynamic View Change – Traces the last selected point in the **2D Sketch View** so that the last selected position is always in the center of the screen.

See Also

[Tools > Sketch Entities > Ellipse](#)

[Tools > Sketch Entities > Partial Ellipse](#)

[Tools > Sketch Entities > Parabola](#)

Tools > Sketch Entities > Centerline

What

The **Centerline** command draws lines for construction.



Why

The **Centerline** command is useful for:

- Drawing lines for construction on a base sketch plane
- Creating construction lines that can be used as a reference for drawing other sketch entities or an axis for constructing revolved features

How to Use

How to draw centerlines

1. Go to **Insert > Sketch** or click the icon in the toolbar. Select a base sketch plane to draw sketches on the base sketch plane in the **Sketch** mode. For more information, see [Insert > Sketch](#).
2. Go to **Tools > Sketch Entities > Centerline** or click the icon in the toolbar.
3. Pick a starting and ending point to create a centerline.

4. Click the right mouse button to complete the construction.
5. Click the **OK** button to complete the command.

Options in Detail

Start – Shows the starting position of a line to be generated.

End – Shows the ending position of a line to be generated.

For Construction – Creates a line used for constructing a solid or surface body that cannot be used for a profile. This option is checked on by default. To create a normal line, check this option off.



Centerline



Normal Line

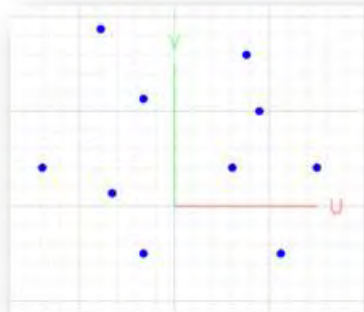
See Also

[Tools > Sketch Entities > Line](#)

Tools > Sketch Entities > Point

What

The **Point** command adds a point on a base sketch plane.



Why

The **Point** command is useful for:

- Adding points on base sketch plane
- Creating a point that can be used as a reference for drawing other sketch entities or a reference point for constructing geometric features

How to Use

How to add points

1. Go to **Insert > Sketch** or click the icon in the toolbar. Select a base sketch plane to draw sketches on in the **Sketch** mode. For more information, see [Insert > Sketch](#).
2. Go to **Tools > Sketch Entities > Point** or click the icon in the toolbar.
3. Select a point.
4. Click the **OK** button to complete the command.

Options in Detail

Position – Shows the position of a point to be generated

X – Shows the X coordinate of a point.

Y – Shows the Y coordinate of a point.

See Also

[Insert > Ref.Geometry > Point](#)

Tools > Sketch Entities > Text

What

The **Text** command adds text on a base sketch plane.

Enjoy
Rapidform
Software

Why

The **Text** command is useful for:

- Add text sketches on a base sketch plane
- Engraving text, symbols, or pattern shapes on features

How to Use

How to add text

1. Go to **Insert > Sketch** or click the icon in the toolbar. Select a base sketch plane to draw sketches on in the **Mesh Sketch** mode. For more information, see [Insert > Sketch](#).
2. Go to **Tools > Sketch Entities > Text** or click the icon in the toolbar.
3. Type text in the **Text** box and click **Enter** to see a preview.



4. Adjust the size of text by specifying the **Height**, **Width Ratio**, and **Space Ratio** options.

Note:

The size of text can be interactively adjusted by manipulating the control points in the **Model View**.

5. Click the **OK** button to complete the command.

Options in Detail

Text – Inputs text.

Size – Specifies the size of text.

Height – Specifies the height of text. The height can be adjusted by manipulating the green control points in the **Model View**.

Width Ratio – Specifies the width ratio of text. The width ratio can be adjusted by manipulating the red control points in the **Model View**.

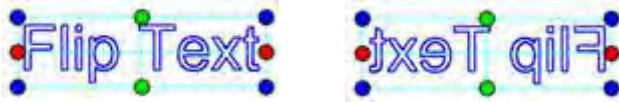
Space Ratio – Specifies the ratio between each letter.

Font – Specifies a font style.

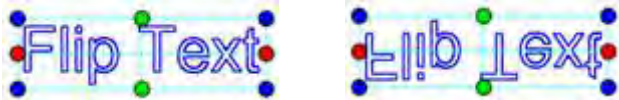
Position & Align – Specifies the position of text and aligns it to a specific axis or curve.

Position – Shows the center position of text. The center point of text can be adjusted by dragging the text box in the **Model View**.

Flip Horizontal – Flips text horizontally.



Flip Vertical – Flip text vertically.



Along A Curve – Aligns text along a curve. A Ref. Vector, Curve or Edge can be selected as a guide curve.



See Also

[Tools > Sketch Entities > Line](#)

[Tools > Sketch Entities > Circle](#)

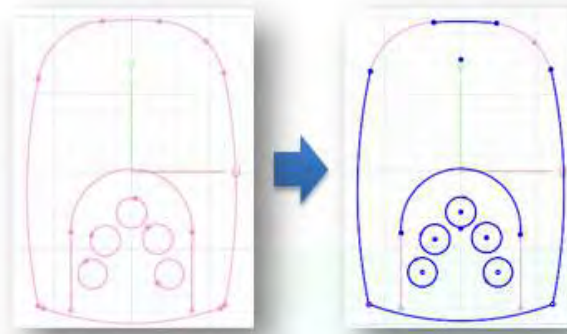
[Tools > Sketch Entities > Tangent Arc](#)

[Tools > Sketch Entities > Spline](#)

Tools > Sketch Entities > By Fitting Polyline

What

The **By Fitting Polyline** command creates lines, arcs or circles by fitting section polylines on a base sketch plane. This command is available in the **Mesh Sketch** mode only.



Why

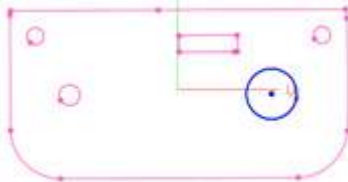
The **By Fitting Polyline** command is useful for:

- Creating lines, arcs or circles by simultaneously fitting them to a section polyline.

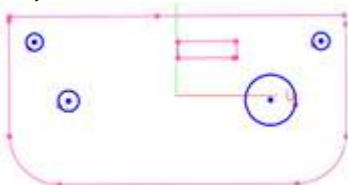
How to Use

How to create circle or arc sketch entities by fitting section polylines in the Mesh Sketch mode

1. Go to **Insert > Mesh Sketch** or click on the icon in the toolbar to enter the **Mesh Sketch** mode.
Create section polylines. For more information, see [Insert > Mesh Sketch](#).
2. Go to **Tools > Sketch Entities > By Fitting Polyline** or click on the icon in the toolbar.
3. Choose the **Circle** option. Select circular section polylines and click the **Accept Fitting** button to accept the preview.



4. Using the same way procedure as **Step 3**, complete remaining circular section polylines.



5. Click the **OK** button to complete the command.

Options in Detail

Fitting – Fits a sketch entity to a current segment type.

Note:

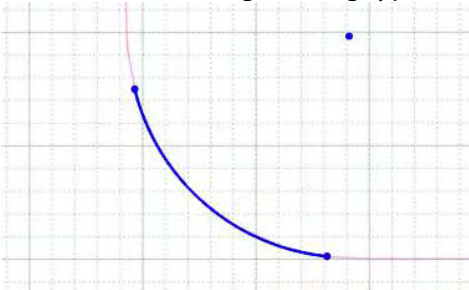
A segment type is divided into line and circle types. When the type of a selected segment is detected as a circle, a sketch entity can be fitted to circle or arc.

Fit Circle Type Segment To – Choose a fitting type for circular type segments.

Circle – Defines the target fitting type as a circle.



Arc - Defines the target fitting type as an arc.



See Also

[Tools > Sketch Entities > Line](#)

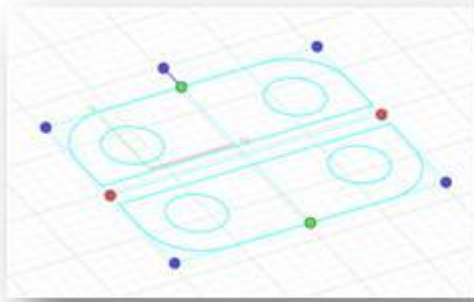
[Tools > Sketch Entities > Circle](#)

[Tools > Sketch Entities > Centerpoint Arc](#)

Tools > Sketch Entities > By Import

What

The **By Import** command imports 2D CAD drawing files as sketch entities onto a base sketch plane.



Why

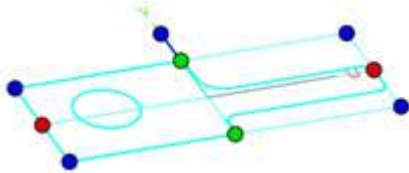
The **By Import** command is useful for:

- Importing 2D CAD drawing file as sketch entities onto a base sketch plane.

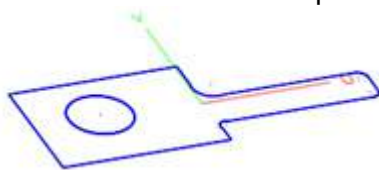
How to Use

How to import 2D CAD drawing files as sketch entities onto a base sketch plane

1. Go to **Tools > Sketch Entities > By Import** in the menu.
2. Select a 2D CAD drawing file in the DXF, IGES, STEP, or VDA file formats.
3. Define size and position.



4. Click the **OK** button to complete the command.

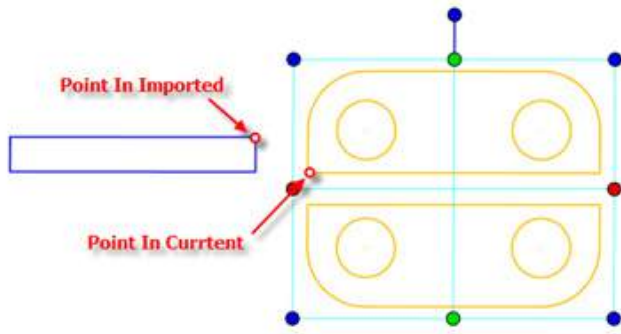


Options in Detail

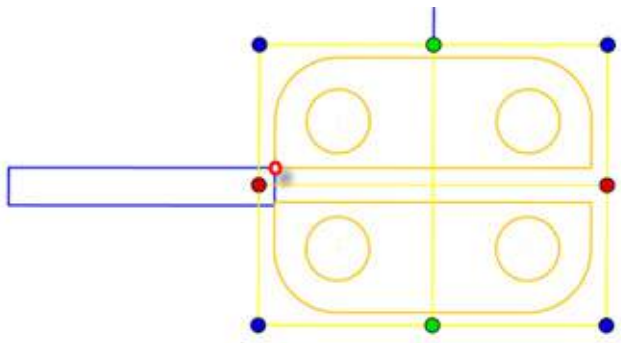
Flip Horizontally – Flips an imported sketch horizontally.

Flip Vertically – Flips an imported sketch vertically.

Anchor – Moves an imported sketch to an existing sketch entity by using anchor points. Pick two points on an imported sketch and the existing sketch to match them. If matching points need to be offset in the X and Y directions, specify the values in the **Offset From Target** option.

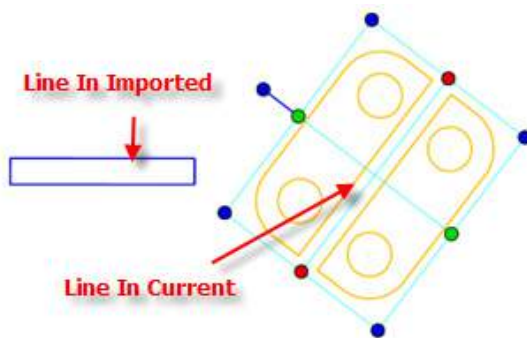


Before Applying an Anchor

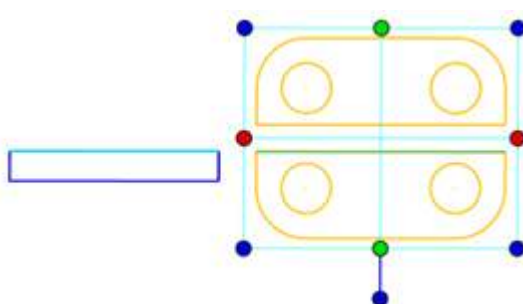


After Applying an Anchor

Align – Aligns an imported sketch to an existing model by matching lines. Select a line on an imported sketch as a **Moving Line** and select a linear entity on the existing model as a **Target Line**.



Before Align



After Align

Rotation From Target – Rotates an imported sketch using a target as a rotational point. This option is available after a Moving Line and a Target Line are defined.

Scale – Specifies a scale factor used to scale an imported sketch.

See Also

[Tools > Sketch Entities > Line](#)

[Tools > Sketch Entities > Circle](#)

[Tools > Sketch Entities > Centerpoint Arc](#)

7.5.7. Tools > Sketch Tools

The **Sketch Tools** menu features various commands used to edit sketch entities such as **Fillet**, **Chamfer**, **Trim**, for completely constructing sketch profile.

The **Sketch Tools** menu provide 2 sets of sketch edit tools, a set of useful pattern copy tools, and a set of converting tools used to:

Modify Sketch Entities

- **Fillet**
Modifies sketch entities by adding a tangent arc in between the sketch entities
- **Chamfer**
Modifies sketch entities by adding an angled line in between the sketch entities
- **Offset**
Offsets sketch entities by a specified distance
- **Convert Entities**
Converts other entities into sketch entities
- **Silhouette**
Converts outer boundaries of features into sketch entities

Edit Sketch Entities

- **Trim**
Trims sketch entities
- **Extend**
Extends sketch entities
- **Split**
Splits a sketch entity into multiple parts
- **Mirror**
Mirrors a part of a sketch entity

- **Resize**
Resizes a sketch entity
- **Merge**
Merges sketch entities into a single sketch entity

Adding Patterned Sketch Entities

- **Linear Pattern**
Creates linear patterns from sketch entities
- **Circular Pattern**
Creates circular patterns from sketch entities

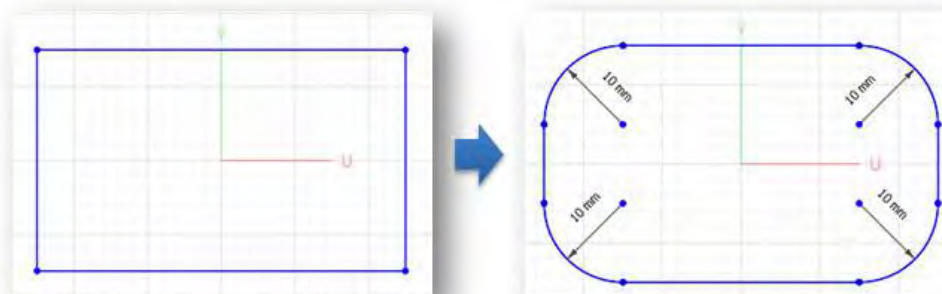
Converting Sketch Entities

- **Fit To Spline**
Converts sketch entities into splines
- **Text To Splines**
Converts text sketch entities into splines
- **Insert Polyline**
Inserts a Ref.Polyline onto a base sketch plane as a section polyline

Tools > Sketch Tools > Fillet

What

The **Fillet** command modifies sketch entities by adding a tangent arc in between sketch entities.



Why

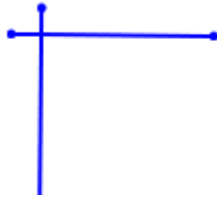
The **Fillet** command is useful for:

- Modifying sketch entities by adding a tangent arc in between the sketch entities
- Constructing a sketch profile for adding rounded features when creating a surface or solid body

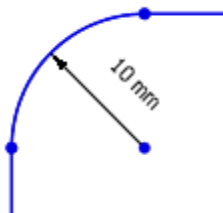
How to Use

How to modify sketch entities by adding a tangent arc in between the sketch entities

1. Go to **Tools > Sketch Tools > Fillet** or click the icon in the toolbar.



2. Check the **Specify Value** option and specify **10mm** as the fillet value.
3. Select sketch entities to add a fillet between by a specified radius.



4. Click the **OK** button to complete the command.

Options in Detail

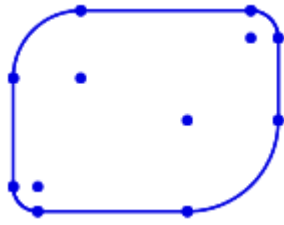
Specify Value – Adds a fillet radius between selected sketch entities.

Note:

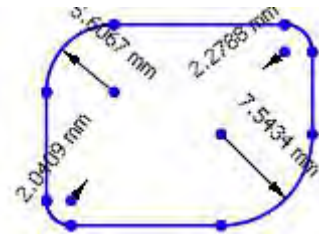
The Fillet command interactively adds fillets by dragging a tangent arc to be generated. Select a first sketch entity and select a second sketch entity while holding the left mouse button and drag the tangent arc to interactively add fillet.



Add Dimension – Adds dimensions after filleting.



Without Add Dimension



With Add Dimension

Note:

Once placed, dimensions can be removed if necessary.

See Also

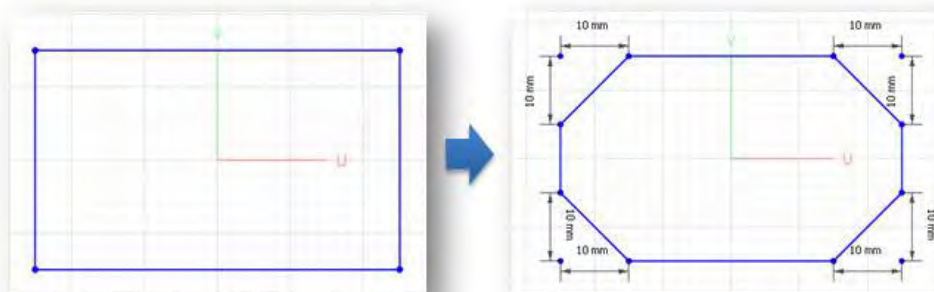
[Tools > Sketch Entities > Tangent Arc](#)

[Tools > Sketch Tools > Chamfer](#)

Tools > Sketch Tools > Chamfer

What

The **Chamfer** command modifies sketch entities by adding an angled line between the sketch entities.



Why

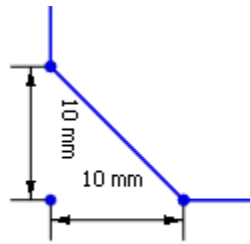
The **Chamfer** command is useful for:

- Modifying sketch entities by adding an angled line between the sketch entities

How to Use

How to modify sketch entities by adding angled line in between the sketch entities

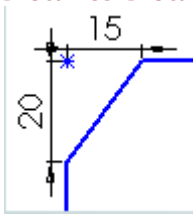
1. Go to **Tools > Sketch Tools > Chamfer** or click on the icon in the toolbar.
2. Choose **Distance-Distance** as the chamfer type.
3. Select sketch entities to add chamfer to between the sketch entities.



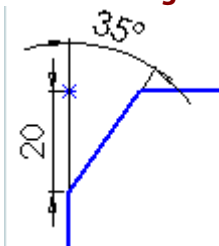
4. Click the **OK** button to complete the command.

Options in Detail

Distance-Distance – Adds chamfer by using distances.



Distance-Angle – Adds chamfer by angle and distance.



Specify Value – Specifies the position values for a chamfer to be placed. When this option is disabled, the chamfer will be placed at the position of selected sketch entities.

Distance1 – Specifies the distance of a chamfer from the first selected sketch entity.

Distance2 – Specifies the distance of a chamfer from the second selected sketch entity.

Angle – Specifies an angle of a chamfer from the first selected sketch entity.

See Also

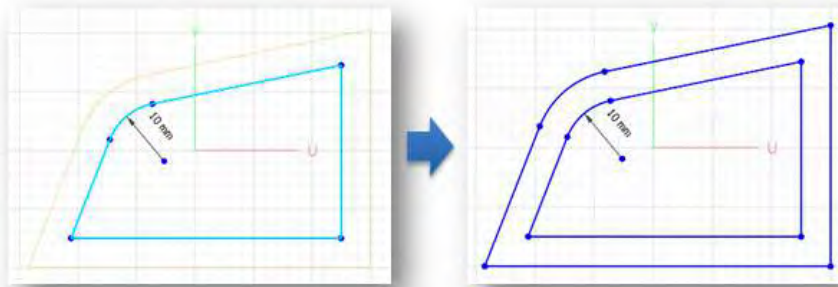
[Tools > Sketch Entities > Line](#)

[Tools > Sketch Tools > Fillet](#)

Tools > Sketch Tools > Offset

What

The **Offset** command offsets sketch entities by a specified distance.



Why

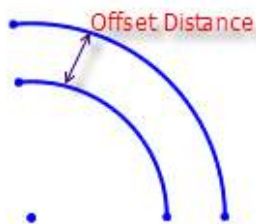
The **Offset** command is useful for:

- Offsetting sketch entities by a specified distance
- Constructing sketch profiles for creating solid bodies with constant thickness

How to Use

How to offset sketch entities by a specified distance

1. Go to **Tools > Sketch Tools > Offset** or click the icon in the toolbar.
2. Select sketch entities as target **Profiles**.
3. Specify **5mm** as the **Offset Distance** and choose the **Direction 1** option.



4. Click the **OK** button to complete the command.

Options in Detail

Select Entities

Profiles – Select sketch entities. When sketch entities are connected by a coincidence constraint, those sketches will be offset as a sketch loop.

Note:

The offset command is unavailable for splines, ellipses and parabola sketch entities.

Options

Distance – Specifies an offset distance.

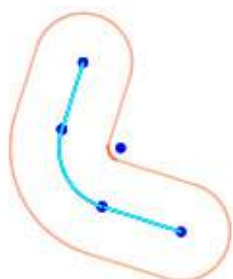
Direction – Specifies an offset direction.

Direction 1 – Selects the default positive direction to offset.

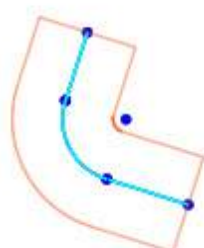
Direction 2 – Selects the opposite, negative direction from **Direction 1**.

Both Side – Offsets sketch entities in both two directions.

Cap Ends – Closes the top and bottom sides of an offset using lines or arcs. This option is available when the **Both Side** option is used.



Cap Ends: Arcs



Cap Ends: Lines

More Options

Remove Original Sketch – Removes original sketch entities after offsetting.

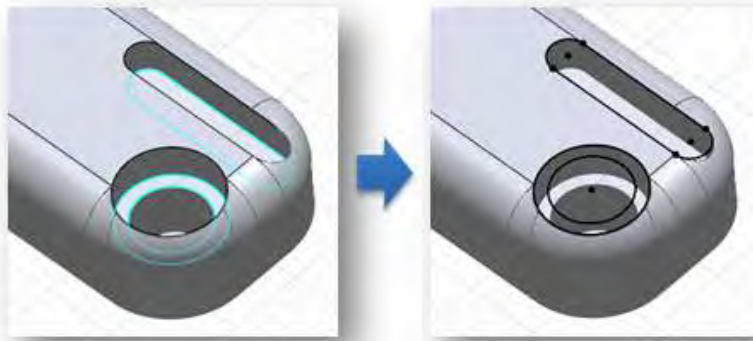
See Also

[Tools > Sketch Tools > Extend](#)

Tools > Sketch Tools > Convert Entities

What

The **Convert Entities** command converts entities into sketch entities by projecting them onto a base sketch plane.



Why

The **Convert Entities** command is useful for:

- Converting other entities into sketch entities using a base sketch plane

How to Use

How to convert other entities into sketch entities

1. Go to **Tools > Sketch Tools > Convert Entities** or click the icon in the toolbar.
2. Select an edge of a body as a target **Entities**.



3. Click the **OK** button to complete the command. Check that the selected edges are converted into sketch entities on the base sketch plane.

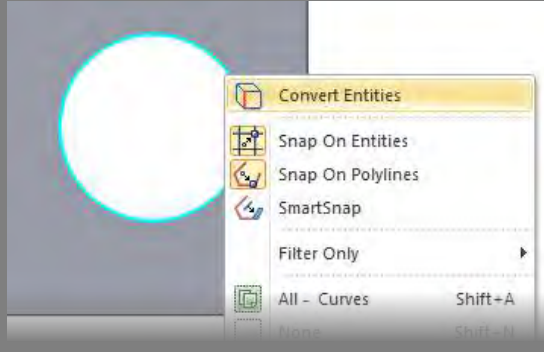


Options in Detail

Entities – Selects entities that are needed to be converted into sketch entities. Curve points, Ref.Points, Ref.Vectors, Curves, Vertices, and Edges can be target entities.

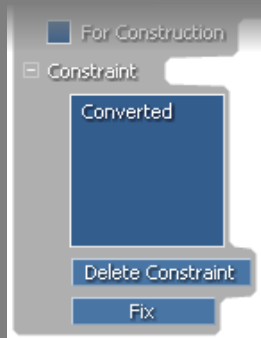
Note:

The **Convert Entities** command can be directly used in the pop-up menu. Select other entities, click the right mouse button and choose the **Convert Entities** option in the pop-up menu to convert selected entities into sketch entities.



Note:

Sketch entities converted by the Convert Entities command are fixed and cannot be modified directly. To edit the converted sketch entities, double-click them after converting entities into sketch entities and deleting the Converted constraints by using the Delete Constraint button.



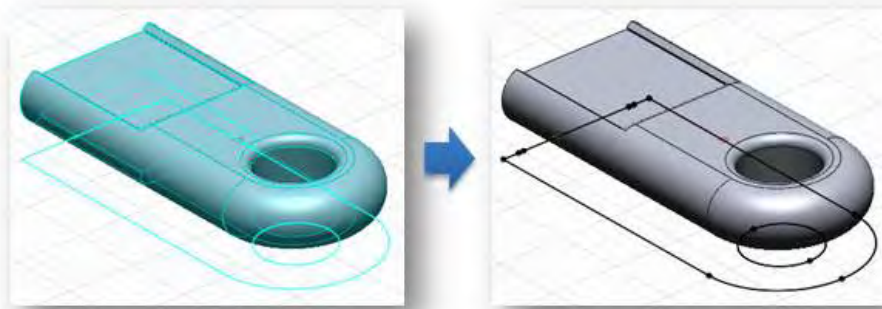
See Also

[Tools > Sketch Tools > Silhouette](#)

Tools > Sketch Tools > Silhouette

What

The **Silhouette** command converts outer boundaries of features into sketch entities on a base sketch plane.



Why

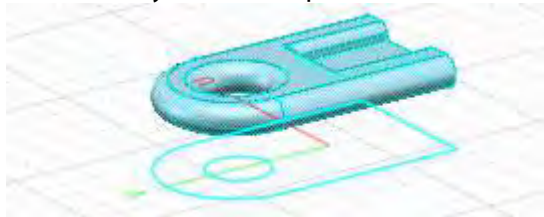
The **Silhouette** command is useful for:

- Converting outer boundaries of features into sketch entities on a base sketch plane.

How to Use

How to convert outer boundary of features into sketch entities

1. Go to **Tools > Sketch Tools > Silhouette** in the menu.
2. Select a body. Check the preview sketches.



3. Click the **OK** button to complete the command. Check that the outer boundary of the selected body is converted into sketch entities on the base sketch plane.



Options in Detail

Entities – Selects faces or solid/surface bodies as target **Entities**.

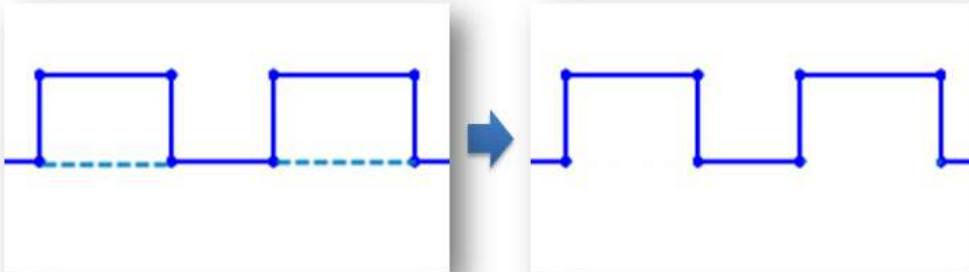
See Also

[Tools > Sketch Tools > Convert Entities](#)

Tools > Sketch Tools > Trim

What

The **Trim** command trims sketch entities.



Why

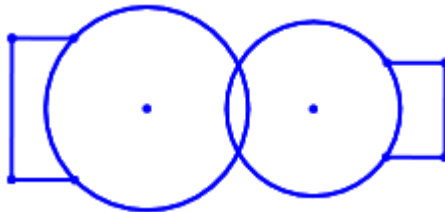
The **Trim** command is useful for:

- Trimming sketch entities and removing unnecessary parts of sketch entities

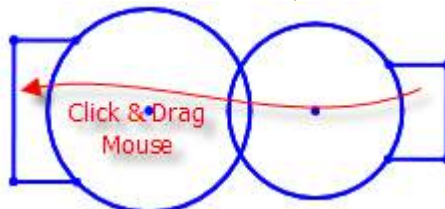
How to Use

A. How to trim sketch entities by using the Power Trim method

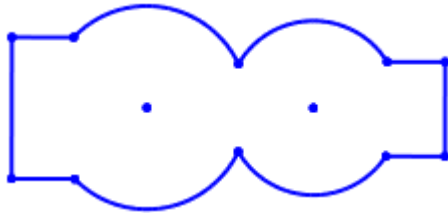
1. Go to **Tools > Sketch Tools > Trim** in the menu.



2. Choose the **Power Trim** method and click-drag the left mouse button over sketch entities to remove unnecessary parts.

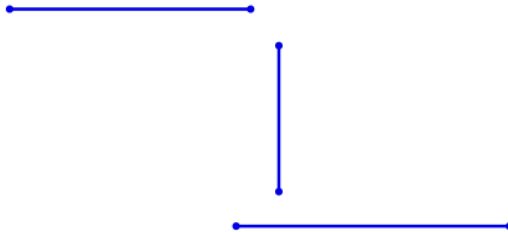


3. Check that the sketch entities are trimmed. Click the **OK** button to complete the command.

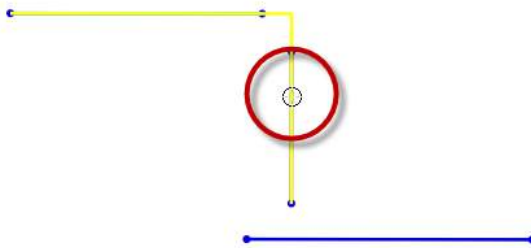


B. How to trim sketch entities by using the Corner Trim method

1. Go to **Tools > Sketch Tools > Trim** or click the icon in the toolbar.



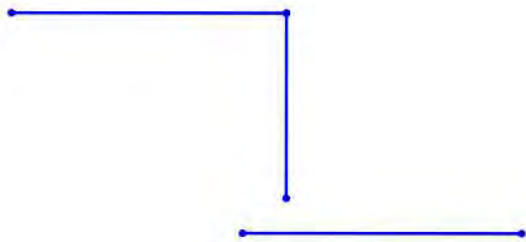
2. Choose the **Corner Trim** method and select the upper horizontal and vertical lines to extend those lines at their intersection point. The result will be highlighted when selecting a second line.



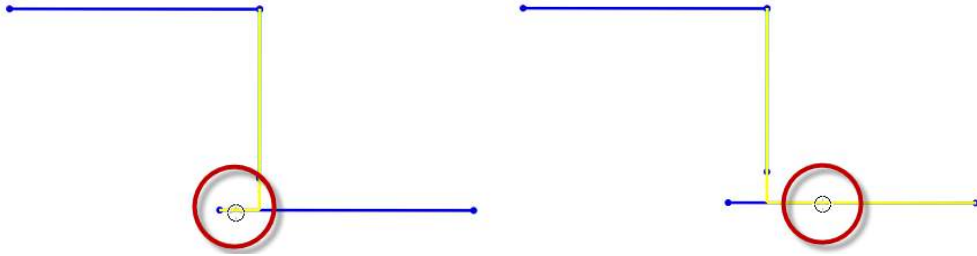
Note:

The **Corner Trim** method can also be used by click-dragging.

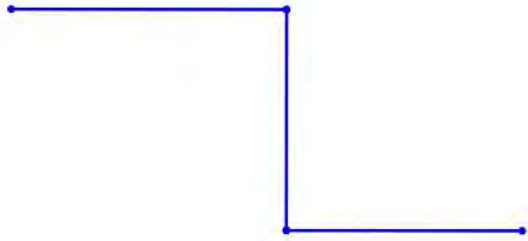
3. Check the results.



4. Select the lower horizontal and vertical line to extend those lines at their intersection point. The result will be highlighted when selecting a second line depending on where the mouse cursor is placed.



5. Check the results. The results assume that the second line was selected when the mouse cursor was placed in the right side of the vertical line.

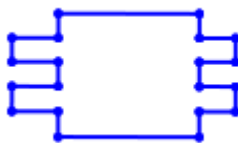


Options in Detail

Power Trim – Trims multiple and adjacent sketch entities by click-dragging the left mouse button across sketch entities.

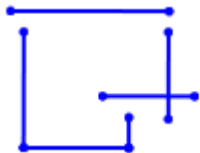


Before Power Trim

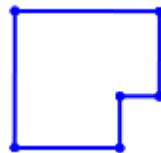


After Power Trim

Corner Trim – Extends sketch entities so that they can be intersected with each other and trims sketch entities.



Before Corner Trim



After Corner Trim

See Also

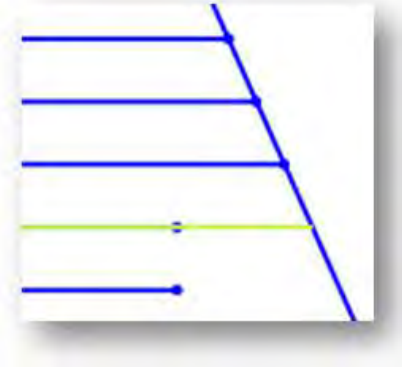
[Tools > Sketch Tools > Extend](#)

[Tools > Sketch Tools > Split](#)

Tools > Sketch Tools > Extend

What

The **Extend** command extends a sketch entity until it meets another sketch entity.



Why

The **Extend** command is useful for:

- Extending a sketch entity until it meets another sketch entity

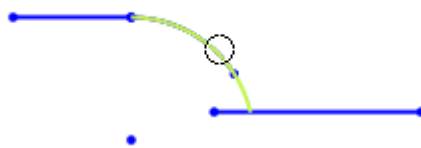
How to Use

How to extend a sketch entity

1. Go to **Tools > Sketch Tools > Extend** or click the icon in the toolbar.



2. Hover the mouse cursor over a sketch entity and check the preview.



3. Click on the highlighted sketch entity to extend it.



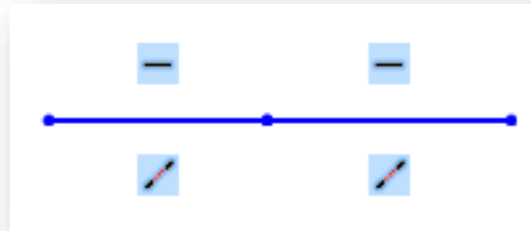
See Also

[Tools > Sketch Tools > Resize](#)

Tools > Sketch Tools > Split

What

The **Split** command splits a sketch entity into multiple parts.



Why

The **Split** command is useful for:

- Splitting a sketch entity into multiple parts

How to Use

How to split a sketch entity into multiple parts

1. Go to **Tools > Sketch Tools > Split** or click the icon in the toolbar.



2. Hover the mouse cursor over a sketch entity and check the position to be split.



3. Click a specific position on the highlighted sketch entity and check the results.



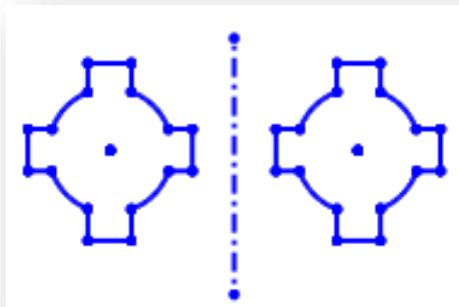
See Also

[Tools > Sketch Tools > Trim](#)

Tools > Sketch Tools > Mirror

What

The **Mirror** command mirrors a part of a sketch entity.



Why

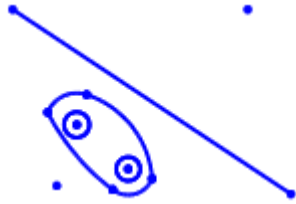
The **Mirror** command is useful for:

- Mirroring a part of a sketch entity
- Copying sketch entities symmetrically

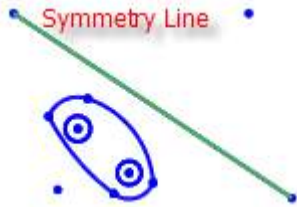
How to Use

How to mirror a part of a sketch entity

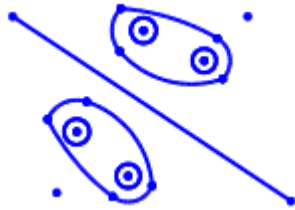
1. Go to **Tools > Sketch Tools > Mirror** or click the icon in the toolbar.



2. Select a line as a **Symmetry Line** and target entities to be mirrored as **Entities**.



3. Click the **OK** button to complete the command.



Options in Detail

Symmetry Line – Selects a symmetry line to use during mirroring.

Entities – Selects sketch entities to be mirrored.

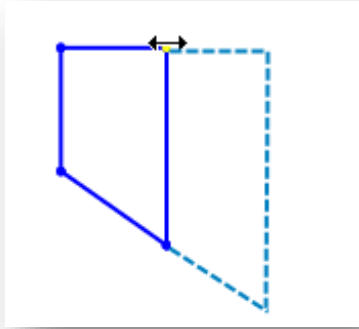
See Also

[Tools > 3D Sketch Tools > Mirror](#)

Tools > Sketch Tools > Resize

What

The **Resize** command resizes a sketch entity.



Why

The **Resize** command is useful for:

- Resizing the length of a sketch entity while maintaining its direction

How to Use

How to resize a sketch entity

1. Go to **Tools > Sketch Tools > Resize** or click the icon in the toolbar.



2. Select a starting or an ending point of a sketch entity and click-drag it to resize its length.



3. Click the **OK** button to complete the command.

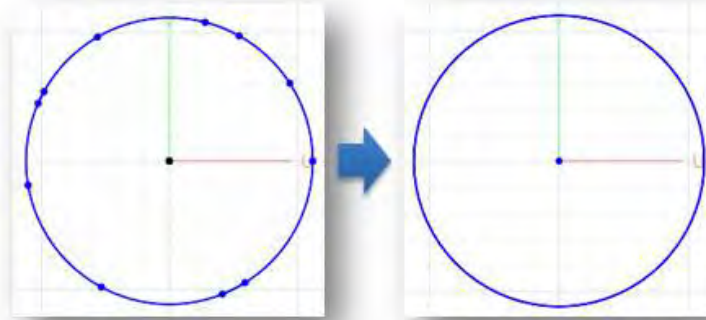


See Also

Tools > Sketch Tools > Merge

What

The **Merge** command merges multiple sketch entities into a single sketch entity.



Why

The **Merge** command is useful for:

- Merging separated sketch entities into a single sketch entity
- Joining split sketch entities created by the Auto Sketch command

How to Use

How to merges multiple sketch entities into a single sketch entity

1. Go to **Tools > Sketch Tools > Merge** or click the icon in the toolbar.



2. Select split sketch entities.



3. Click the **OK** button to complete the command.



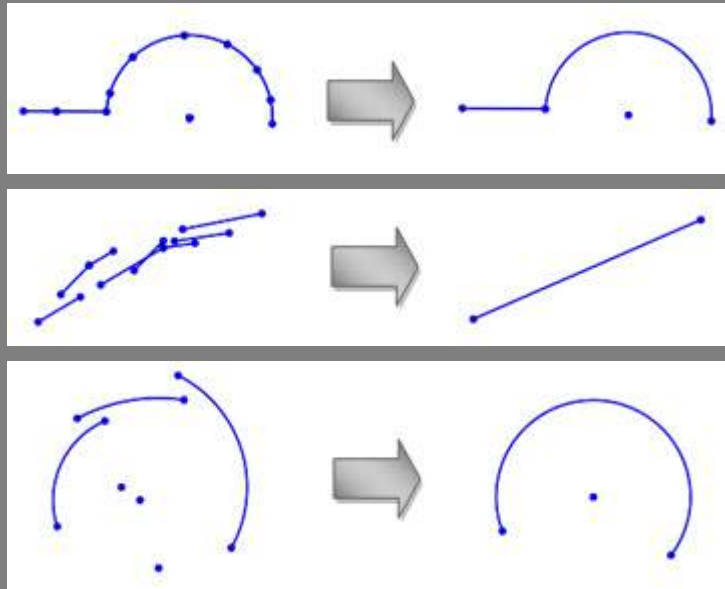
Options in Detail

Entities – Selects split sketch entities to merge them into a single sketch entity.

Note:

When select entities are split and connected, they will be merged and the points at the split position will be removed. When a set of separated sketch entities are selected as target entities, an averaged sketch entities will be created from them.

This command merges separated sketch entities into a single sketch entity when the results of using the **Auto Sketch** command are messy and complex.



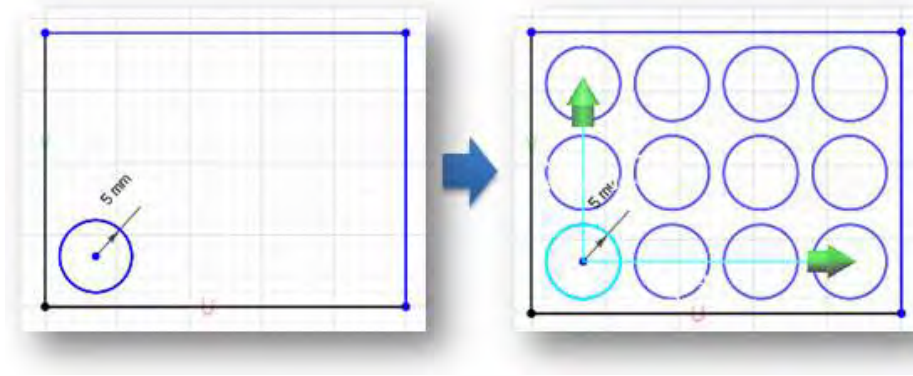
See Also

[Tools > 3D Sketch Tools > Merge](#)

Tools > Sketch Tools > Linear Pattern

What

The **Linear Pattern** command creates linear patterns from sketch entities.



Why

The **Linear Pattern** command is useful for:

- Creating multiple instances of sketch entities with uniform distances along one or two linear paths

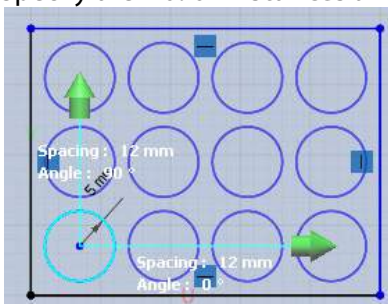
How to Use

How to create linear patterns from sketch entities

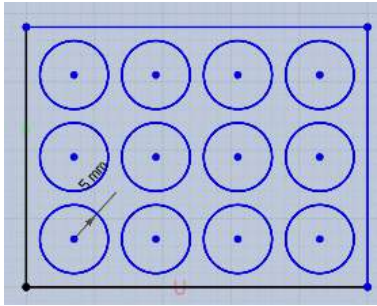
1. Go to **Tools > Sketch Tools > Linear Pattern** or click the icon in the toolbar.
2. Select a sketch entity to be patterned.



3. Specify the **No. of Instances** and the **Spacing** under **Direction 1**.
4. Specify the **No. of Instances** and the **Spacing** under **Direction 2**.



5. Click the **OK** button to complete the command.



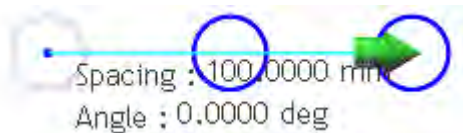
Options in Detail

Select Entities

Entities – Selects sketch entities to be patterned.

Options

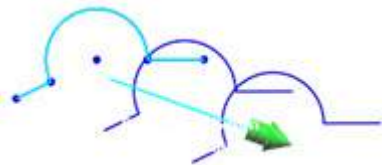
Direction1 – Specifies parameters that can be used for creating patterned instances along **Direction1**.



Note:

The U axis of a base sketch plane is used as Direction 1 by default.

No. of Instances – Specifies the number of instances to be patterned.

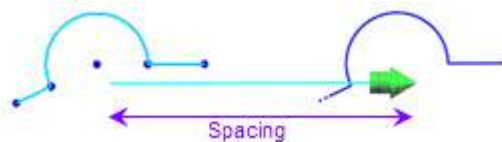


No. of Instances: 3

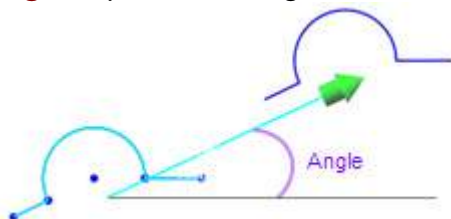


No. of Instances: 5

Spacing – Specifies the distance between instances.




Angle – Specifies the angle between **Direction 1** and the U axis of a base sketch plane.



Note:

The value can be set by manually measuring the angle with the **Measure Angle** button .

To measure the angle between sketch entities, click the **Measure Angle** button  and select two sketch entities.

Direction2 – Specifies parameters that can be used for creating patterned instances along **Direction2**.



Note:

The V axis of a base sketch plane is used as Direction 2 by default.

No. of Instances – Refer to the sub-option in the Direction 1.

Spacing – Refer to the sub-option in the Direction 1.

Angle – Refer to the sub-option in the Direction 1.

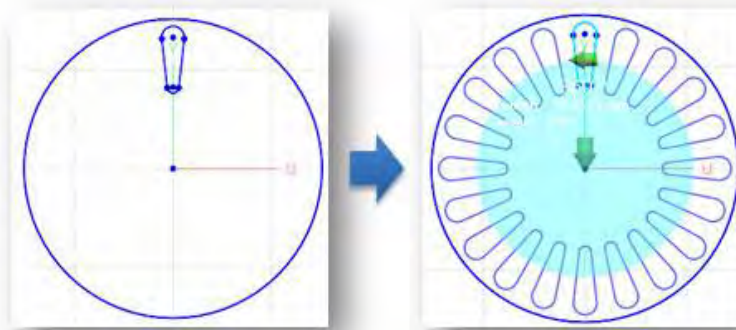
See Also

[Tools > Sketch Tools > Circular Pattern](#)

Tools > Sketch Tools > Circular Pattern

What

The **Circular Pattern** command creates circular patterns from sketch entities.



Why

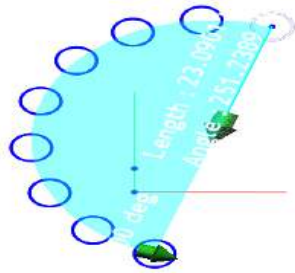
The **Circular Pattern** command is useful for:

- Creating multiple instances of sketch entities with uniform spacing along a circular angle from an anchor point

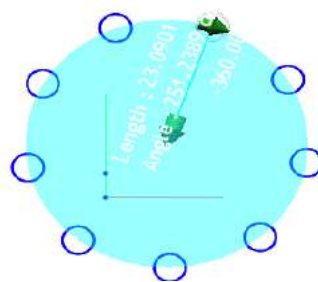
How to Use

How to create circular patterns from sketch entities

1. Go to **Tools > Sketch Tools > Circular Pattern** or click the icon in the toolbar.
2. Select sketch entities to be patterned.
3. Specify the offset distance for **Base X** and **Base Y**. Base X and Base Y determines the origin position of a circular pattern.
4. Specify the **No. of Instances** and the **Total Angle**.

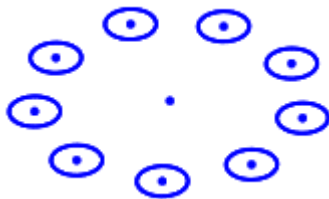


Pattern created with Angle 180



Pattern created with Angle 360

5. Click the **OK** button to complete the command.



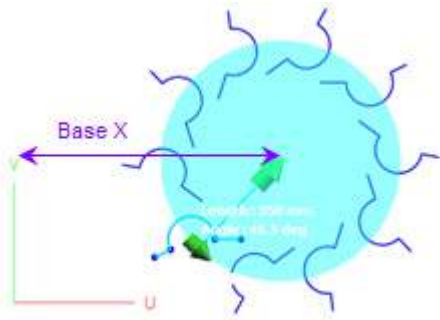
Options in Detail

Select Entities

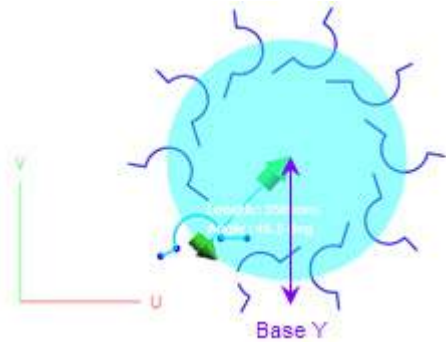
Entities – Selects sketch entities to be patterned.

Options

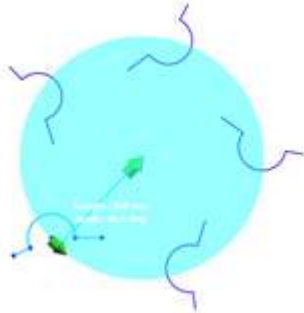
Base X – Specifies the offset distance from the origin position of a base sketch plane along the U axis.



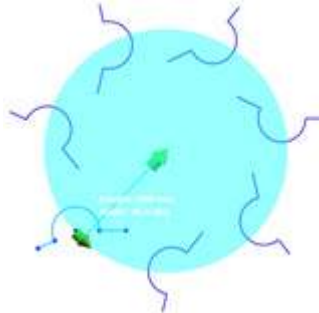
Base Y – Specifies the offset distance from the origin position of a base sketch plane along the V axis.




No. Of Instances – Specifies the number of instances to be patterned.

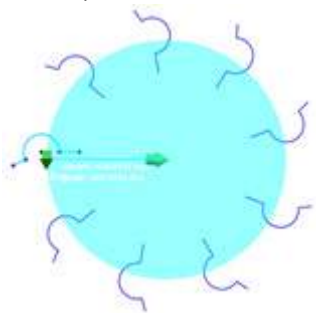


No. of Instances: 5

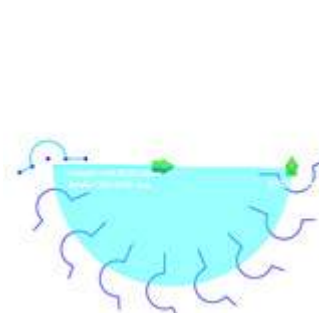


No of Instances: 7

Total Angle – Specifies the total angle in which a pattern should be created. To reverse the angle direction, click the **Reverse** button .




Total Angle: 360



No of Instances: 180

Note:

The value can be set by manually measuring the angle with the **Measure Angle** button .

To measure the angle between sketch entities, click the **Measure Angle** button  and select two

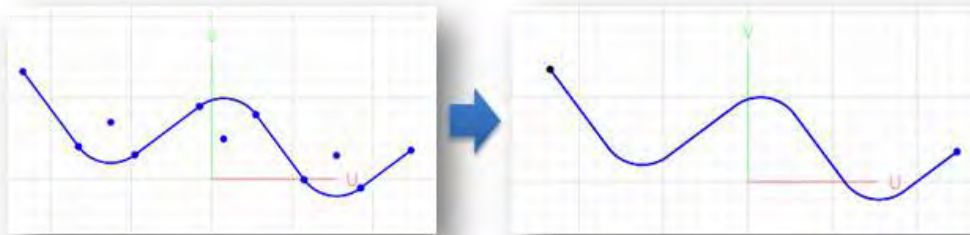
See Also

[Tools > Sketch Tools > Linear Pattern](#)

Tools > Sketch Tools > Fit To Spline

What

The **Fit To Spline** command converts sketch entities into splines.



Why

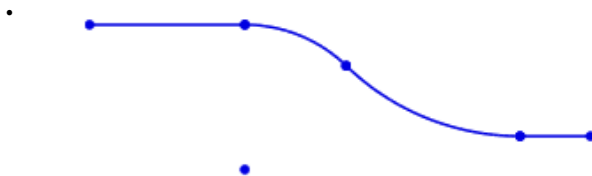
The **Fit To Spline** command is useful for:

- Converting sketch entities into splines
- Removing edges from a modeled feature based on sketch entities

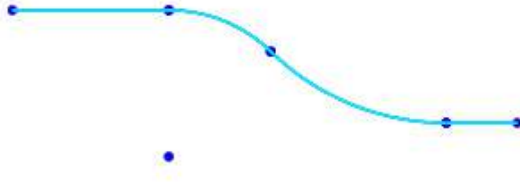
How to Use

How to convert sketch entities into splines

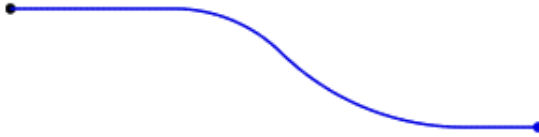
- 1 Go to **Tools > Sketch Tools > Fit To Spline** or click the icon in the toolbar.



- 2 Select sketch entities and check the **Remove Original Entities** option to remove original target sketch entities.

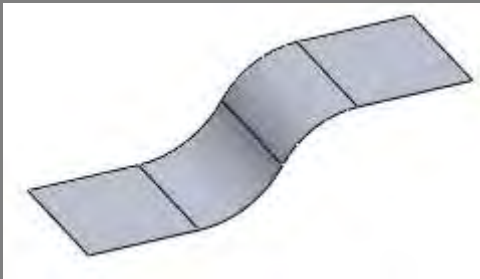


3 Click the **OK** button to complete the command.

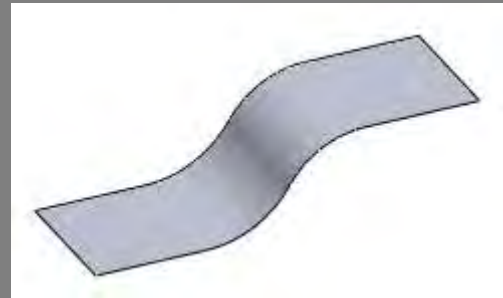


Note:

When an extrude feature is generated by using sketch entities the Fit To Spline command converts the sketch entities into splines removes the edges that occurs when an extruded feature is generated. For more information, see [Insert > Solid > Extrude](#) or [Insert > Surface > Extrude](#).



Extrusion from sketch entities consisted of arcs and lines



Extrusion from spline

Options in Detail

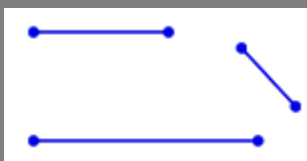
Fit Tolerance – Specifies the maximum deviation that is allowed when an original sketch entities is converted to spline.

Remove Original Entities – Removes original sketch entities.

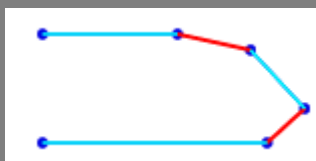
Entities – Selects sketch entities.

Note:

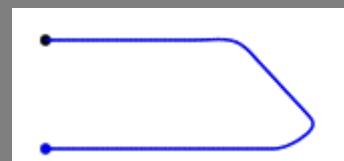
Disjoined entities can also be converted into a spline by adding lines between the original sketch entities.



Original Entities



Selecting Entities



Fit Spline Result

See Also

[Tools > Sketch Tools > Text To Spline](#)

Tools > Sketch Tools > Text To Splines

What

The **Text To Splines** command converts text sketch entities into splines.



Why

The **Text To Splines** command is useful for:

- Converting text sketch entities into splines
- Manually modifying the shape of text

How to Use

How to convert text sketch entities into splines

1. Go to **Tools > Sketch Tools > Text To Splines** or click the icon in the toolbar.



2. Select a text sketch entity.
3. Click the **OK** button to complete the command. Check that the text is converted into splines.



Options in Detail

Texts – Selects texts to convert into splines.

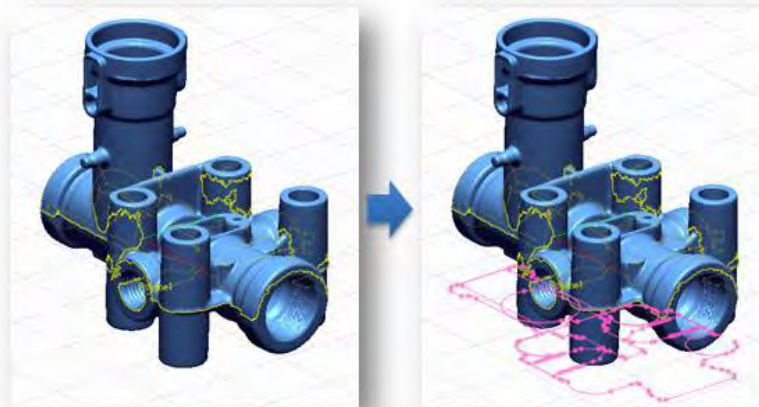
See Also

[Tools > Sketch Tools > Fit To Spline](#)

Tools > Sketch Tools > Insert Polyline

What

The **Insert Polyline** command inserts a Ref.Polyline onto a base sketch plane as a section polyline.



Why

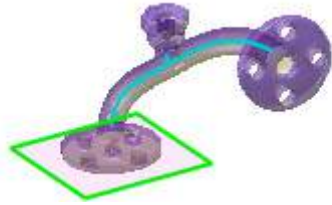
The **Insert Polyline** command is useful for:

- Inserting a Ref.Polyline onto a base sketch plane as a section polyline
- Converting design intent such as a bead line, silhouette, etc into a section polyline and drawing sketch entities based on the converted section polyline

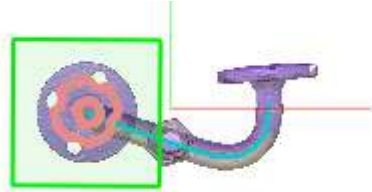
How to Use

How to Insert a Ref.Polyline onto a base sketch plane as a section polyline

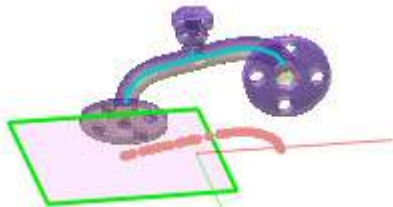
1. Go to **Insert > Mesh Sketch** or click on the icon in the toolbar to enter the **Mesh Sketch** mode and select a base sketch plane. For more information, see [Insert > Mesh Sketch](#).



2. Click the **OK** button.



3. Go to **Tools > Sketch Tools > Insert Polyline** or click the icon in the toolbar. Select a Ref.Polyline.
4. Click the **OK** button to complete the command. Check that the Ref.Polyline is converted into a section polyline.



Options in Detail

Ref.Polyline – Selects a Ref.Polyline to insert onto a current base sketch plane as a section polyline.

Note:

To select a Ref.Polyline as a target entity, the Ref.Polyline needs to be prepared before inserting into a section polyline. Create a Ref.Polyline by using [Insert > Ref.Geometry > Polyline](#) command before entering the Mesh Sketch mode. For more information, see [Insert > Ref.Geometry > Polyline](#).

See Also

[Insert > Ref.Geometry > Polyline](#)

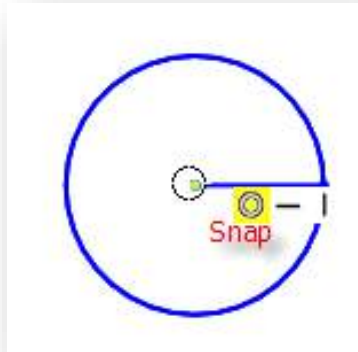
7.5.8. Tools > Sketch Options

The **Sketch Options** menu features various commands used to draw sketch entities on a base sketch plane are provided in **Sketch Options** menu. The options can be toggled on or off while drawing sketch entities.

Tools > Sketch Options > Snap On Entities

What

The **Snap On Entities** option snaps sketch entities being created onto other sketch entities with constraints.



Why

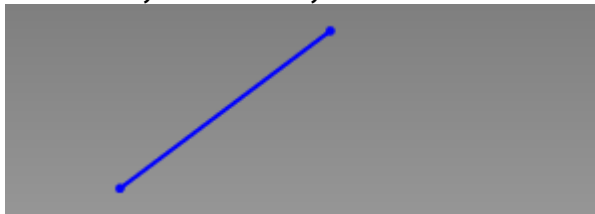
The **Snap On Entities** command is useful for:

- Snapping sketch entities being created onto other sketch entities

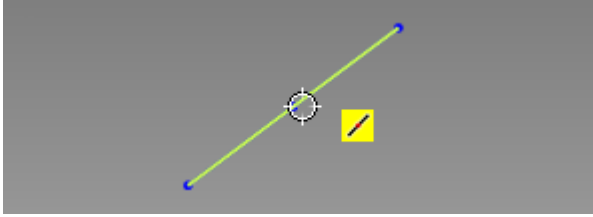
How to Use Snap

How to snap sketch entities to other sketch entities when drawing


1. Go to **Tools > Sketch Options > Snap On Entity** or toggle the icon on in the menu. This functionality is enabled by default.



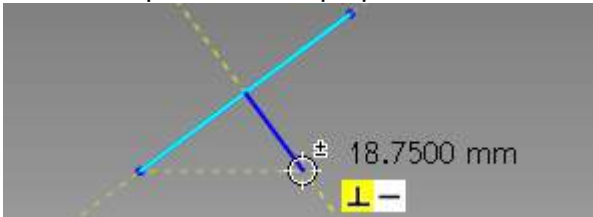
2. Go to **Tools > Sketch Entities > Line** or click the icon in the toolbar. Move the mouse cursor over a line to find the middle position.



Note:

The  icon indicates the middle position of a line.

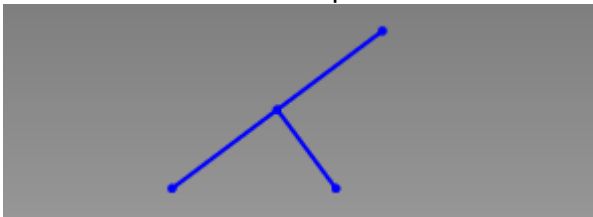
3. Pick an end point where a perpendicular and horizontal constraints icon is shown.



Note:

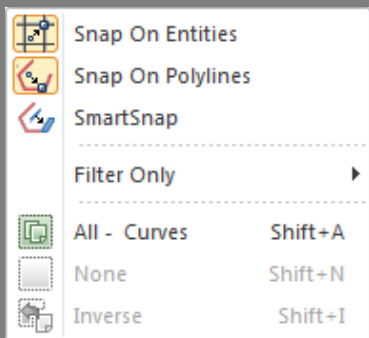
The  icon indicates that a created sketch entity can be set with perpendicular and horizontal constraints.

4. Click the **OK** button to complete the command.



Note:

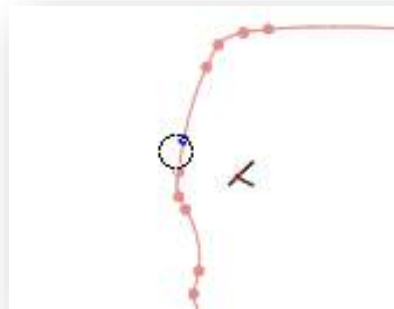
Clicking the right mouse button on a blank area in the 2D Sketch View will directly access the **Snap On Entities** command in the pop-up menu.



Tools > Sketch Options > Snap On Polylines

What

The **Snap On Polylines** option snaps 2D entities being created onto section polylines.



Why

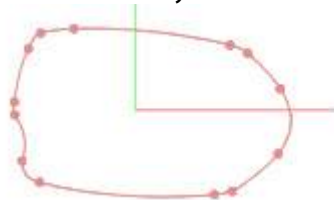
The **Snap On Polylines** command is useful for:

- To snap sketch entities on section polylines.

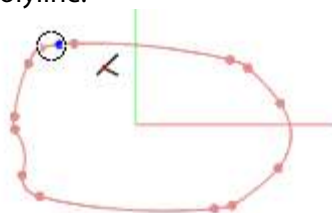
How to Use

How to draw sketch entities snapped to a section polyline

1. Go to **Tools > Sketch Options > Snap On Polylines** or toggle the icon on in the toolbar. This functionality is enabled by default.



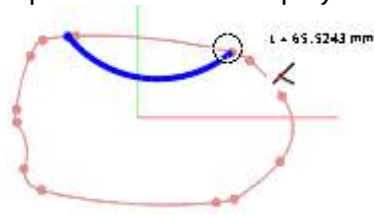
2. Go to **Tools > Sketch Entities > 3 Point Arc** or click the icon in the toolbar to create an arc. Move the mouse cursor until the snap to polyline icon is displayed. Pick a point on the section polyline.



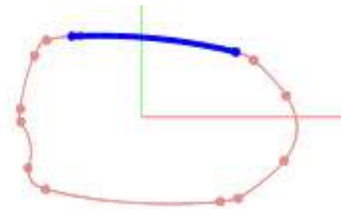
Note:

The  icon indicates that a created sketch entity can be snapped to a polyline.

3. Pick a second point on the section polyline.

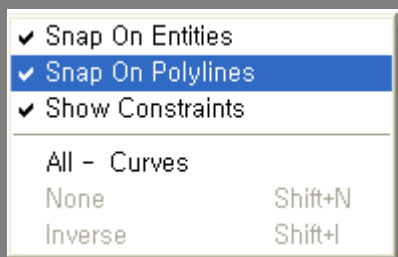


4. Pick a third and final point to complete the arc. Click the **OK** button to finish the command.



Note:

Clicking the right mouse button on a blank area in the 2D Sketch View will directly access the **Snap On Polylines** command in the pop-up menu.



See Also

[Tools > Sketch Options > Smartsnap](#)

Tools > Sketch Options > Smartsnap

What

The **Smartsnap** option snaps 2D entities onto section polylines.



Why

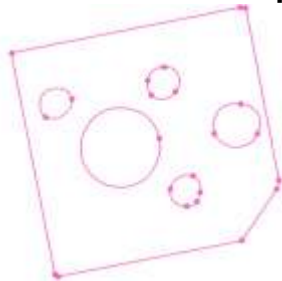
The **Smartsnap** command is useful for:

- Roughly and quickly draw 2D entities onto section polylines.

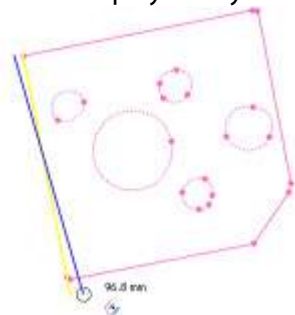
How to Use

How to draw sketch entities using Smartsnap

1. Go to **Tools > Sketch Options > Smartsnap** or toggle the icon on in the toolbar.



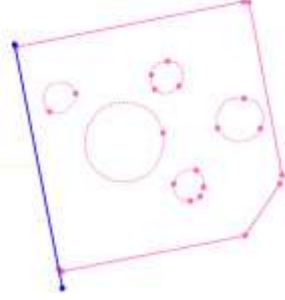
2. Go to **Tools > Sketch Entities > Line** and pick a starting point. Fitting target polylines will be displayed in yellow during the creation of a line.



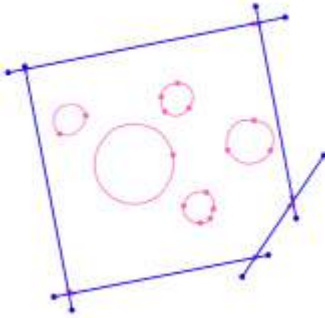
Note:

The Smartsnap function can also be used by pressing the **Shift** key while drawing sketches without toggling the Smartsnap sketch option.

3. Pick the ending point to complete a line. The line will be attached to the polyline.



4. Using the same method, create the remaining lines.



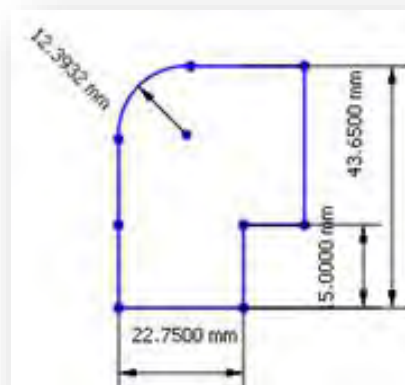
See Also

[Tools > Sketch Options > Snap On Polylines](#)

7.5.9. Tools > Smart Dimension

What

The **Smart Dimension** command sets dimensions such as **Distance**, **Angle**, and **Radius** on sketch entities.



Why

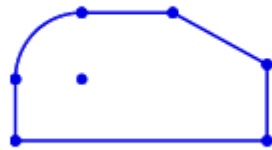
The **Smart Dimension** command is useful for:

- Setting dimensions on sketch entities

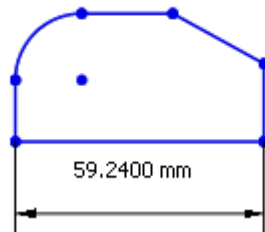
How to Use

How to set dimensions on sketch entities

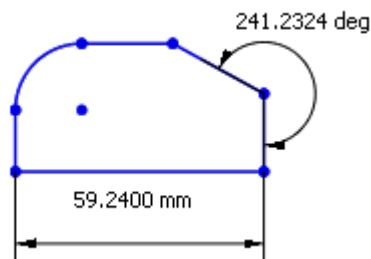
1. Go to **Tools > Smart Dimension** or click the icon in the toolbar.



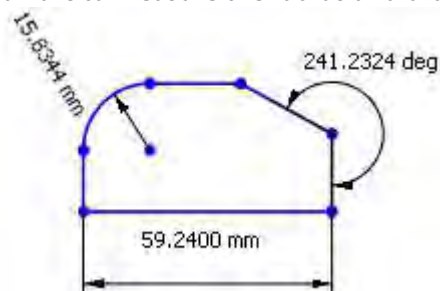
2. Click a line of a sketch entity to create a distance measurement for it and drag the measurement to a desired location.



3. Click two lines on the side to measure the angle between them and drag the measurement to a desired location.

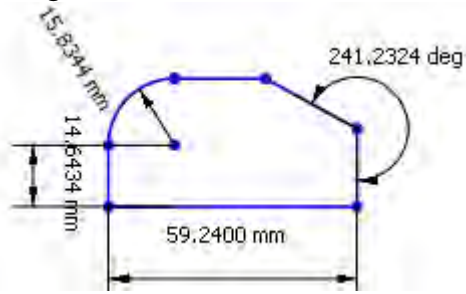


4. Click an arc to measure the radius and drag the measurement to a desired location.

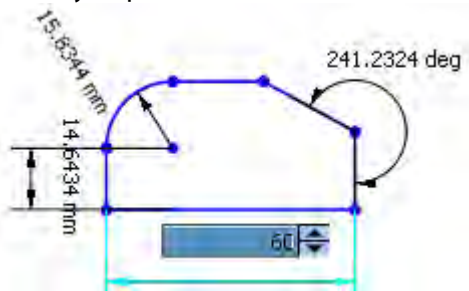


5. Click the center of an arc and a bottom line to measure the distance between them

and drag the measurement to a desired location.



6. Double-click a dimension and directly edit the value to change the value of sketch entities by a specific amount.

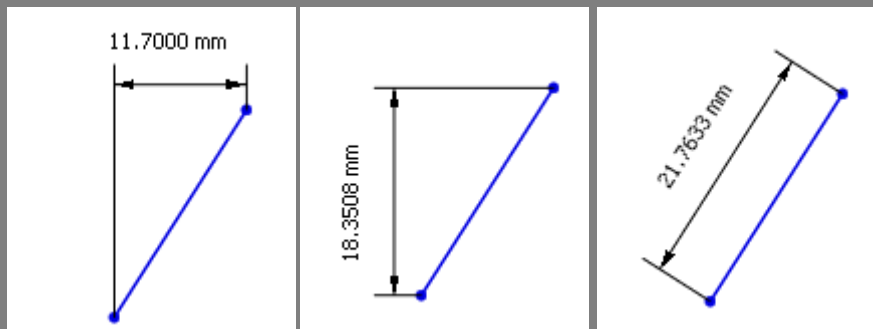


Options in Detail

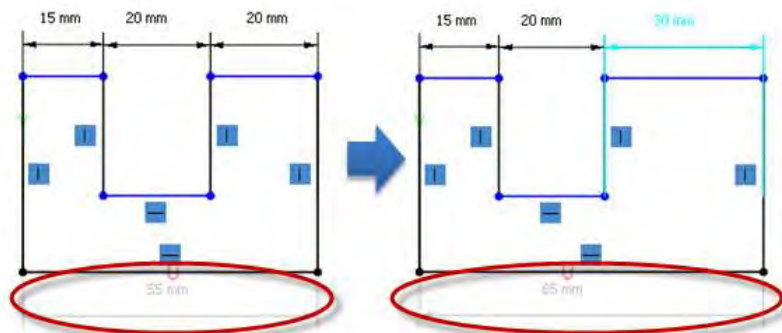
Value – Specifies the value of a dimension.

Note:

A distance can be measured with different results according to the location of the annotation.



Driven – Sets the dimension as a driven dimension so that it can be dynamically changed depending on changes of other dimensions. When dimensions are defined around the sketch and become as over-constrained, this option is useful to make dimension as flexible changed.



Note:

This option can be set when entering editing mode by double-clicking the specified dimension.

7.5.10. Tools > 3D Sketch Entities

The **3D Sketch Entities** menu features various commands used to draw various 3D sketch entities such as **Line**, **Pencil**, **Section**, and **Boundary**.

The **3D Sketch Entities** menu provides 3 sets of sketch edit tools used to:

Create 3D Sketch Entities

- **Spline**
Creates 3D curves on mesh or in free 3D space
- **Pencil**
Manually draw splines on 3D mesh
- **Section**
Creates section curves on mesh and solid surface bodies
- **Intersection**
Creates intersection splines between bodies
- **Boundary**
Extracts curves from the boundary of a mesh

Create 3D Sketch Entities Using Other Objects

- **Surface UV Curves**
Creates 3D curves on a face along its U and V directions
- **Fit From Points**
Creates 3D curves from points or region boundaries by fitting to them

Create Curves From Mesh Features

- **Trace Feature Line**
Automatically extracts character lines from mesh
- **Extract Feature Curves**

Detects and extracts 3D lines from areas of high curvature

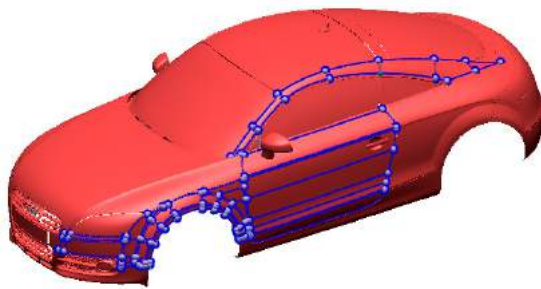
- **Create Curve Network**

Generates a 3D curve network on a mesh

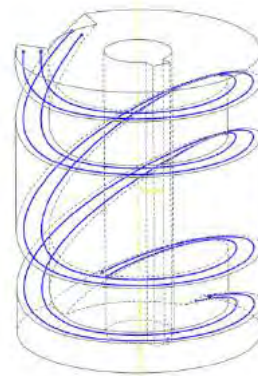
Tools > 3D Sketch Entities > Spline

What

The **Spline** command is used to create a 3D curve on mesh or in free 3D space. This command is available in both **3D Sketch mode** and **3D Mesh Sketch mode**.



Example of Splines on mesh



Example of a spline in 3D space

Why

The **Spline** command is useful for:

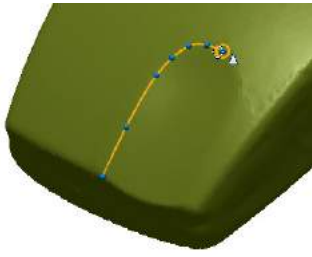
- Creating a curve network for a boundary fitted surface
- Creating a path for a swept or lofted body
- Creating an original user-designed shape from scratch instead of copying the shape of an existing object
- Working with highly curved 3D features such as a corkscrew feature
- Working with freeform shapes that require more manual control than an automated tool can provide
- Splitting a section made out of lines and splines to create new enclosed sections

How to Use

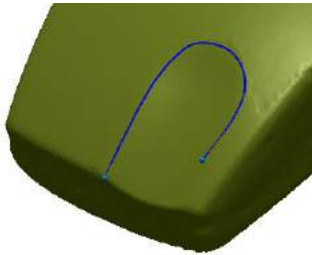
A. How to create a Spline and alter it in 3D Mesh Sketch Mode

1. Go to **Tools > 3D Sketch Entities > Spline**, or click the icon in **3D Mesh Sketch mode**.
2. Select a point on a mesh to place a spline. Create extra nodes to further define the shape

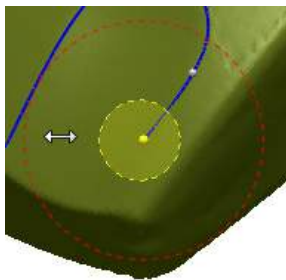
by adding a node with a left mouse click.



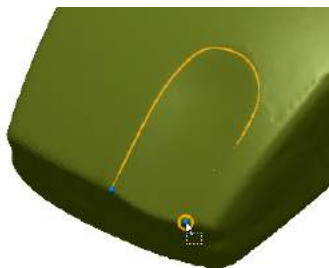
3. To finish a spline you can either right-click, double left-click, or press the **Enter** button on the keyboard. Click **OK** when the spline has been completed.



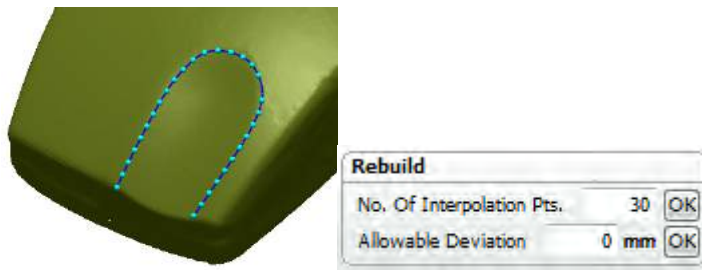
4. When the mouse cursor is hovered over a node the selection area will be shown in a red circle. Hold down the **Alt** key on the keyboard and move the mouse from left to right to change the size of the selection area.



5. If further altering of the spline is desired, select a node by left-clicking on it and dragging it to a desired location then release the mouse button.

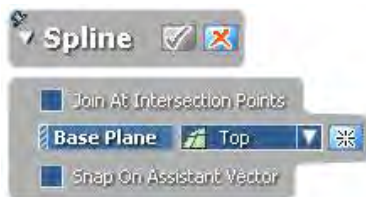


6. When more or less interpolation points are desired for the spline use the **Rebuild** command found in the toolbar. In this example the number of interpolation points has been set to 30.

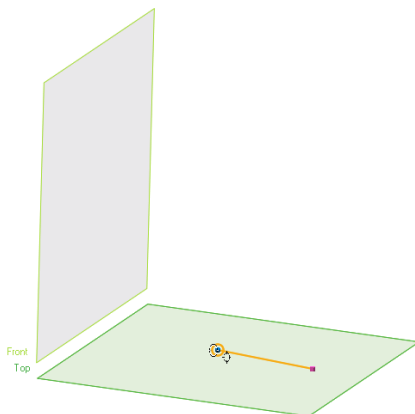


B. How to create a Spline in 3D space in 3D Sketch Mode

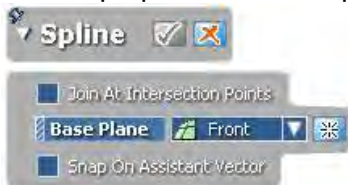
1. Go to **Tools > 3D Sketch Entities > Spline**, or click icon in the **3D Sketch mode**.
2. Select a base plane. For this example use the **Top** base plane.



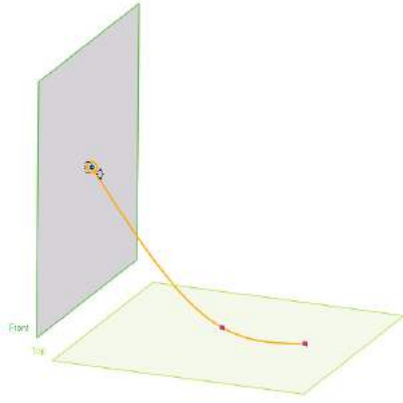
3. Left-click to begin drawing a spline. The first click will create the first node of the spline.



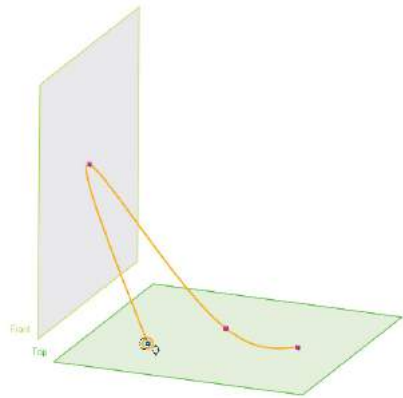
4. Without clicking anywhere else in the model view, click the **Base Plane** option again and select a perpendicular base plane, in this case, the **Front** base plane.



5. Click again to place another node of the spline on the perpendicular plane. Doing this allows the spline to first be aligned to the Top plane and then aligned to the Front plane mid-spline.



6. Finish the spline by switching back to the Top base plane and double-left click, right click, or press the **Enter** key on the keyboard to finish the spline. The spline will curve based on where the nodes were placed. Click **OK** to finish the command.



Note:

When a spline is drawn on a mesh in **3D Sketch mode** the snap function will automatically be enabled. If snapping to mesh is not desired, use the **Alt** key while creating the spline.

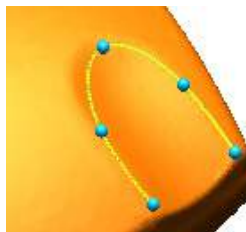
Options in Detail

Base Plane – Select base planes.

Generate Intermediate Points – Automatically creates intermediate points between picked points to reduce deviation.

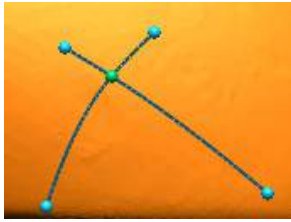


Enable Option

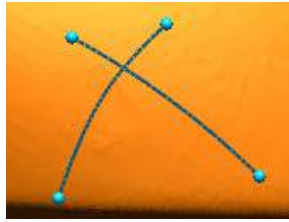


Disable Option

Join At Intersection Points – Automatically creates intersection points between intersecting curves. The Tolerance range can be set to determine if curves intersect or not.



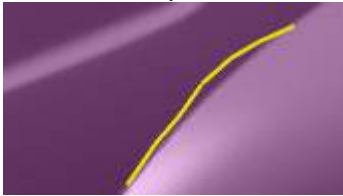
Enable Option



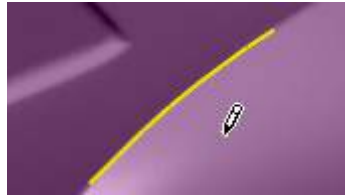
Disable Option

Tolerance – Sets the tolerance range.

Refit To Feature Lines – Searches for feature lines around picked points to use to create a spline and refits the spline to those features.



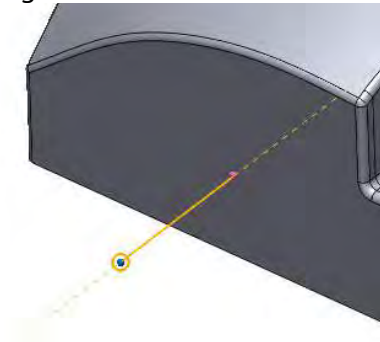
Enable Option



Disable Option

Snap On Curvature Flow – Snaps a spline to a feature curvature while drawing the spline.

Snap On Assistance Vector – Enable to snap a point of curve on an assistance vector that is automatically extracted from target body entity when drawing a curve in the Model View. Dot-lined assistance vectors will be displayed when hovering the mouse cursor on body faces or edges.



Note:

The **Snap On Assistance Vector** option is useful for creating a curve that has a specific directional property, such as perpendicular, parallel, and tangent, from a target entity.

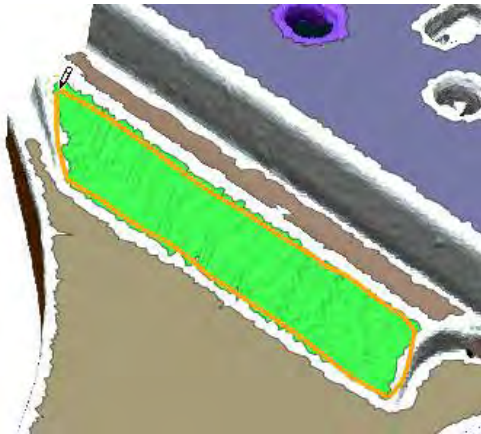
See Also

[Tools > 3D Sketch Entities > Pencil](#)

Tools > 3D Sketch Entities > Pencil

What

The **Pencil** command is used to manually draw splines on 3D mesh. It can only be used in **3D Mesh Sketch mode**.



Why

The **Pencil** command is useful for:

- Sketching on models that have irregular features or areas that require precise manual control

How to Use

How to draw a line on mesh

1. Click **Tools > 3D Sketch Entities > Pencil** in the **3D Mesh Sketch mode**.



2. Draw lines using the pencil.



3. Click **OK** to complete the command.



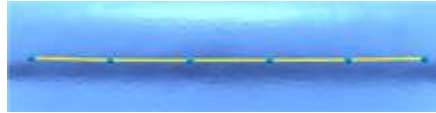
Options in Detail

Smoothness – Defines the smoothness of a curve. The option slider can be set between Min and Max.

Interpolation Pts. Sampling Distance – Specifies the distance between node points of a spline.



Sampling Distance: 1mm

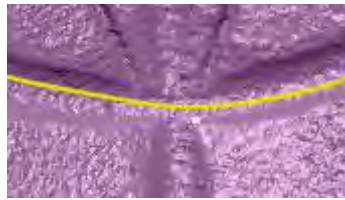


Sampling Distance: 5mm

Merge Strokes – Merges drawn strokes.



Sampling Distance: 1mm



Sampling Distance: 5mm

Refit To Feature Lines – Tries to find and fit feature lines when drawing with the pencil tool.



Sampling Distance: 1mm



Sampling Distance: 5mm

Search Radius – Changes the radius distance for searching for feature lines around a drawn stroke.

See Also

[Tools > 3D Sketch Entities > Spline](#)

Tools > 3D Sketch Entities > Section

What

The **Section** command is used to create section curves on mesh and solid surface bodies. This command is available in both **3D Sketch mode** and **3D Mesh Sketch Mode**.



Why

The **Section** command is useful for:

- Creating a curve network for a boundary fitted surface
- Creating a path for a swept or lofted body
- Making a profile of a model to simplify in making a lofted body

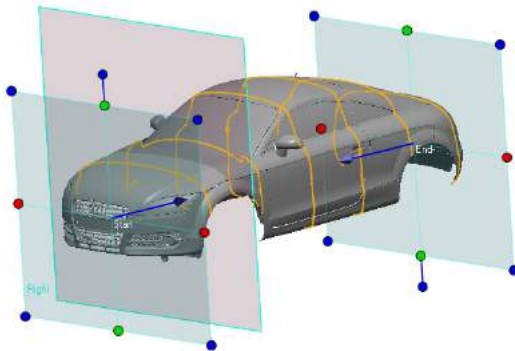
How to Use

How to create a model profile in 3D Sketch mode

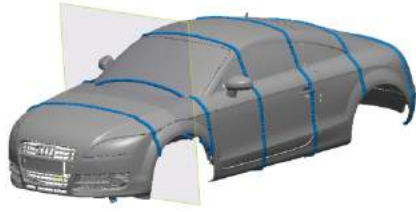
1. Enter **3D Sketch mode** and go to **Tools > 3D Sketch Entities > Section**, or click on the icon in the toolbar.



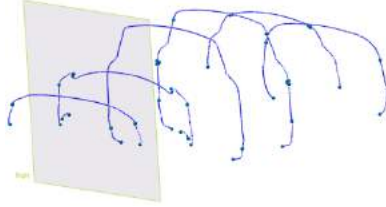
2. Click on the mesh to select the entire model as the **Target Entity**. This will select all entities available. Click the **Next Stage** button to proceed.
3. Go to the **N-Division Between Planes** method and select the **Right plane** as the reference plane for the **Base Plane** input box. Set the **No. Of Sections** option to 7 and click **OK**.



4. The model will be sectioned into seven sections of equal length.



5. Exit the **3D Sketch mode** and hide the mesh to see the created sections.



Options in Detail

Methods

The following methods are available:

- **Draw Line On Screen**
Creates sections from a line.
 - **Select Plane**
Creates sections from an intersecting plane.
 - **N-Division Between Planes**
Creates sections based on a chosen base plane.
 - **N-Division Along Curve**
Creates sections based on a chosen curve.
 - **Radial**
Creates sections based radial segments.
 - **Cylindrical**
Creates sections based on cylindrical segments.
 - **Conical**
Creates sections based on conical segments.
-

Draw Line On Screen – Creates sectional curves by drawing lines on the screen.



Cutting Line



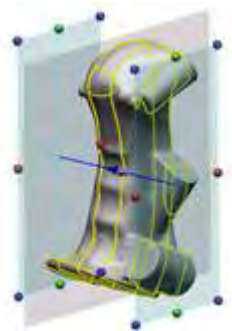
Resulting Section Spline

Note:

The **Shift** key can be used to draw horizontal or vertical lines. Multiple cutting lines can be created at

one time by using the **Draw Line On Screen** method. The **Undo** command or **Ctrl + Z** keyboard shortcut will delete the last created cutting line.

N-Division Between Planes – Creates a specified number of sections based on chosen base planes.



N-Division



Resulting Section Spline

Distance Between Sections – Sets the distance between sections.

No. of Sections – Sets the total number of sections.

Equal Spacing – Creates an equal amount of space between all sections.

No. of Sections – Sets the total number of sections to be created which includes the two base planes selected for the option.

Show Section Base Geometry – Shows the base geometry of the section.

N-Division Along Curve – Creates the specified number of sections with respect to the curve.

Uniform – Creates multiple sections by either a total number of sections or by equal distances.

No. of Sections – Uses a set number of sections as the criteria.

Equal Spacing – Uses equal spacing as the criteria.

Show Section Base Geometry – Shows the base geometry of the section.

Curvature – Creates more sections in areas of higher curvature.

No. of Sections – Uses a set number of sections as the criteria.

Show Section Base Geometry – Shows the base geometry of the section.

Pick Points – Creates sections manually by picking points on a curve. To cancel the last picked point the **Undo** keyboard shortcut **Ctrl + Z** can be used.

Show Section Base Geometry – Shows the base geometry of the section.

Radial – Creates virtual radial sections and generates section curves from them.

Axis – Chooses an axis.

Base Plane – Chooses a base plane.

Flip Start Plane – Flips the starting plane the opposite direction.

No. of Sections – Sets the number of sections.

Total Angle – Sets the total angle value of a radial section.

Equal Spacing – Sets equal spacing between sections.

Show Section Base Geometry – Shows the base geometry of a section.

Cylindrical – Creates cylindrical sections and generates section curves.

Axis – Chooses an axis.

Base Plane – Chooses a base plane.

Flip Start Plane – Flips the starting plane the opposite direction.

No. of Sections – Sets a number of sections.

Total Angle – Sets a total angle value.

Radius Interval – Sets the distance between sections.

Base Radius – Sets the base radius of a starting section.

Height – Sets the height of a cylindrical area.

Section Inside – Sets the creation direction from the base radius.

Show Section Base Geometry – Shows the base geometry of the section.

Conical – Creates conical sections and generates section curves from them.

Axis – Chooses an axis.

Base Plane – Chooses a base plane.

Base Position – Specifies X, Y, or Z as a base position.

Flip Start Plane – Flips the starting plane the opposite direction.

No. of Sections – Sets the number of sections.

Total Angle – Sets a total angle value.

Radius Interval – Sets a radius interval.

Bottom Radius – Sets a value for a bottom radius.

Top Radius – Sets a value for a top radius.

Apex Angle – Sets a value for a slope.

Height – Sets a value for height.

Section Inside – Sets the value for a creation direction.

Show Section Base Geometry – Shows section cylinders.

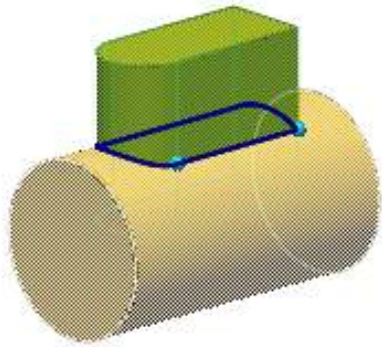
See Also

[Tools > 3D Sketch Entities > Intersection](#)

Tools > 3D Sketch Entities > Intersection

What

The **Intersection** command is used to create intersection splines between bodies. It can only be used in **3D Sketch mode**.



Why

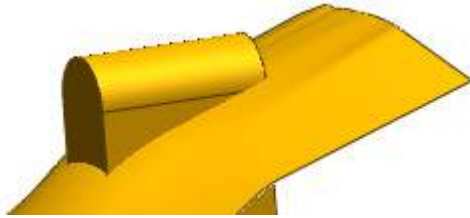
The **Intersection** command is useful for:

- Finding the boundaries where two entities meet

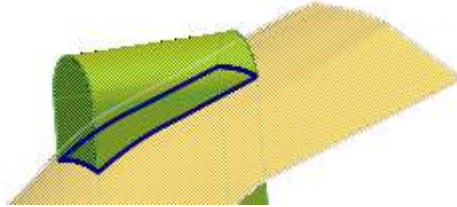
How to Use

How to find an intersection for two entities

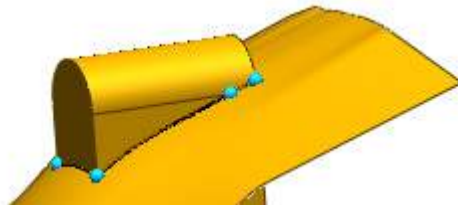
1. Go to **Tools > 3D Sketch Entities > Intersection**, or click on the icon in **3D Sketch mode**.



2. Select target **Tool Bodies** and **Target Bodies**. The **Tool Bodies** will be the bodies that are used as the base bodies and the **Target Bodies** will be the bodies that the intersections using the **Tool Bodies** are found on.



3. Click **OK** to complete the command. The intersection between the bodies will be created.



Options in Detail

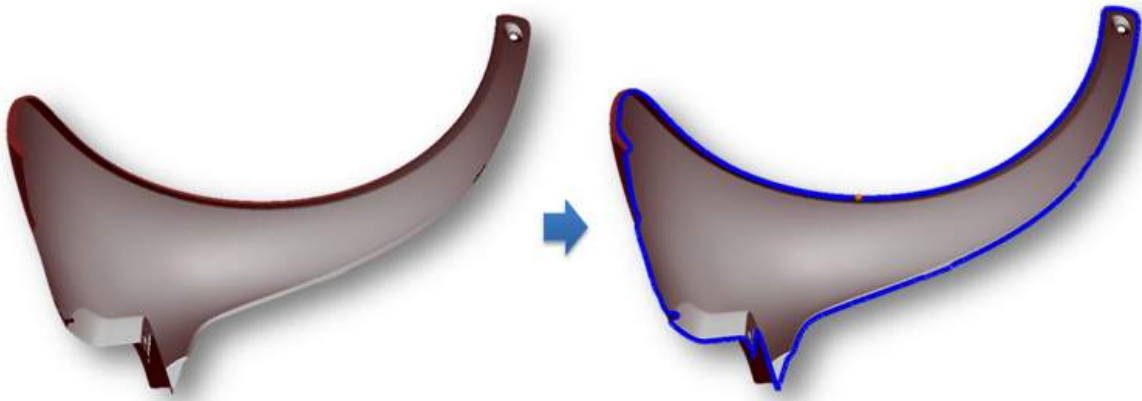
Tool Bodies – Select tool bodies.

Target Bodies – Select target bodies.

Tools > 3D Sketch Entities > Boundary

What

The **Boundary** command is used to extract curves from the boundary of a mesh. This command is available in both **3D Sketch mode** and **3D Mesh Sketch mode**.



Why

The **Boundary** command is useful for:

- Creating a path for a swept or lofted body
- Finding the boundary of a model that is irregularly shaped

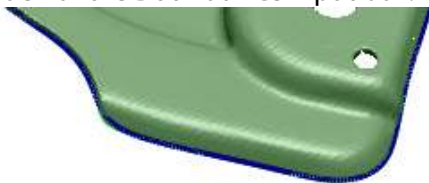
How to Use

How to find boundary for an entity

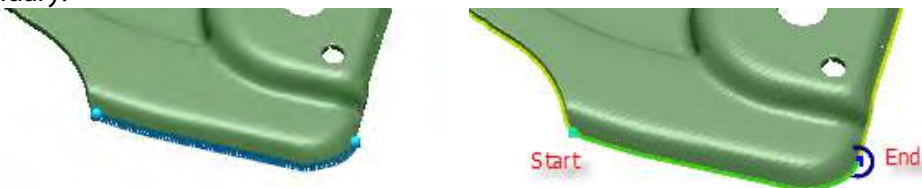
1. Go to **Tools > 3D Sketch Entities > Boundary**, or click on the icon in **3D Mesh Sketch** or **3D Sketch** mode.



2. Select the **Curves From Boundaries** option and choose an outside boundary on the model for the **Boundaries** input box.



3. Click **OK** to extract the selected boundary as a spline. If only a part of the boundary is desired to be extracted then the **Interactively Split Curve Segments From a Single Boundary** option can be used to manually pick the starting and ending points of the boundary.



Options in Detail

Curves From Boundaries – Creates 3D spline curves on selected mesh boundaries.

Interactively Split Curve Segments From a Single Boundary – Creates a partial boundary curve by picking a starting and ending point on a boundary.

Boundaries – Selects boundaries on a mesh.

Smoothness – Defines the smoothness of a boundary curve by using a value between a Min and Max limits.

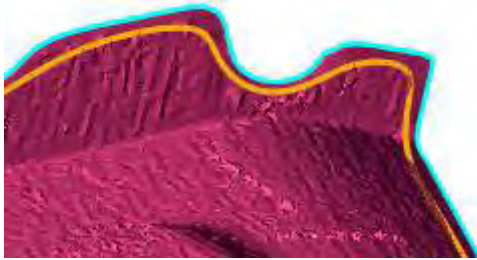


Min. Smoothness



Max. Smoothness

Offset – Creates an offset curve from a boundary.



See Also

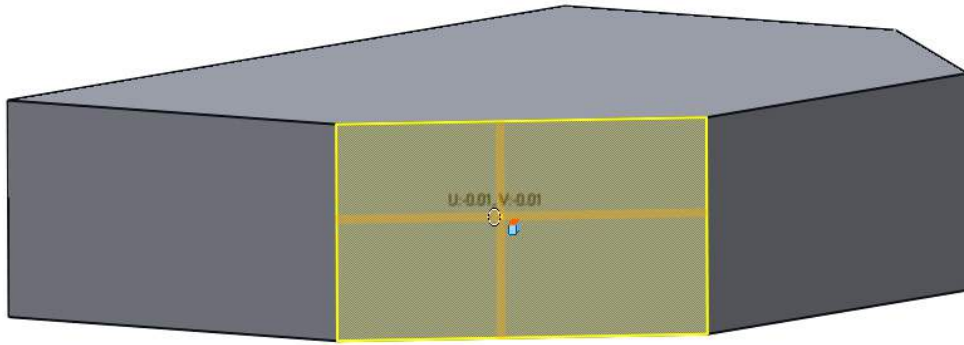
[Tools > 3D Sketch Entities > Fit From Points](#)

[Tools > 3D Sketch Entities > Create Curve Network](#)

Tools > 3D Sketch Entities > Surface UV Curves

What

The **Surface UV Curves** command is used to create 3D curves on a face along its U and V directions. This command is available only in **3D Sketch mode**.



Why

The **Surface UV Curves** command is useful for:

- Creating 3D curve networks on curved faces that are relatively uniform and follow the U and V directions

How to Use

How to find set a UV Curves

1. Go to **Tools > 3D Sketch Entities > Surface UV Curve** or click on the icon in the **3D Sketch Mode**.



2. Select a position on a face.



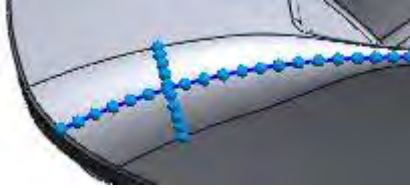
3. Click **OK** to finish the command.



Options in Detail

Parameter Direction – Chooses the U, V, or both the U and V directions for creating curves.

Make Curve Editable – Created curves will have node points that are editable.



Make Node At UV Curves Intersection – Creates intersection nodes at the intersection point of the U and V curves.



See Also

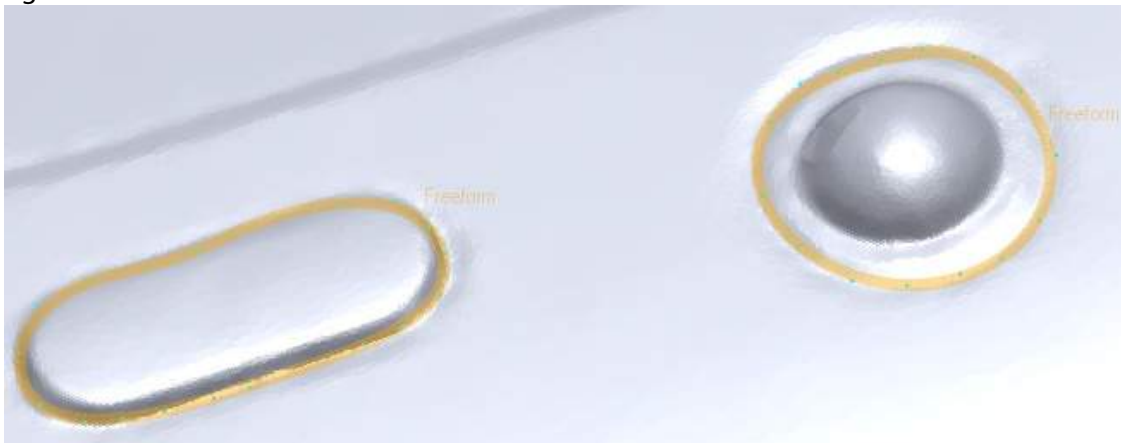
[Tools > 3D Sketch Entities > Spline](#)

[Tools > 3D Sketch Entities > Section](#)

Tools > 3D Sketch Entities > Fit From Points

What

The **Fit From Points** command is used to create 3D curves from points or region boundaries by fitting to them. This command is available in both **3D Sketch mode** and **3D Mesh Sketch mode**.



Why

The **Fit From Points** command is useful for:

- Generating analytic curves such as circles, slots and rectangles from point data

- Generating freeform curves from point data
- Generation of curves on region boundaries and character lines of a model

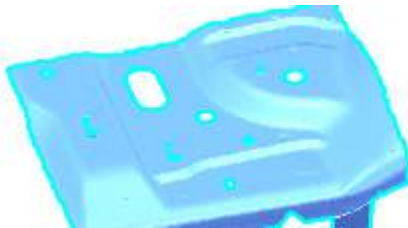
How to Use

How to extract feature shapes using the Fit From Points command

1. Enter **Mesh mode** to extract boundary vertices from a mesh.



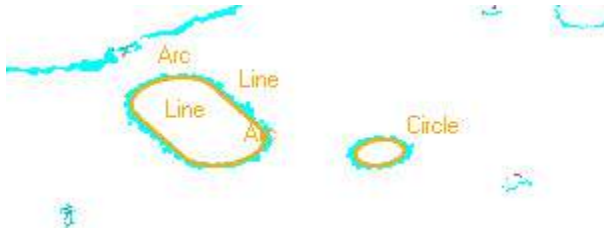
2. Go to **Select > Boundary Entities** and change the **Type To Select** option to **Poly-Vertices** and select all boundaries. Click **OK**.



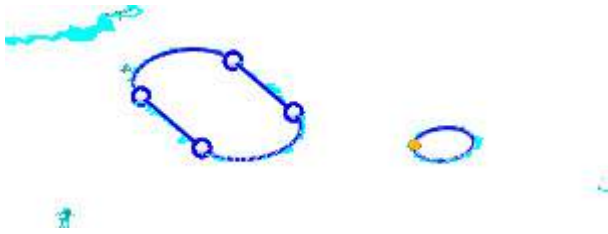
3. Go to **Edit > Copy** to copy selected boundary vertices. Exit the **Mesh mode**.
4. Go to **Edit > Paste**. A new mesh will be created.



5. Select the new mesh and enter the **3D Mesh Sketch mode**.
6. Go to **Tools > 3D Sketch Entities > Fit From Points** and select points.

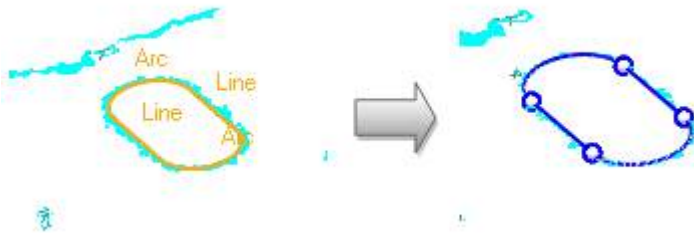


7. Click **OK** to complete the command. Curves will be extracted from the slot and circle shapes.

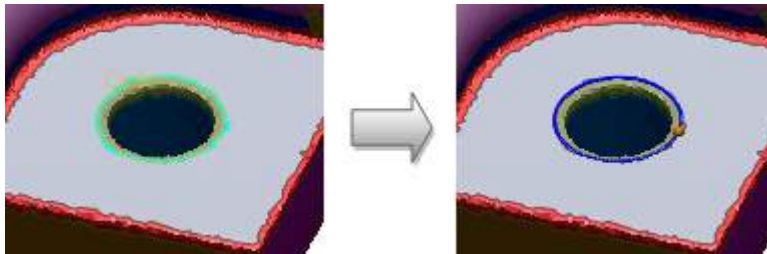


Options in Detail

Target – Select poly-vertices or region boundaries for fitting curves.



Fitting curve from points



Fitting curve from region boundaries

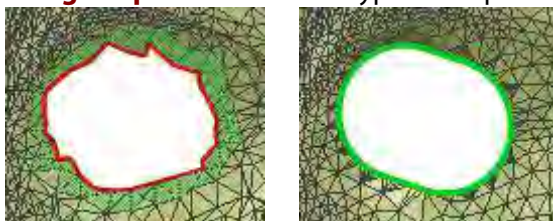
Split Target – Splits a fitting curve. Click the button and pick the starting and ending points for the splitting position on the preview curve and a split point will be added.



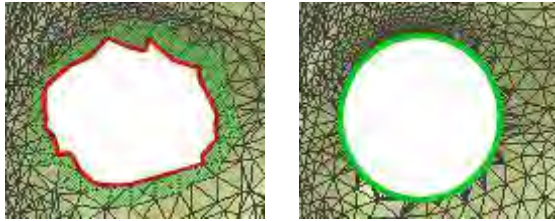
Fitting Options

Fitting Shape – Generates a fitting curves from several shapes: Auto, Circle, Slot, Rectangle Polygon, Line and Freeform.

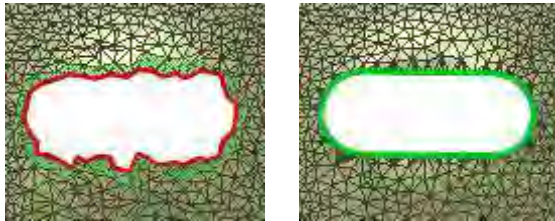
Fitting Shape – Chooses the type of shape for the fitting curve.



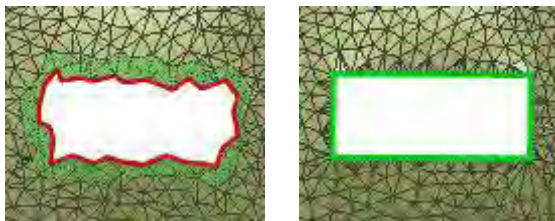
Auto



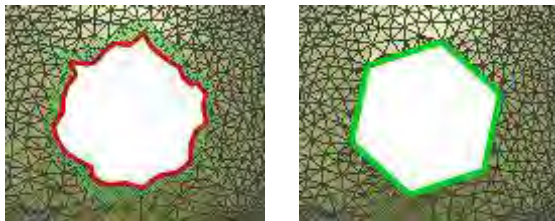
Circle



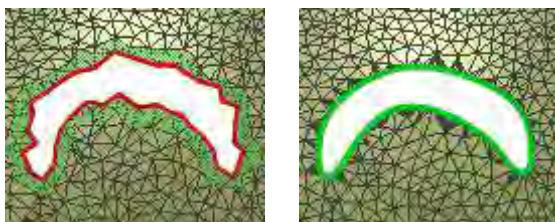
Slot




Rectangle



Polygon

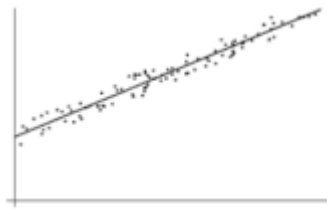


Freeform

Allowable Tolerance – Fits a curve to a shape within a given value. The **Auto Estimate** button  computes the value automatically.

More Options

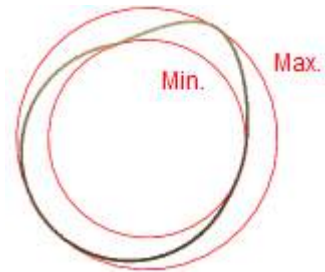
Fitting Type – Creates a best-fitted value by the least square fitting method when the **Best Fit** option is selected. The least squares method is the approximate solution from distributed data. The **Min. Bound** option finds the smallest radial distance value between a pair of concentric circles and **Max. Bound** finds the largest value.



Least Square Method



Best Fit



Min / Max Boundary
from Minimum Separation

Remove Outlier – Removes outlier data from the fitting type using the **N*Sigma, Ratio (%)**, or **Absolute Distance** algorithms.

Sampling Ratio – Samples data points by a specified value. When the ratio is 100%, all selected data will be used. When the ratio is 50%, half of the selected data will be used.

Use Only Effective Data – Uses only effective data when fitting. Effective data is the actual data points used in fitting after sampling and outlier removal.

Rounded Corner – Rounds corners of rectangular shapes.

No. Of Edges – Sets the number of edges created.

Smoothness – Controls the smoothness of extracted curves.

See Also

[Tools > 3D Sketch Entities > Boundary](#)

[Tools > 3D Sketch Entities > Create Curve Network](#)

Tools > 3D Sketch Entities > Trace Feature Line

What

The **Trace Feature Line** command is used to automatically extract character lines from mesh. This command is only available in 3D Mesh Sketch mode.



Why

The **Trace Feature Line** command is useful for:

- Automatically extracting a line from a model that might be difficult to do manually

How to Use

How to extract a feature line from mesh

1. Go to **Tools > 3D Sketch Entities > Trace Feature Line**, or click the icon in **3D Mesh Sketch mode**.



2. Select a poly-vertex to extract a feature line.



3. A preview curve will be generated.



4. Clicking **OK** will extract the feature curve as a spline. If only a part of the feature line is desired to be extracted then the starting and ending points can be selected.



5. Click **OK** or double click on the curve to complete the command.



Options in Detail

Seed Poly-Vertex – Picks a poly-vertex on the mesh to extract a feature line.

Search Depth – Adjusts the depth of the feature line. A value between 1 and 5 can be inputted or selected with a slider bar. The larger the depth, the more character lines can be found, but a larger depth may include noisy character lines.

Smoothness – Specifies the smoothness of a feature line. A value between Min and Max. can be specified. When a high smoothness value is chosen, the resulting curve will be smoother but may lose accuracy.



Min. Smoothness



Max. Smoothness

See Also

[Tools > 3D Sketch Entities > Extract Feature Curves](#)

[Tools > 3D Sketch Entities > Create Curve Network](#)

Tools > 3D Sketch Entities > Extract Feature Curves

What

The **Extract Feature Curve** command is used to detect and extract 3D lines from areas of high curvature. This command is only available in **3D Mesh Sketch mode**.



Why

The **Extract Feature Curve** command is useful for:

- Extracting feature lines from areas that have high degrees of variable curvature

Note:

It is more efficient to use a mesh that has had the **Decimate** command applied to it rather than a dense mesh for the **Extract Feature Curve** tool.

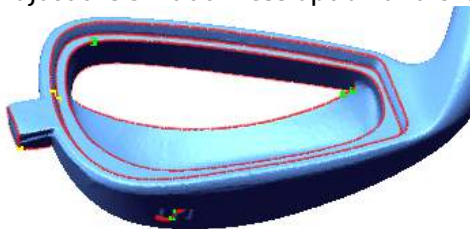
How to Use

How to extract a feature curve from mesh

1. Go to **Tools > 3D Sketch Entities > Extract Feature Curves**, or click the icon in **3D Mesh Sketch** mode.



2. Adjust the **Detection Sensitivity** and **Loop Size Level** options and click the **Next Stage** button.
3. Adjust the **Smoothness** option and click **OK** to finish the command.



Options in Detail

Stages

This command consists of the following stages:

- **Set Properties**

Sets the Detection Sensitivity and Loop Size Level options.

- **Preview**

Sets the Smoothness and previews the feature curve.

Stage 1/2 – Set Properties

Set Properties – Sets the Detection Sensitivity and Loop Size Level options for the command.

Detection Sensitivity – Determines the sensitivity used to extract feature curves. For low curvature models, set the Sensitivity to high and for high curvature model set the Sensitivity to low.

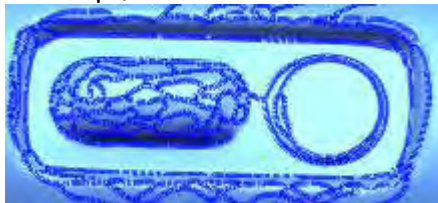


0 Detection Sensitivity



10 Detection Sensitivity

Loop Size Level – Removes noisy curves. If the application unexpectedly creates many small loops, increase the value of the Loop Size Level option.



0 Loop Size Level



10 Loop Size Level

Stage 2/2 – Preview

Smoothness – Specifies the smoothness of a feature line. A value between Min and Max. can be specified. If a high smoothness value is chosen, the resulting curve will be smoother but may lose accuracy.



Min. Smoothness



Max. Smoothness

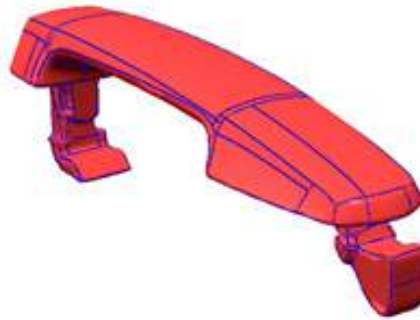
See Also

[Tools > 3D Sketch Entities > Fit From Points](#)

Tools > 3D Sketch Entities > Create Curve Network

What

The **Create Curve Network** command is used to generate a 3D curve network on a mesh. This command is only available in **3D Mesh Sketch mode**.



Why

The **Create Curve Network** command is useful for:

- Quickly creating a 3D curve network that follows character features to enclose a mesh
- Generating a 3D curve network to use with the Boundary Fit Surface command

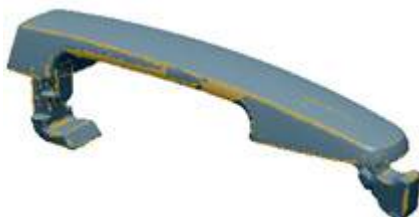
How to Use

How to create a curve network from mesh

1. Go to **Tools > 3D Sketch Entities > Create Curve Network**, or click the icon in the toolbar in **3D Mesh Sketch mode**.



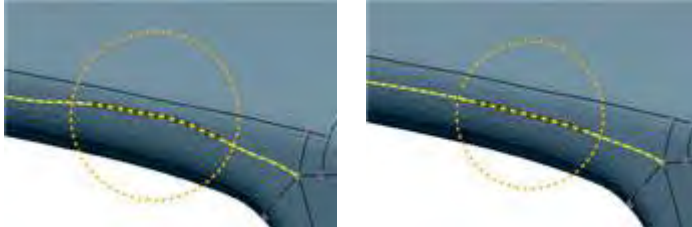
2. Adjust the slider of the **Level of Patch Complexity** option to **Less**.



3. Click the **Next Stage** button. A preview of the curve network will be displayed.



4. The **Deform** command can be used to adjust a curve if it needs to be smoothed.

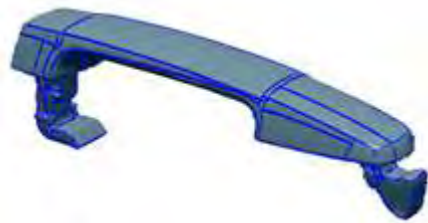


Note:

The editing zone can be resized by using the **Alt** key while dragging the mouse left or right.



5. Click **OK** to complete the command.



Options in Detail

Stages

This command consists of the following stages:

- **Feature Detection**
Used to detect features for creating a curve network
 - **Curve Network Editing**
Edit a created curve network
-

Stage 1/2 – Feature Detection

Level of Patch Complexity – Adjusts the level of patch complexity in a mesh using the slider. The number of curve loops will be increased by adjusting towards "More".

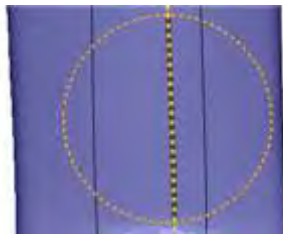
Optimize Patch Network – Optimizes the patch shape for creating a curve network.

Stage 2/2 – Curve Network Editing

Deform(D) – Deforms a curve by selecting and dragging nodes. The shortcut key of this method is **D**.

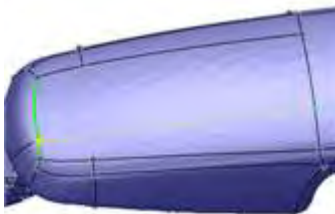


Before Deform

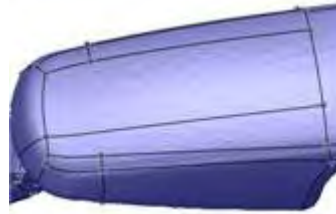


After Deform

Split(S) – Splits a curve loop to generate more loops by picking a point on a curve and picking a second point on another curve displayed in green. The shortcut key of this method is **S**.

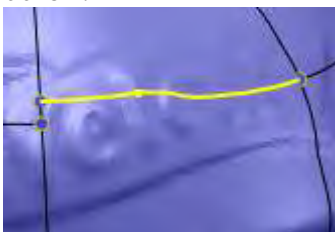


Before Split

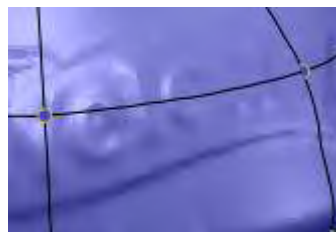


After Split

Slide(E) – Slides a starting or ending point of a curve to another curve. The shortcut key of this method is **E**.

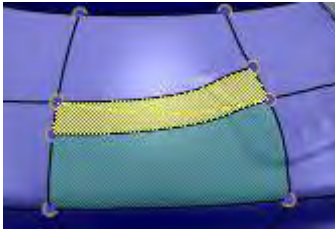


Before Detach And Move

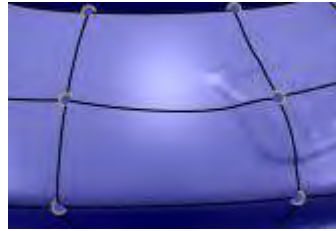


After Detach And Move

Merge(G) – Merges curve loops in the network with another green-colored loops highlighted as being able to be merged. The shortcut key of this method is **G**.

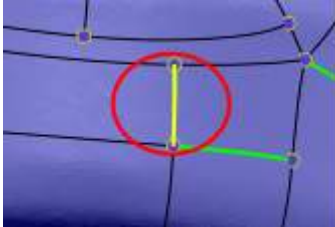


Before Merge

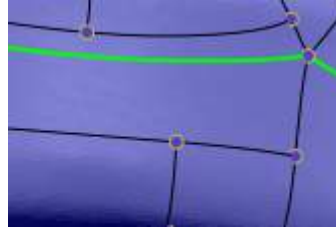


After Merge

Remove(R) – Removes a curve. The shortcut key of this method is **R**.



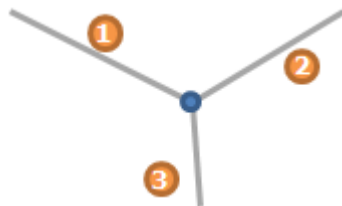
Before Remove



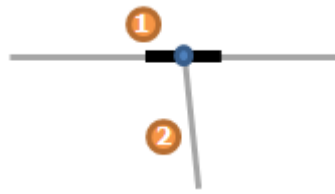
After Remove

Swap Y and T Junction (W) – Swaps the properties of a Y junction and a T junction. Splines able to be swap will be displayed in green. When a green spline is selected, the T junction mark will be swap to the Y junction. The shortcut key of this method is **W**.

Y junction consists of three curves and T junction consists of two curves.

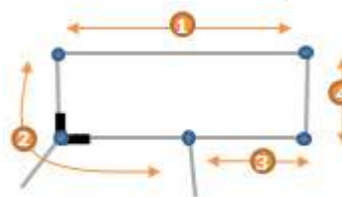


Y Junction

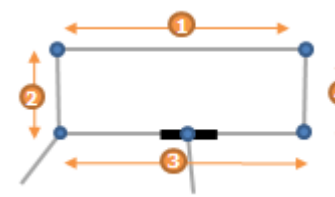


T Junction



If a curve loop is consist of Y and T junction but T junction is located at a corner of a rectangle, the rectangle does not consist of right four edges. By swapping Y and T junction this rectangle can have correct four edges.



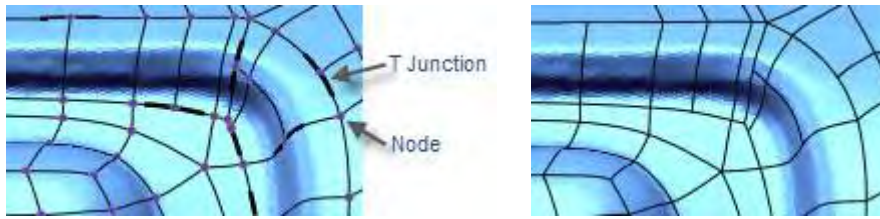
Before Swap



After Swap

Show Erroneous Network – Highlights erroneous curves (gaps between curves or twisted curves expected to be generated error surface patches) in red. The thin red curve shows errors created when fitting or sewing. The thick red curve shows errors created when generating surfaces. Erroneous curves can be navigated through by clicking the **Previous**  or **Next**  buttons.

Show Nodes And T-junctions – Displays interesting nodes and T junction on curve loops.



Show Nodes and T Junctions

Hide Nodes and T Junctions

Deform With Stiffening Around Crossing Nodes – Maintains tangency after deforming curves around an intersecting node, otherwise, sharp edges will be formed.



With Option

Without Option

See Also

[Tools > 3D Sketch Entities > Boundary](#)

[Tools > 3D Sketch Entities > Fit From Points](#)

[Tools > 3D Sketch Entities > Trace Feature Line](#)

[Tools > 3D Sketch Entities > Extract Feature Curves](#)

7.5.11. Tools > 3D Sketch Tools

The **3D Sketch Tools** menu features various commands used to edit 3D sketch entities such as **Offset**, **Project**, **Smooth**, and **Trim**.

The **3D Sketch Tools** menu provides 3 sets of sketch edit tools used to:

Modify 3D Sketch Entities

- **Offset**
Modifies sketch entities by adding a tangent arc in between the sketch entities
- **Project**
Modifies sketch entities by adding an angled line in between the sketch entities
- **Mirror**
Offsets sketch entities by a specified distance
- **Convert Entities**
Converts other entities into sketch entities

Edit 3D Sketch Entities

- **Smooth**
Trims sketch entities
- **Trim**
Extends sketch entities
- **Extend**
Extends sketch entities
- **Split**
Splits a sketch entity into multiple parts
- **Match**
Mirrors a part of a sketch entity
- **Merge**
Merges sketch entities into a single sketch entity

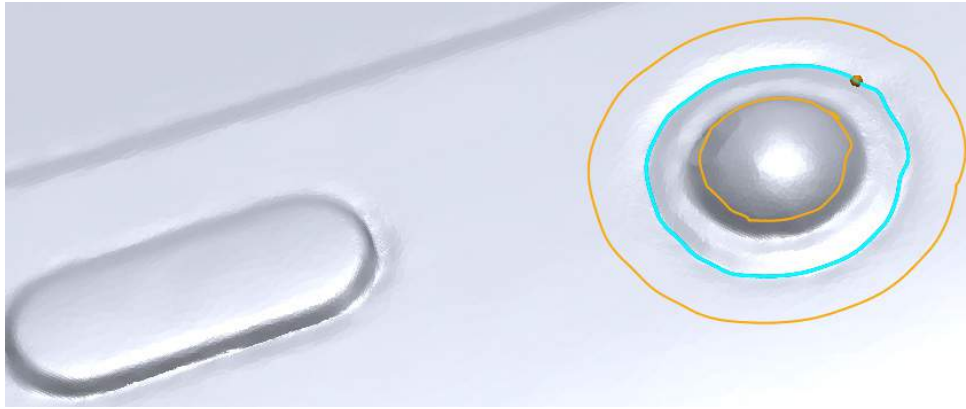
Adding Patterned 3D Sketch Entities

- **Insert Node**
Creates linear patterns from sketch entities
- **Remove Node**
Creates circular patterns from sketch entities
- **Remove Constraint**
Creates circular patterns from sketch entities

Tools > 3D Sketch Tools > Offset

What

The **Offset** command is used to offset existing curves or lines.



Why

The **Offset** command is useful for:

- Offsetting a line or a curve to create lines or curves with the same properties and shape.

How to Use

How to offset a curve on a mesh

1. Go to **Tools > 3D Sketch Tools > Offset**, or click the icon in the toolbar.
2. Select a curve to offset.
3. Specify the distance desired for the offset curve. **Direction1** creates a curve in a direction from the existing curve. **Direction2** creates a curve in the other direction. **Both Directions** creates both curves.



Offset curve in Direction1



Offset curve in Both Directions

Options in Detail

Base Entity – Selects base entities.

Curves – Selects curves to offset.

Distance – Sets the distance of the offset.

Direction1 – Creates a curve in a first direction from the original curve.

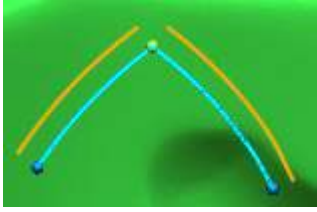
Direction2 – Creates a curve in a second direction from the original curve.

Both Directions – Creates two curves in both directions from the original curve.

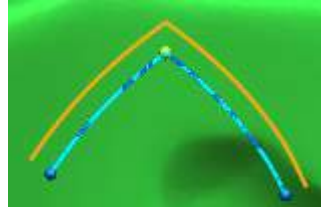
More Options

Remove Original Curves – Removes the original curve after the offset is completed.

Offset Curve Deviation – Connects curves within a given deviation value.



Tolerance: 0.01



Tolerance: 0.1

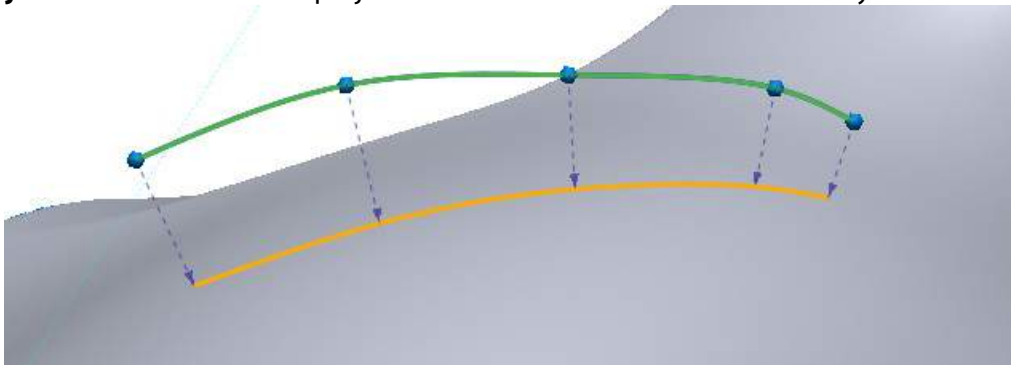
See Also

[Tools > Sketch Tools > Offset](#)

Tools > 3D Sketch Tools > Project

What

The **Project** command is used to project curves or nodes onto a selected entity.



Why

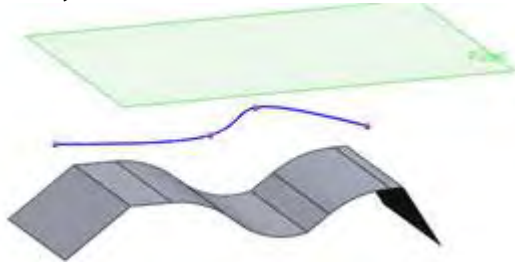
The **Project** command is useful for:

- Creating a copy of a line onto an entity

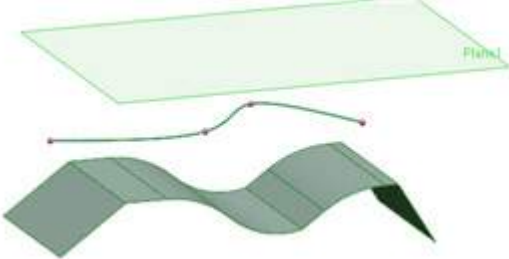
How to Use

How to project a curve onto a surface

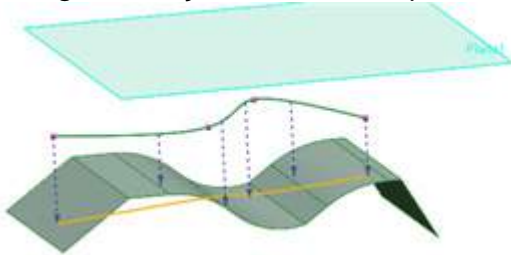
1. Go to **Tools > 3D Sketch Tools > Project** or click the icon in the toolbar. This command can only be used in **3D Sketch Mode**.



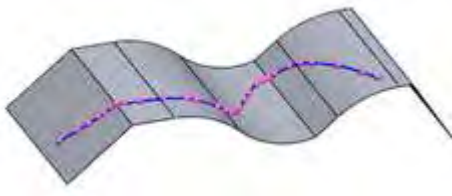
2. Select a target curve and a body as the target entity.



3. Change the **Projection Method** option to **Along Vector** and select a reference plane.



4. Click **OK** to complete the command.



Options in Detail

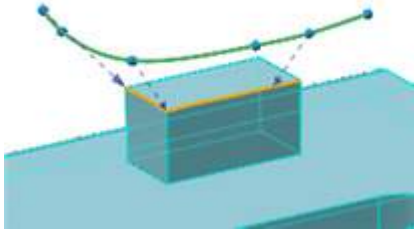
Curves / Nodes – Selects curves or nodes to project.

Target Entity – Selects target entity to project the target curve onto.

Project Duplicated Curve – Retains an original curve.

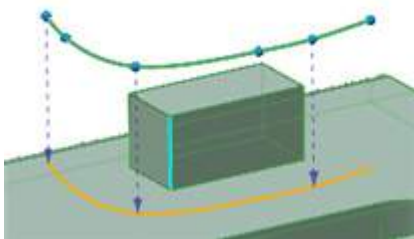
Projection Method

Min. Distance – Projects a curve to a minimum distance.



Along Vector – Projects a curve in a selected direction. A ref. plane, vector, curve or edge can be used.

Tangent – Selects an entity to use.

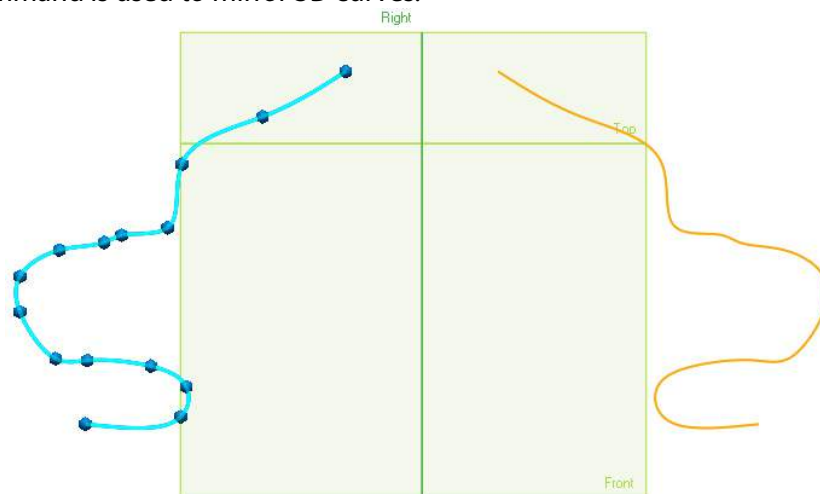


Allowable Deviation – Sets a deviation value between a projected curve and a target entity.

Tools > 3D Sketch Tools > Mirror

What

The **Mirror** command is used to mirror 3D curves.



Why

The **Mirror** command is useful for:

- Creating a mirrored copy of a 3D curve

Note:

A curve mirrored in **3D Mesh Sketch mode** will be projected onto the mesh.

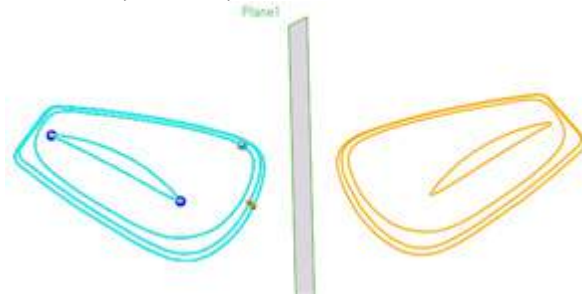
How to Use

How to mirror a curve

1. Go to **Tools > 3D Sketch Tools > Mirror**, or click the icon in the toolbar.



2. Select a **Symmetry Plane** and **Curves** as the entities that will be mirrored.



3. Click **OK** to complete the command.

Options in Detail

Symmetry Plane – Selects a ref. plane or planar face as a Symmetry Plane.

Curves – Select curves to mirror.

Remove Original Curves – Removes original curves after mirroring.

See Also

[Tools > Sketch Tools > Mirror](#)

Tools > 3D Sketch Tools > Convert Entities

What

The **Convert Entities** command is used to convert entities such as sketches, edges, and polylines into 3D curves.



Why

The **Convert Entities** command is useful for:

- Creating a 3D curve out of a 2D entity

Note:

The **Convert Entities** tool available in **3D Mesh Sketch mode** has a few differences from the **Convert Entities** tool found in the 3D Sketch Tools menu. The differences are:

- There are no relationships between the target entity and converted curves in this mode
- A converted curve will be projected onto a mesh and registered as a Mesh 3D curve

How to Use

How to convert an entity

1. Go to **Tools > 3D Sketch Tools > Convert Entities**, or click the icon in the toolbar.



2. Select edges to convert.

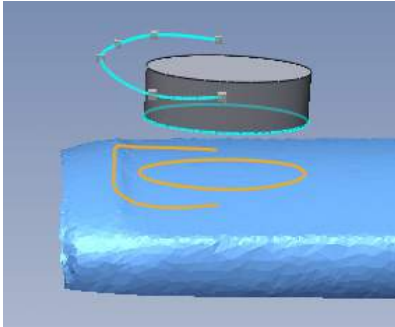


3. Click **OK** to complete the command. A 3D curve will be generated.



Options in Detail

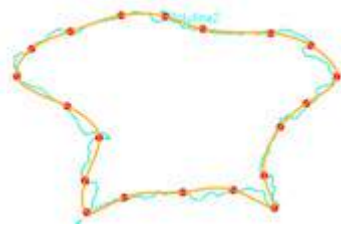
Entities – Selects entities to convert. Entities can be polylines, curves on a 2D Sketch or edges. Curves converted from polylines in the **Sketch mode** have constraints so that if changes are made, the converted curves will also change.



Note:

A spline that is black contains constraints. To remove constraints use the **Remove Constraints** command.

Fitting Accuracy – Sets the percentage of fitting accuracy when converting polylines.



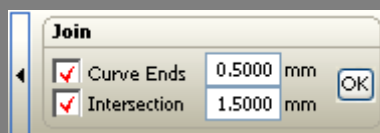
Accuracy: Loose



Accuracy: Tight

Note:

The **Smooth** command regenerates intersection points between curves. The parameters for intersection points can be set in the **Join** tool palette.



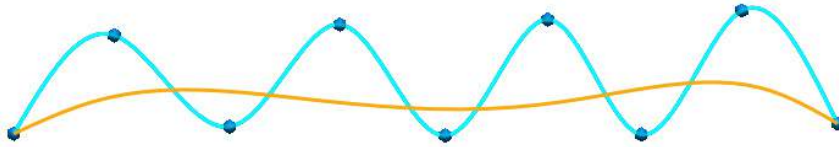
See Also

[Tools > Sketch Tools > Convert Entities](#)

Tools > 3D Sketch Tools > Smooth

What

The **Smooth** command is used to smooth curves.



Why

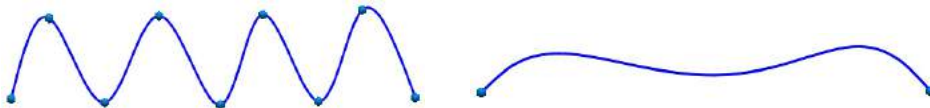
The **Smooth** command is useful for:

- Smoothing irregular curves
- Smoothing a portion of a curve
- Saving time over smoothing curves manually

How to Use

How to smooth a curve

1. Go to **Tools > 3D Sketch Tools > Smooth** or click the icon in the toolbar.
2. Select a curve and adjust the **Smoothness** setting. Click **OK** to complete the command.



Before Smoothing

After Smoothing

Options in Detail

Methods

All Range – Applies smoothing to a selected curve.

Partial Range – Determines the strength of the smoothing procedure.

Sub Options

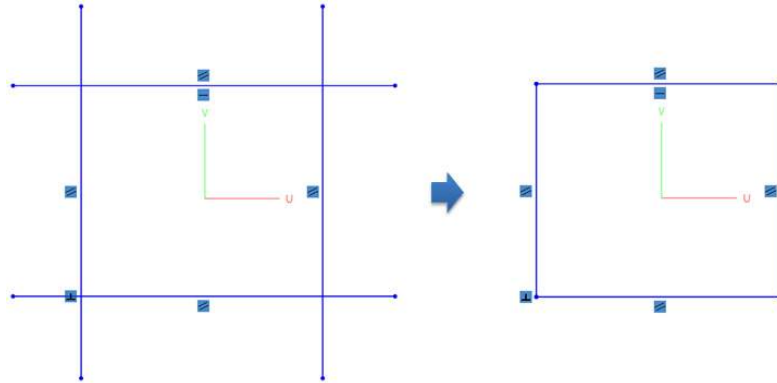
Curves – Selects curves to smooth.

Smoothness – Sets a level of smoothness from **Min.** to **Max.**

Tools > 3D Sketch Tools > Trim

What

The **Trim** command is used to trim curves.



Why

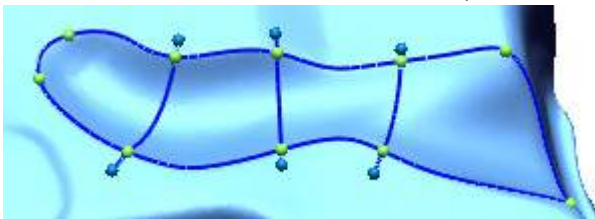
The **Trim** command is useful for:

- Trimming away unwanted sections of a curve
- Combining curves

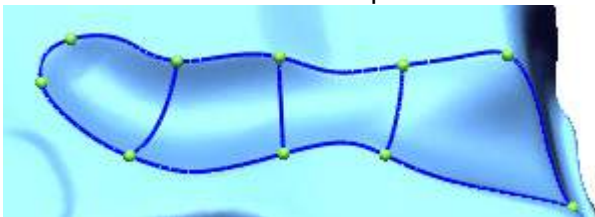
How to Use

How to trim curves

1. Go to **Tools > 3D Sketch Tools > Trim**, or click the icon in the toolbar.



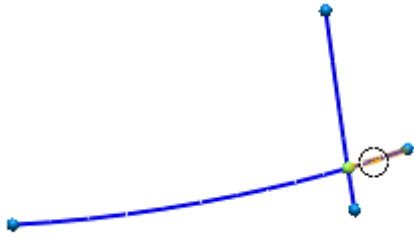
2. Select curves. Click **OK** to complete the command.



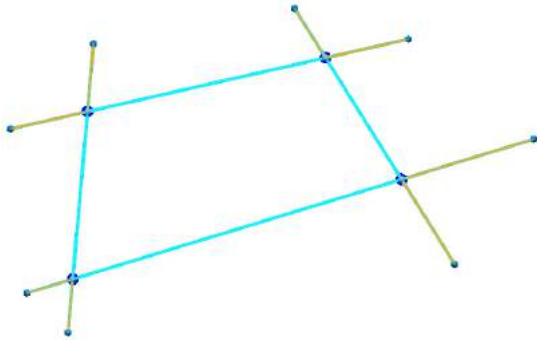
Options in Detail

Methods

Pick Curve Portion To Trim – Trims selected curves.



Trim Short Portion Of Curves – Trims the short portions from selected curves.



Curves – Selects curves to trim.

Portion Rate – Sets the ratio of short curves compared to an original curve length.

Intersection Tolerance – Determines whether curves intersect or not by setting the tolerance for gaps between curves.

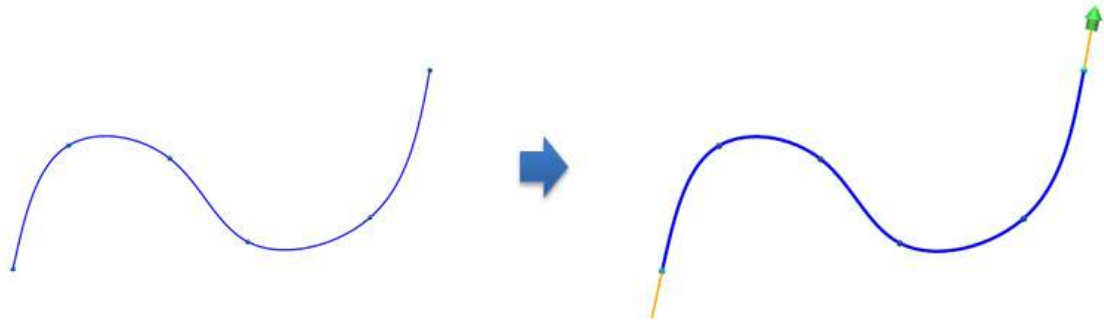
See Also

[Tools > Sketch Tools > Trim](#)

Tools > 3D Sketch Tools > Extend

What

The **Extend** command is used to extend 3D curves and lines.



Why

The **Extend** tool is useful for:

- Extending a 3D curve or line

Note:

Extending a curve in **3D Mesh Sketch mode** will follow mesh data. To extend a curve in a tangential direction without following the mesh, copy the curve in the **3D Mesh Sketch mode** into the **3D Sketch mode** and then apply the **Extend** command.

How to Use

How to extend a curve

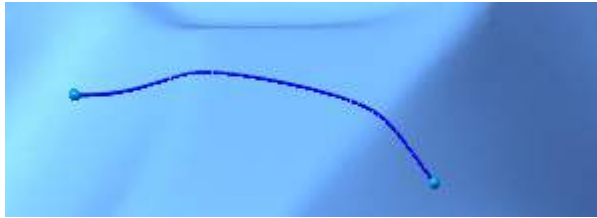
1. Go to **Tools > 3D Sketch Tools > Extend** or click the icon in the toolbar.



2. Select a curve to extend and click-drag the arrow on the screen. To extend a curve in both directions click on the curve itself, or to extend in only one direction click on the end of the curve in the desired direction.



3. Click **OK** to complete the command.



Options in Detail

Curves – Selects curves to extend

Options

Shape – Selects the shape of an extension.

Tangent – Extends a curve in a tangential direction from a starting position.

Curvature – Extends a curve in a curvature direction from a starting position.

Reflective – Extends a curve as a reflection of the original shape.

Distance – Sets a distance to extend.

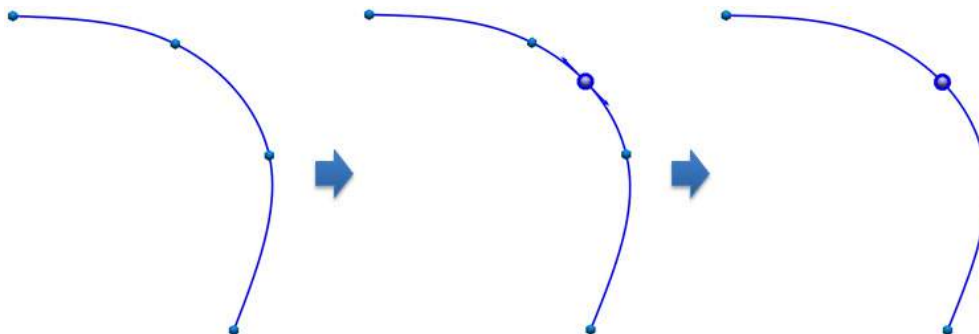
See Also

[Tools > Sketch Tools > Extend](#)

Tools > 3D Sketch Tools > Split

What

The **Split** command is used to divide a curve.



Why

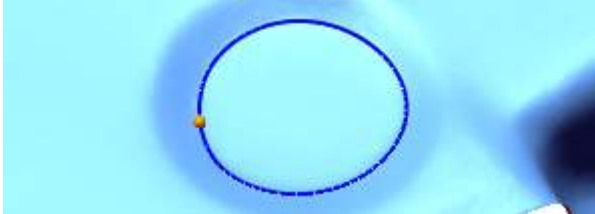
The **Split** command useful for:

- Dividing a curve by creating a node if one does not exist

How to Use

How to use the Split command to create nodes on a curve

1. Go to **Tools > 3D Sketch Tools > Split** or click the icon in the toolbar.



2. Pick points on the curve to split and click **OK**. The curve will now have nodes where the split command was used.



Note:

The **Split** command can move the seam position of a closed curve. A split position on a closed curved will become the new seam point.



Original Seam Point



Changed Seam Point

Options in Detail

Pick Points – Splits a curve at a picked point.

Curves At Intersection Points – Splits a curve at an intersecting point.

Plane Or Face Intersection Points – Splits a curve on intersecting positions with other planes or faces.

Curves – Selects curves to split.

Intersection Tolerance – Sets the tolerance for splitting curves at an intersection point.

Tool Entity – Selects tool entity.

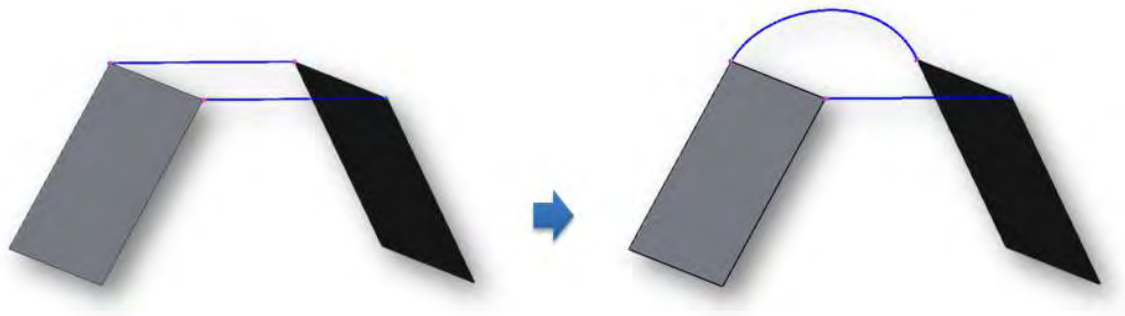
See Also

[Tools > Sketch Tools > Split](#)

Tools > 3D Sketch Tools > Match

What

The **Match** command is used to match tangencies and curvatures between a curve and another entity.



Why

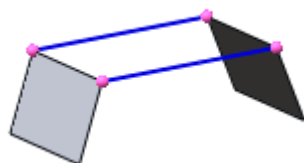
The **Match** command is useful for:

- Creating areas that a scanner might not be able to get data for

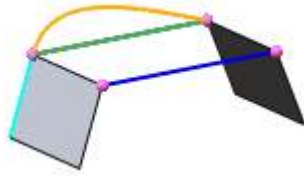
How to Use

How to extract a curve from two solid faces

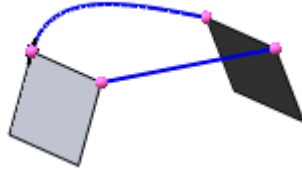
1. Click the **3D Sketch mode** icon.
2. Go to **Tools > 3D Sketch Tools > Match** or click the icon in the toolbar.



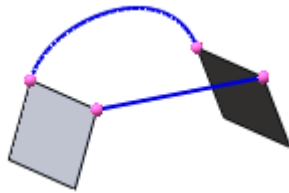
3. Select a **Curve** to match and a body edge as the **Target Entity**.



- Click **OK** to match the selected curves.



- Apply the matching process in steps 2-4 again on the other side of the model to complete the match.



Options in Detail

Tangent – Makes a tangency (G1) connection to the target entity.

Curvature – Makes a curvature (C2) connection to the target entity.

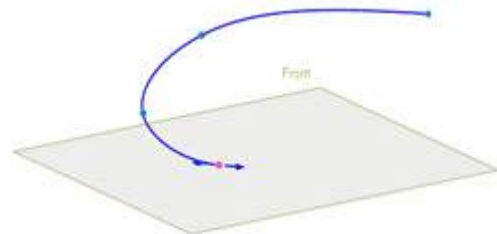
Orthogonal – Makes an orthogonal connection to the target entity. Only reference planes and faces can be used for this method.

Curve – Selects a curve to match.

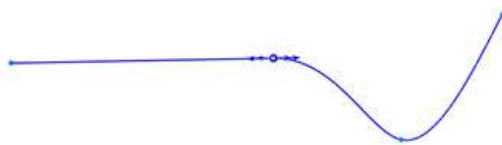
Target Entity – Selects an entity to use as a matching reference. The target entity can be a ref. vector, ref. plane, curve, edge, or face.



Curve to Ref. Vector



Curve to Ref. Plane



Curve to Curve



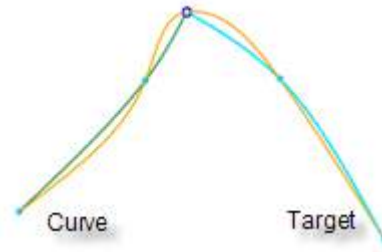
Curve to Edge

Curve to Face

Average – Modifies a target curve to create a smooth connection.




Disable Average



Enable Average

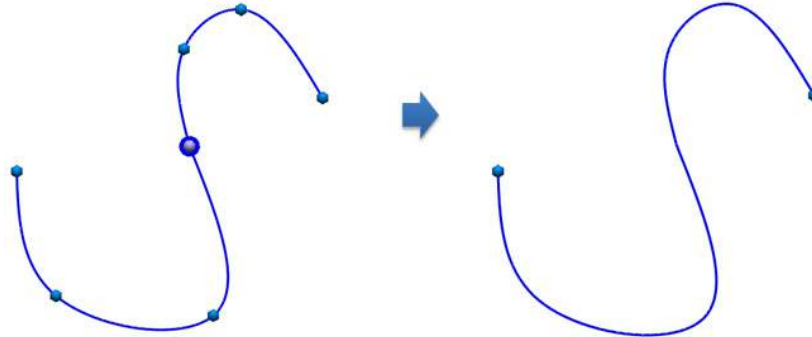
Note:

A matched constraint will be shown with blue arrows. To remove constraints, click the **Remove Constraint** button  on the toolbar and select the constraints in the list box.

Tools > 3D Sketch Tools > Merge

What

The **Merge** command is used to merge separate curves to form a single curve.



Why

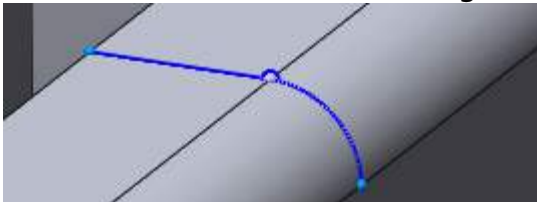
The **Merge** command is useful for:

- Merging separate lines into one line

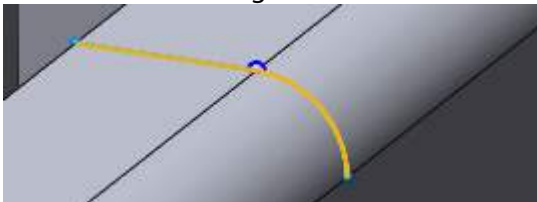
How to Use

How to merge curves

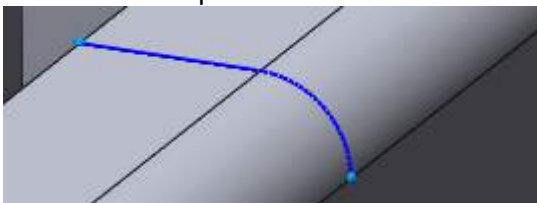
1. Go to **Tools > 3D Sketch Tools > Merge** or click the icon in the toolbar.



2. Select curves to merge.



3. Click **OK** to complete the command and merge the selected curves.



Options in Detail

Methods

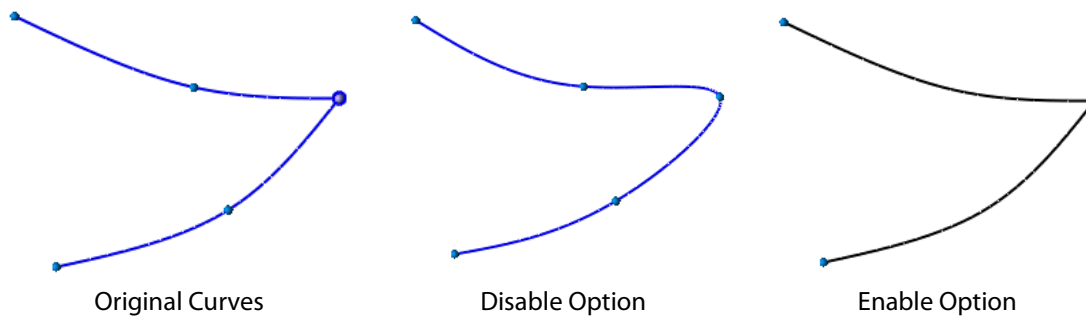
2 Curve Ends – Connects the ends of two curves.

Average Multiple Curves – Combines neighboring curves to generate a single averaged curve.

More Options

Curves – Selects curves to merge.

Fix Shape – Preserves the original shape of curves. This option can be used to preserve a sharp edge after merging and is available only in 3D Sketch mode.



Smoothness – Sets the smoothness when averaging multiple curves.

Merge Range – Merges the values between curves.

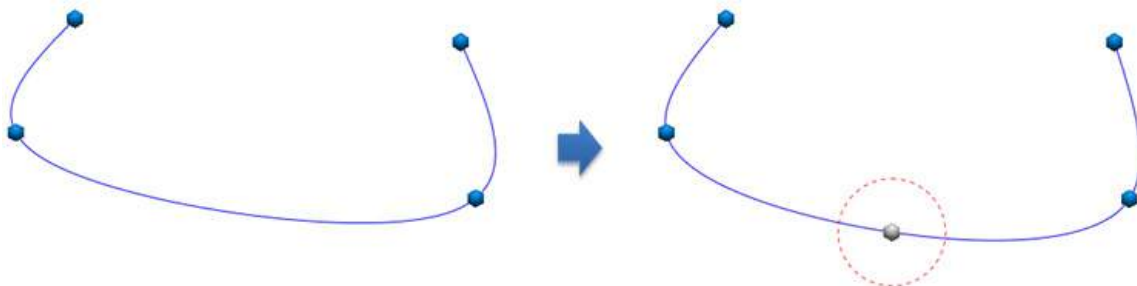
See Also

[Tools > Sketch Tools > Merge](#)

Tools > 3D Sketch Tools > Insert Node

What

The **Insert Node** command is used to insert a node onto a curve or line.



Why

The **Insert Node** command is useful for:

- Creating a break in a line
- Creating a node to connect another line to

How to Use

How to insert a node

1. Go to **Tools > 3D Sketch Tools > Insert Node** or click on the icon in the toolbar.



2. Click on a curve to insert nodes. Click **OK** to finish the command.



Note:

A node can be added onto a line or curve by dragging an existing node while holding down the **Ctrl** key on the keyboard.



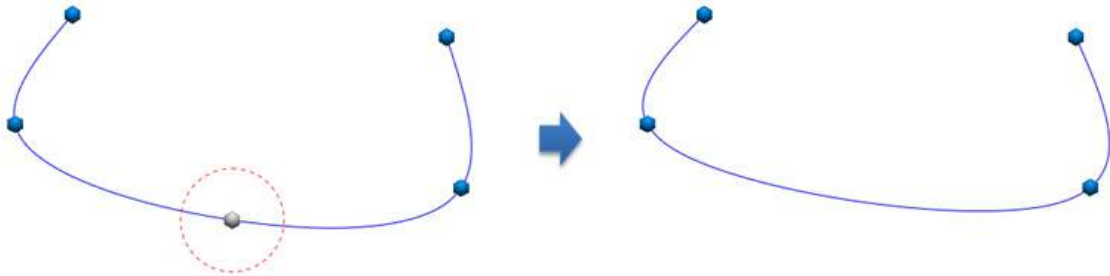
See Also

[Tools > 3D Sketch Tools > Remove Node](#)

Tools > 3D Sketch Tools > Remove Node

What

The **Remove Node** command is used to remove a node from a curve or line.



Why

The **Remove Node** command is useful for:

- Removing nodes in a line

How to use

How to remove a node

1. Go to **Tools > 3D Sketch Tools > Remove Node** or click on the icon in the toolbar.
2. Select nodes to remove.



Before Node Removal



After Node Removal

Note:

A node can be removed by pressing the **Delete** key on the keyboard.

See Also

[Tools > 3D Sketch Tools > Insert Node](#)

Tools > 3D Sketch Tools > Remove Constraint

What

The **Remove Constraint** command is used to remove existing constraints on curves and nodes.



Why

The **Remove Constraint** command is useful for:

- Removing constraints on curves and nodes
- Removing temporary created constraints to make modeling easier

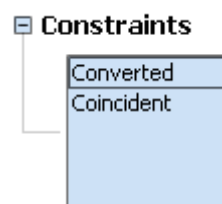
How to Use

How to remove constraints on a curve

1. Go to **Tools > 3D Sketch Tools > Remove Constraint** or click the icon in the toolbar.



2. Select a curve and all constraints on the curve will be displayed.



3. Select constraints to be removed and click **OK** to complete the command. Constraints will be removed and the converted curve will be displayed in blue.



Options in Detail

Curve / Nodes – Selects curves or nodes to remove constraints.

Constraints To Remove – Shows a list of constraints that can be removed by selecting the constraint in the list box and clicking the **Delete** button on the keyboard.

Note:

Constraints can also be removed by using the Trash Can icon that will appear next to a constraint when selected.

No. Of Interpolation Points – Sets the number of interpolation points and regenerates the curve when removing constraints from an imported curve.

7.5.12. Tools > Body Tools

The **Body Tools** menu features methods to deal with surfaces.

Tools > Body Tools > Find Defect

What

The **Find Defect** command is used to detect defects on solid and surface bodies.



Why

The **Find Defect** command is useful for:

- Detecting defects that are too small to see manually
- Finding the location of defects
- Finding self-intersected or discontinuous faces

How to Use

How to find defects on a solid body

1. Go to **Tools > Body Tools > Find Defect** or click the icon in the toolbar.



2. Select a body and check the **Self Intersection Test** and **Face Continuity Test** options to set the defect detection parameters. Click the **Next Stage** button to continue to the next step.
3. Defective faces that self-intersect or have continuity issues will be displayed on the model in a different color.



Options in Detail

Bodies – Selects bodies to inspect.

Self Intersection Test – Detects self-intersecting faces.

Face Continuity Test – Detect discontinuous faces.

See Also

[Tools > Body Tools > Find Defect](#)

7.5.13. Tools > Align

The **Align Tools** menu features methods to align mesh data to global or user defined coordinates or original CAD data.

The **Align Tools** menu provides 3 sets of alignment tools used to:

Align by Wizards

- **Wizard**
Aligns a mesh to a desired set of coordinates
- **Interactive Alignment**
Interactively aligns a mesh using one of several available methods

Align by Fitting

- **Quick Fit**
Roughly aligns a mesh to a solid body by matching principal axis
- **Best Fit**
Registers a scan to a body as accurately as possible by utilizing the overlapping regions between them

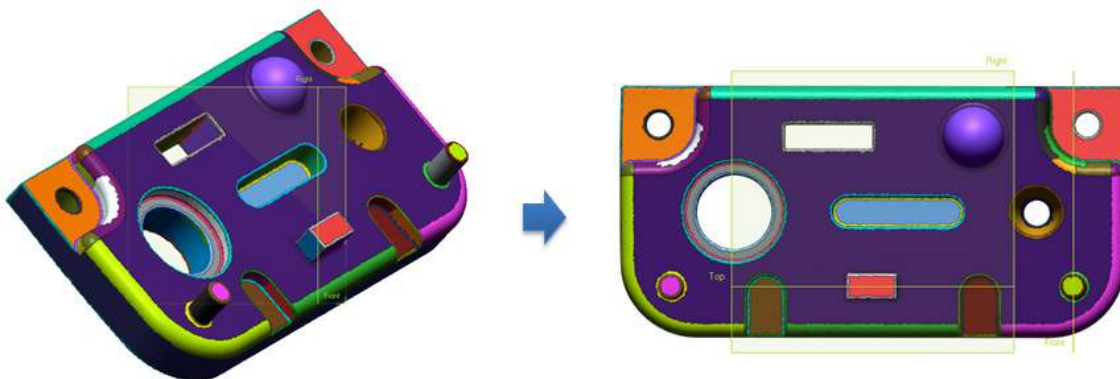
Align by Datums

- **Datum Match**
Registers a scan to a body by using datums

Tools > Align > Wizard

What

The **Align Wizard** command is used to align a mesh to a desired set of coordinates.



Why

The **Align Wizard** command is useful for:

- Finding and choosing suitable candidate alignments
- Extracting alignments from mesh features

How to Use

How to find and select a coordinate candidate

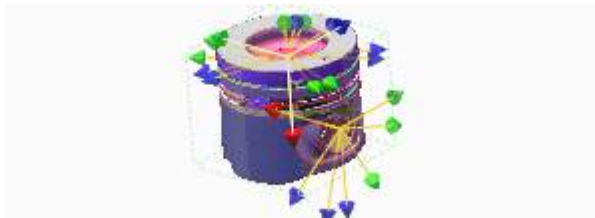
1. Go to **Tools > Align > Wizard** or click the icon in the toolbar.



Note:

A mesh needs regions before executing the **Align Wizard** command. Regions can be created by using the **Auto Segment** command in the **Region Group mode**.

2. Select a mesh and click the **Next Step** icon. The application will show candidate coordinates for the mesh.



3. Select **Coordinate 1** and click the **OK** button to align to that coordinate and finish the command.



Options in Detail

Stages

This command consists of the following stages:

- **Moving Mesh**

Select a mesh or point cloud to align

- **Candidate**

Chooses a candidate coordinate. The ideal coordinate is listed at the top of the list box.

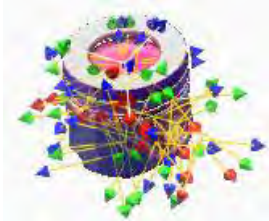
Stage 1/2 – Moving Mesh

Moving Mesh – Selects a mesh or point cloud to align.

Stage 2/2 – Candidate

Candidate – Chooses a candidate coordinate.

Max. Num Of Candidate – Controls the maximum number of candidate coordinates.



Max. Number of Candidates



Min. Number of Candidates

See Also

[Tools > Align > Interactive Alignment](#)

[Tools > Align > Quick Fit](#)

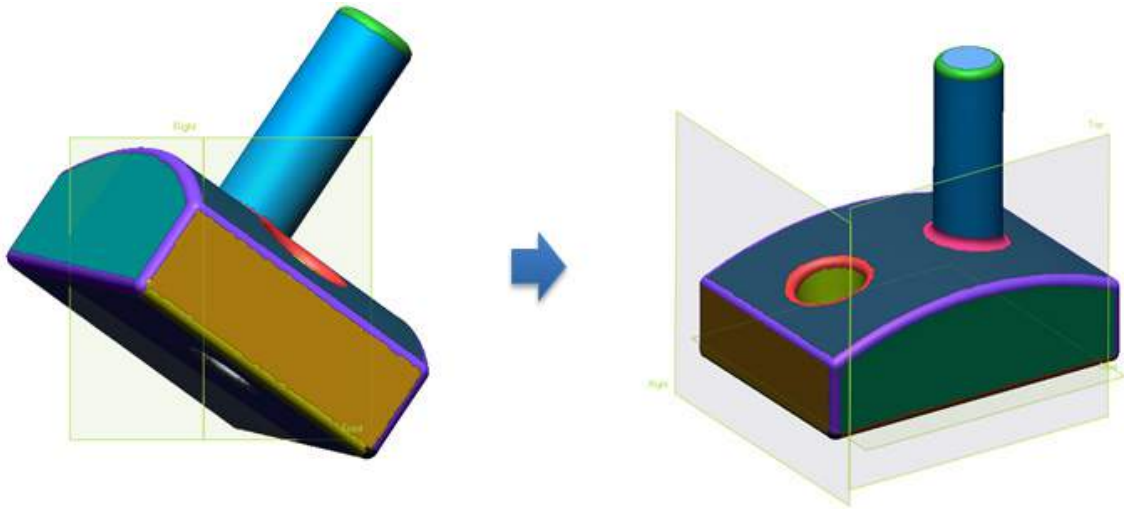
[Tools > Align > Best Fit](#)

[Tools > Align > Datum Match](#)

Tools > Align > Interactive Alignment

What

The **Interactive Alignment** command is used to interactively align a mesh using one of several available methods.



Why

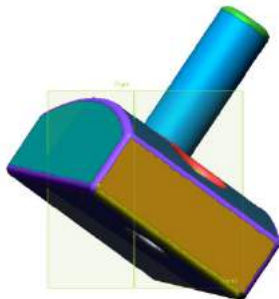
The **Interactive Alignment** command is useful for:

- Manually aligning a model using one of two different alignment types

How to Use

How to use the X-Y-Z interactive alignment method

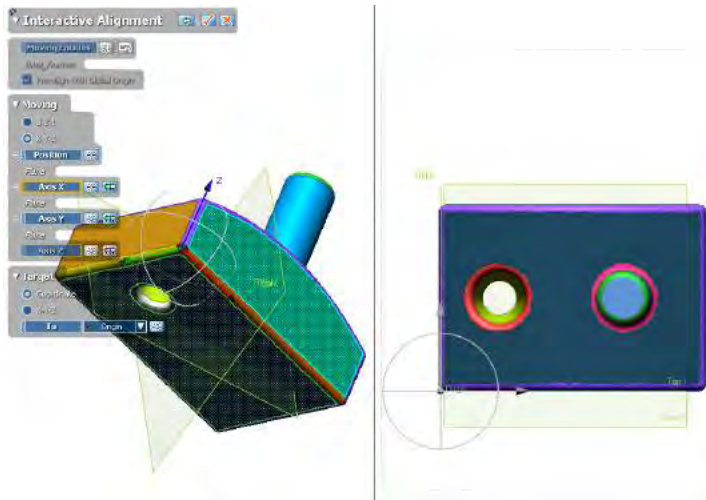
1. Go to **Tools > Align > Interactive Alignment** or click the icon in the toolbar.



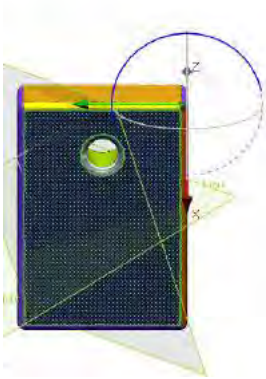
Note:

A mesh needs regions before executing the **Interactive Alignment** command. Regions can be created by using the **Auto Segment** command in the **Region Group mode**.

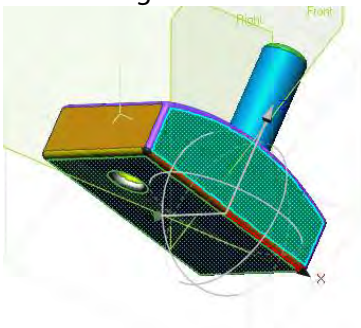
2. Select a mesh and click the **Next Stage** button to divide the **Model View** vertically into two views. The left view is the **Source** and the right view is the **Reference**. Choose the **X-Y-Z** method.



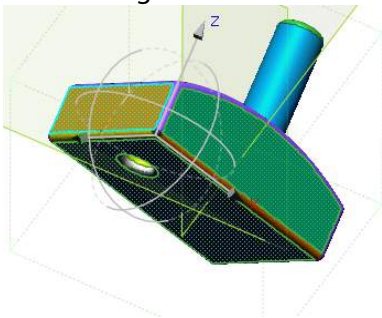
3. Click the **Plane** button and select the bottom planar region.



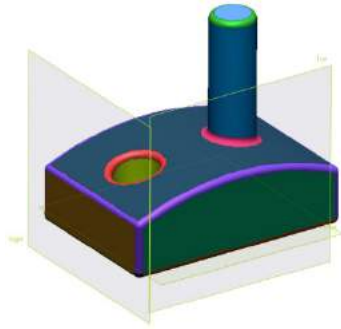
4. Click the **Vector** button and select a side region that perpendicular to the previously selected region.



5. Click the **Position** button and select the side region perpendicular to the previously selected region.



6. Click **OK**. Each perpendicular face is now aligned to the reference planes.



Options in Detail

Stages

This tool consists of the following stages:

- **Moving Entities**

Select a mesh or point cloud to align

- **Moving**

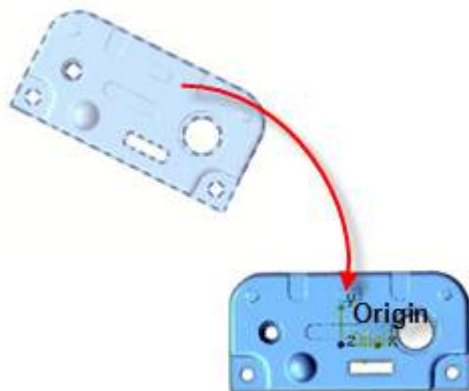
3-2-1 and **X-Y-Z** are the available alignment methods in this step. The **Target** can be set to either Coordinate or **X-Y-Z**.

Stage 1/2 – Choosing Moving Entities

This stage is for choosing the **Moving Entities** to align. The chosen entities can be aligned to the global origin.

Moving Entities – Selects meshes or point clouds to align.

Pre-align With Global Origin – Transforms mesh to the global origin. The application will find the minimal bounding box of the mesh and set the longest and the shortest direction to the X-axis and Z-axis respectively, aligning to the global coordinate.

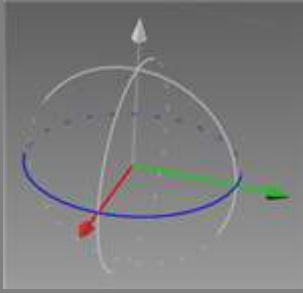


Stage 2/2 – Moving Alignment Method


The **3-2-1** and **X-Y-Z** alignment methods are available in this step as well as the ability to set the **Position**, **Axis X**, **Axis Y** and **Axis Z** parameters for the alignment.

Note:

Scans can be manually aligned using the manipulators in the **Model view**. The scans can be transformed by dragging the **X**, **Y**, and **Z** directional arrows, or rotated by dragging the appropriate circle. Entities selected as a **Plane** or a **Vector** will have constraint directions and axes dimmed in grey.




Moving

3-2-1 – Alignment using entities on the mesh as a moving plane, moving vector, and moving position. The **Flip** button  reverses the direction of the selection.

Plane – Sets an entity as the moving plane.

Vector – Sets an entity as the moving vector.

Position – Sets an entity as the moving position.

X-Y-Z – Alignment using entities as moving points and axes. The **Flip** button  reverses the direction of the selection.

Position – Sets an entity as the moving position.

Axis X – Sets an entity as the moving X axis.


Axis Y – Sets an entity as the moving Y axis.

Axis Z – Sets an entity as the moving Z axis.

X-Y-Z

Coordinate – Alignment using target coordinates.

To – Sets the coordinate to align to.

X-Y-Z – Alignment using entities as points and axes. The **Flip** button  reverses the direction of the selection.

Position – Sets an entity as the target position.

Axis X – Sets an entity as the target X axis.

Axis Y – Sets an entity as the target Y axis.

Axis Z – Sets an entity as the target Z axis.

See Also

[Tools > Align > Wizard](#)

[Tools > Align > Quick Fit](#)

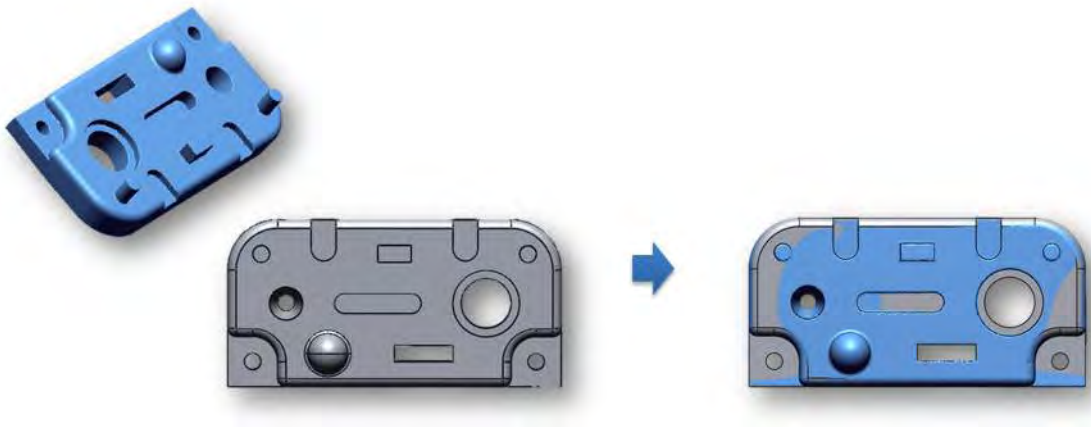
[Tools > Align > Best Fit](#)

[Tools > Align > Datum Match](#)

Tools > Align > Quick Fit

What

The **Quick Fit** command is used to roughly align a mesh to a solid body by matching principal axis.



Why

The **Quick Fit** command is useful for:

- Comparing an original scan to a CAD model

How to Use

How to use the Quick Fit command to align a mesh to a solid body

1. Go to **Tools > Align > Quick Fit** or click the icon in the toolbar.



2. Select a mesh as the **Moving Entities** and a body as the **Target Body**.
3. The **Best Fit Alignment** command will automatically execute after the **Quick Fit** command. Use the default setting and click the **OK** button to finish the command. The mesh will now be aligned to the solid body.



Options in Detail

Moving Entities – Selects a mesh or point cloud to align.

Target Body – Selects a body to align.

More Options

Run Best Fit Alignment – Automatically applies **Best Fit** alignment.

See Also

[Tools > Align > Wizard](#)

[Tools > Align > Interactive Alignment](#)

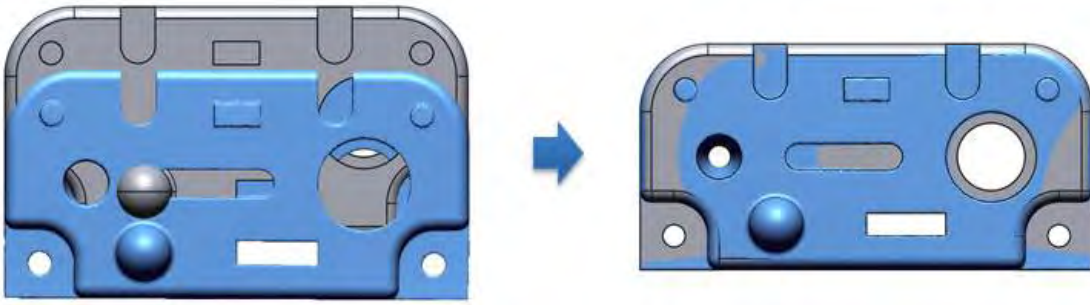
[Tools > Align > Best Fit](#)

[Tools > Align > Datum Match](#)

Tools > Align > Best Fit

What

The **Best Fit** command registers a scan to a body as accurately as possible by utilizing the overlapping regions between them.



Why

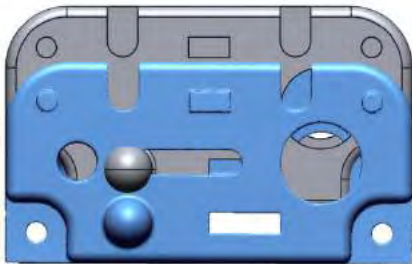
The **Best Fit** command is useful for:

- Comparing an original scan to a CAD model

How to Use

How to use the Best Fit command to align a mesh to a solid body

1. Go to **Tools > Align > Best Fit** or click the icon in the toolbar.



2. Select a mesh as the **Moving Entities** and a body as the **Target Body**.
3. Click **OK** to complete the command.

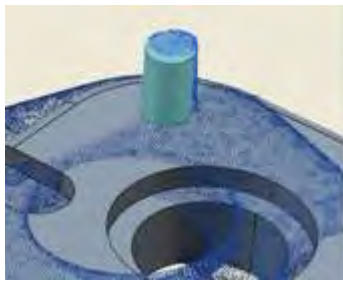


Options in Detail

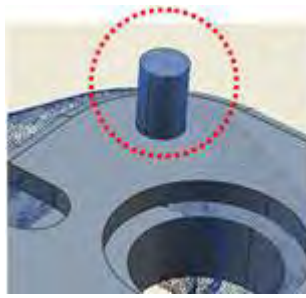
Method

Global – Applies the Best Fit alignment command to an entire mesh.

Partial – Aligns features of scan data with a body. Faces on the body can be used as Reference targets.



Before Alignment



After Partial Alignment

Selection

Moving Entities – Selects a mesh or point cloud.

Target Body – Selects a solid or surface body.

Best Fit Options

Max. Distance – Sets the maximum distance between alignment entities.

Sampling Ratio – Samples data points by a specified value. When the ratio is 100%, all selected data will be used. When the ratio is 50%, half of the selected data will be used.

Max. Iteration Count – Limits the number of times a registration can reiterate.

Max. Average Deviation – Sets deviation criteria to determine if registration results are acceptable and can be manually input, automatically calculated, or intelligently decided by the application wizard.

Compensate Offset – Compensates for the deviation from an offset between the design data and the scan data.



Before Alignment



Compensate Offset: 2

Constraint Options

Anchor Type – Defines the reference position for constraints of translation or rotation. Selects the type of anchor used in the command. If the **Scan Data Center** option is selected, the center of the bounding box for selected scan data is shown and used as the anchor point. If the **Model Center** option is selected, the center of the target model will be used as an anchor point. If **Local Coordinate** is selected, you can define custom coordinate as the anchor point. If the **Global Origin** option is selected, the origin point in the global coordinate will be used as the anchor point.

Lock Translation/Rotation – Limits movement and rotation in each direction while scan data is aligned.

More Options

Ignore Outlier – Ignores irregular data during alignment.

See Also

[Tools > Align > Wizard](#)

[Tools > Align > Interactive Alignment](#)

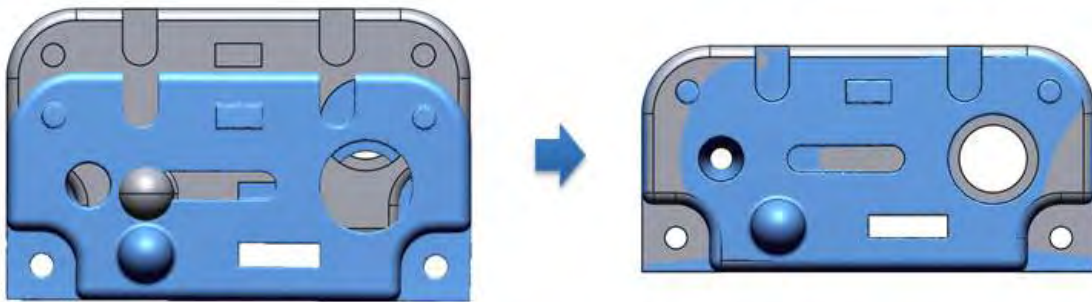
[Tools > Align > Quick Fit](#)

[Tools > Align > Datum Match](#)

Tools > Align > Datum Match

What

The **Datum Match** command registers a scan to a body by using datums.



Why

The **Datum Match** command is useful for:

- Comparing an original scan to a CAD model
- Aligning a scan coordinate system to the global coordinate system or another specific coordinate system

How to Use

How to use the Datum Match command to align a mesh to a solid body

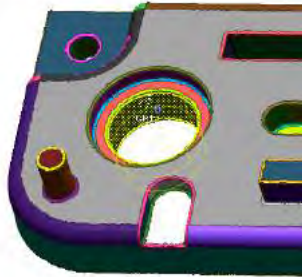
1. Go to **Tools > Align > Datum Match** or click the icon in the toolbar.



Note:

A mesh needs regions before executing the **Datum Match** command. Regions can be created by using the **Auto Segment** command in the **Region Group mode**.

2. Select a mesh and click the **Next Stage** button.
3. Select the large cylindrical region on the mesh and select the matching region in the design data and click the **Flip** button.



Cylinder on Mesh Data



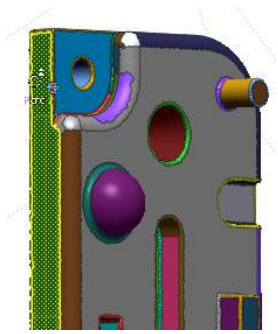
Cylinder on Design Data

4. The cylinders will be matched to one another.

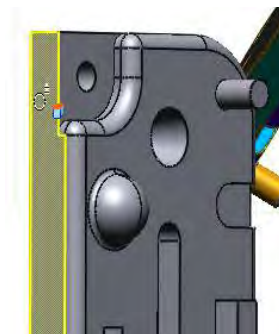


Cylinder on Mesh Data

5. Select the planar region on the mesh and design data.

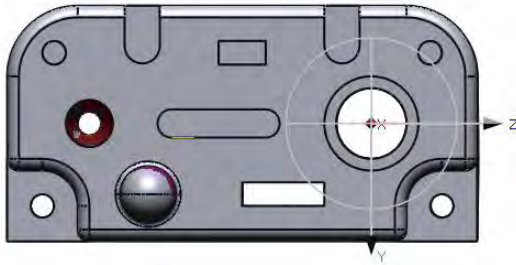


Plane on Mesh Data



Plane on Design Data

6. Click **OK** to finish the command.



Options in Detail

Stages

This tool consists of 2 stages:

- **Moving Entities**
Selects a mesh or point cloud to align
 - **Datum Pairs**
Selects pairs of datums from a mesh and body to align
-

Stage 1/2 – Moving Entities

Moving Mesh – Selects a mesh or point cloud to align.

Stage 2/2 – Datum Pairs

Datum Pairs – Selects datum pairs to match and align.

See Also

[Tools > Align > Wizard](#)

[Tools > Align > Interactive Alignment](#)

[Tools > Align > Quick Fit](#)

[Tools > Align > Best Fit](#)

7.6. Measure

The **Measure** menu features functions to measure a distance between plane-plane, plane-line, plane-point, line-line, line-point and point-point.

It also provides methods to measure an angle between plane-plane, plane-line, plane-point-point, line-line, line-point-point and point-point-point. The radius of an entity can be measured as well.

The **Measure** menu provides tools used to:

Measure

- **Distance**

Measure the distance between entities

- **Angle**

Measure the angle between two edges or three picked points

- **Radius**

Measure the radius of an edge or three picked points

- **Section**

Temporarily create sections on a mesh and body to measure the deviation between them

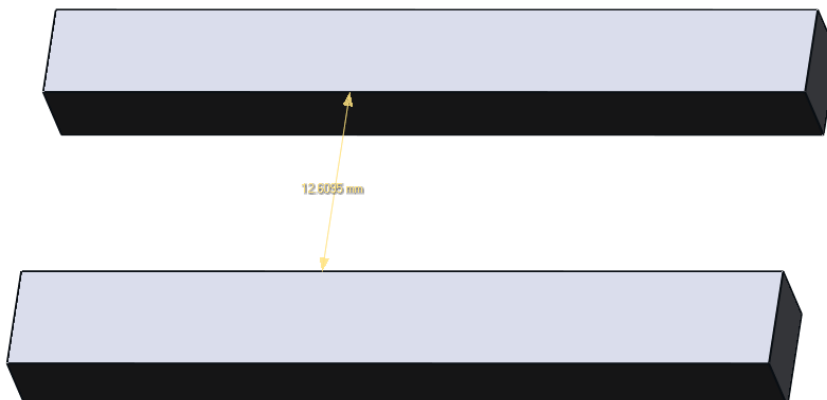
- **Mesh Deviations**

Show the deviation between point clouds and meshes using a defined color map

7.6.1. Measure > Distance

What

The **Distance** command measures the distance between entities.



Why

The **Distance** command is useful for:

- Comparing distances between plane-plane
- Comparing distances between plane-line
- Comparing distances between plane-point
- Comparing distances between line-line
- Comparing distances between line-point
- Comparing distances between point-point

Note:

Plane-plane, line-line and plane-line measurements should be parallel.

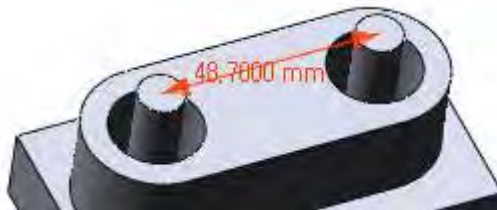
How to Use

How to find the distance between two edges

1. Go to **Measure > Distance** or click the icon in the toolbar.



2. To measure the distance between two cylinders, click the cylinder edges. When doing a mouse-over a measurement the measured entities will be shown.



Note:

To pick a position on a line or a plane, hold down the **Ctrl** button on the keyboard while clicking.

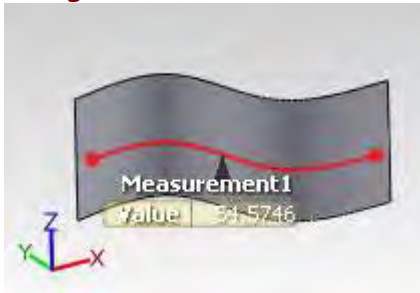
Options in Detail

Methods

Linear – Measures distances between entities with the **Auto** method automatically assessing

entity types.

Along Faces – Measures distance along the surface of a model between two points.

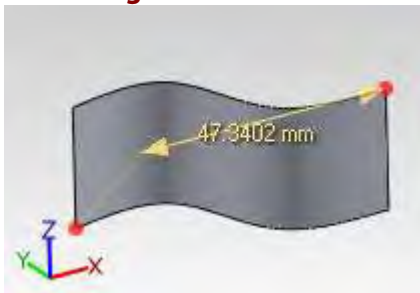


Axis Align

No Align – Calculates the direct distance between entities.



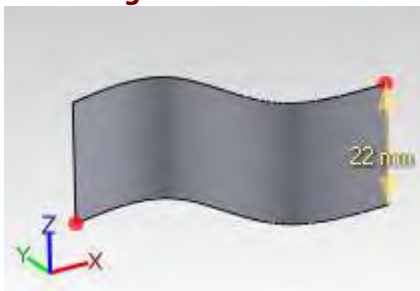
X Axis Align – Calculates the distance in the X direction.



Y Axis Align – Calculates the distance in the Y direction.



Z Axis Align – Calculates the distance in the Z direction.



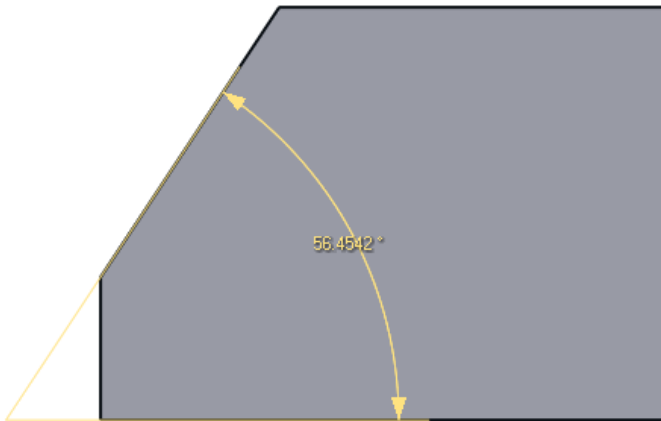
See Also

[Tools > Sketch Entities](#)

7.6.2. Measure > Angle

What

The **Angle** command measures the angle between two edges or three picked points.



Why

The **Angle** command is useful for:

- Comparing angles between plane-plane
- Comparing angles between plane-line
- Comparing angles between plane-point
- Comparing angles between line-line
- Comparing angles between line-point-point
- Comparing angles between point-point-point

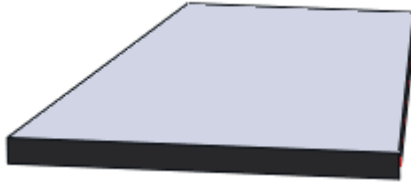
Note:

Angles between parallel entities cannot be measured.

How to Use

How to find the angle between two edges

1. Go to **Measure > Angle** or click on the icon in the toolbar.



2. Select two edges to measure the angle. Click **OK** to finish the command. When doing a mouse-over a measurement the measured entities will be shown.



Note:

To pick a position on a line or a plane, hold down the **Ctrl** button on the keyboard while clicking.

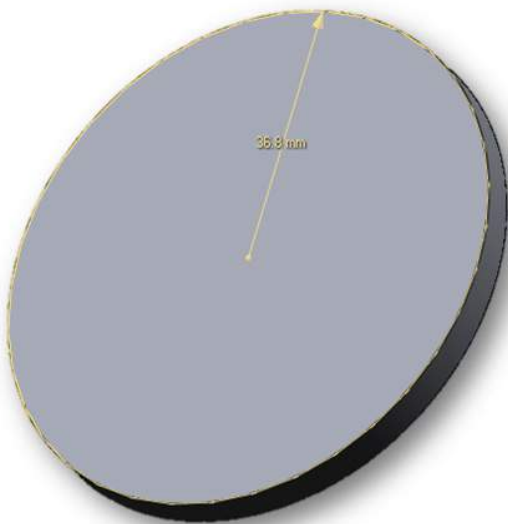
See Also

[Tools > Sketch Entities](#)

7.6.3. Measure > Radius

What

The **Radius** command measures the radius of an edge or three picked points.



Why

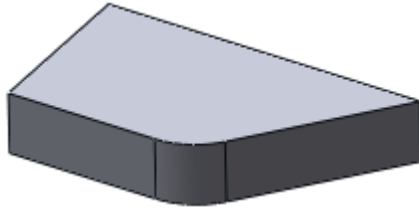
The **Radius** command is useful for:

- Finding the radius of a circle
- Comparing angles between point-point-point

How to Use

How to find the radius of an edge

1. Go to **Measure > Radius** or click the icon in the toolbar.



2. Select an edge to measure a radius from. Click **OK** to finish the command. When doing a mouse-over a measurement the measured entities will be shown.



Options in Detail

Methods

Auto – Automatically finds a radius based on a feature shape

3 Points – Finds a radius based on three selected points.

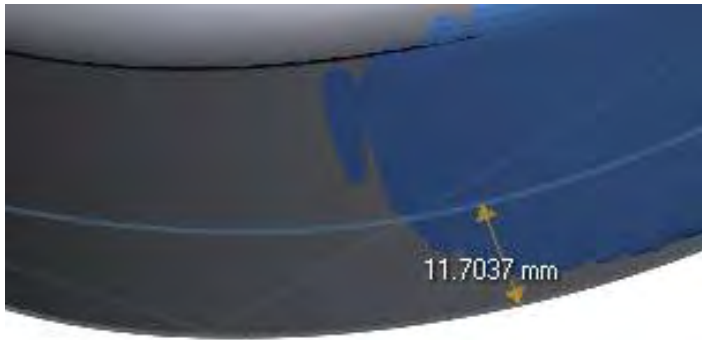
See Also

[Tools > Sketch Entities](#)

7.6.4. Measure > Section

What

The **Section** command temporarily creates sections on a mesh and body to measure the deviation between them.



Why

The **Section** command is useful for:

- Checking the deviation between reverse engineered models and original mesh

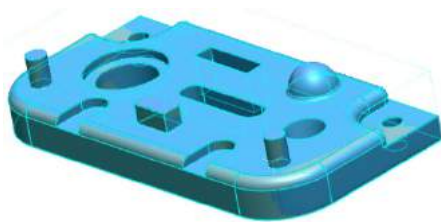
How to Use

How to find sectional differences between a body and a mesh

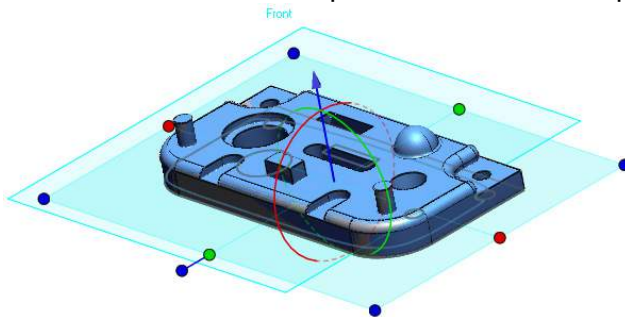
1. Go to **Measure > Section** or click the icon in the toolbar.



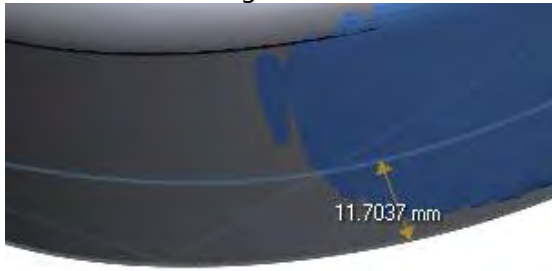
2. Select a mesh and a solid body as the **Target Entity** and click the **Next Stage** button.



3. Choose the **Select Plane** option and select a base plane.



4. Choose the **Min Distance** option under **Measure Method** to find the minimum distance between the body and mesh. Select a point on the body section and then pick the scan section to get the distance.



5. Click **OK** to finish the command.

Options in Detail

Select Entities

This tool consists of 2 stages :

- **Target Entity**
Selects an entity to section
 - **Section Type**
Chooses a candidate coordinate
-

Stage 1/2 – Target Entity

Target Entity – Selects a target entity to measure.

Stage 2/2 – Section Type

Draw Line On Screen – Draws a line on the screen that creates a virtual cutting line used to measure a section.

Select Plane – Selects a plane to use as a cutting plane for measuring a section.

Base Planes – Selects a base plane.

N-Division Between Planes – Uses N-Divisions between planes to measure a section.

Base Plane – Selects a base plane.

N-Division Along Curve – Uses N-Division along a curve to measure a section.

Path Curve – Selects a path curve.

Radial – Uses a radius to measure a section.

Axis – Selects an axis.

Base Plane – Selects a base plane.

Cylindrical – Uses a cylinder to measure a section.

Axis – Selects an axis.

Base Plane – Selects a base plane.

Conical – Uses a cone to measure a section.

Axis – Selects an axis.

Base Plane – Selects a base plane.

Sub Methods

Uniform – Creates uniform sections based on set distance between sections or set number of sections.

Curvature – Creates a higher concentration of sections in higher curvature areas.

Pick Points – Creates sections based on manually picked points.

Options

Flip Start Plane – Flips the direction of the starting plane.

No. Of Sections – Sets the number of sections.

Distance Between Sections – Sets the distance between sections.

Equal Spacing – Sets the spacing between sections at an equal amount.

Total Angle – Sets the total angle of a cylindrical and conical section.

Show Section Base Geometry – Shows the base geometry for sections.

Radius Interval – Sets the total radius interval of a cylindrical and conical section.

Bottom Radius – Sets the total bottom radius of a conical section.

Top Radius – Sets the total top radius of a conical section.

Apex Angle – Sets the angle of the apex for a conical section.

Base Radius – Sets the base radius for a cylindrical section.

Height – Sets the total height for a cylindrical and conical section.

Section Inside – Sets sections inside the base geometry

Note:

If the option is unchecked, the sections will be set outside of the base geometry.

Base Position – Sets the base position for a conical section

Top – Sets the base position to the Top.

Center – Sets the base position to the Center.

Bottom – Sets the base position to the Bottom.

Position – Sets the X, Y, and Z position for a conical section.

X – Sets the X position.

Y – Sets the Y position.

Z – Sets the Z position.

Measure Method

Distance By Picking – Measures the distance between two selected points.

Min. Distance – Measures the minimum distance from a selected point to a section. The two selected entities should be on the same plane.

Normal Distance – Measures the distance from the normal direction from selected point to a section. The two selected entities should be on the same plane.

Section Property – Shows the properties of a selected section. Only a scan section can be selected.

Create Ref. Plane

Planar Sections – Selects planar sections for the creation of a Ref. Plane at the position of a sectional plane that will remain after exiting the **Section** command.

Create Ref. Polyline

Sections – Selects sections for the creation of a Ref. Polyline at the position of a sectional plane that will remain after exiting the **Section** command.

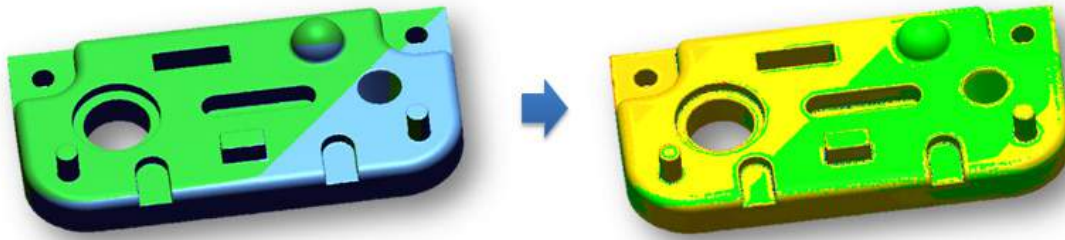
See Also

[Tools > Sketch Entities](#)

7.6.5. Measure > Mesh Deviations

What

The **Mesh Deviations** command shows the deviation between point clouds and meshes using a defined color map.



Why

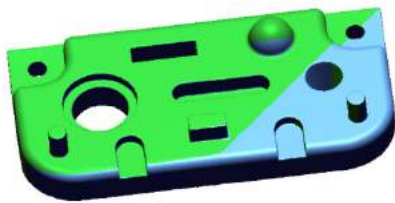
The **Mesh Deviations** command is useful for:

- Checking the deviation between many meshes and point clouds

How to Use

How to find deviation between meshes

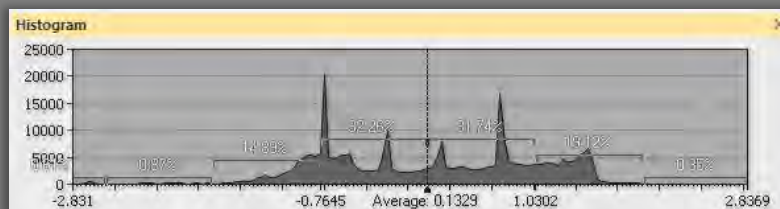
1. After pre-aligning mesh using the **Align Between Scan Data** command, go to **Measure > Mesh Deviations** or click the icon in the toolbar.



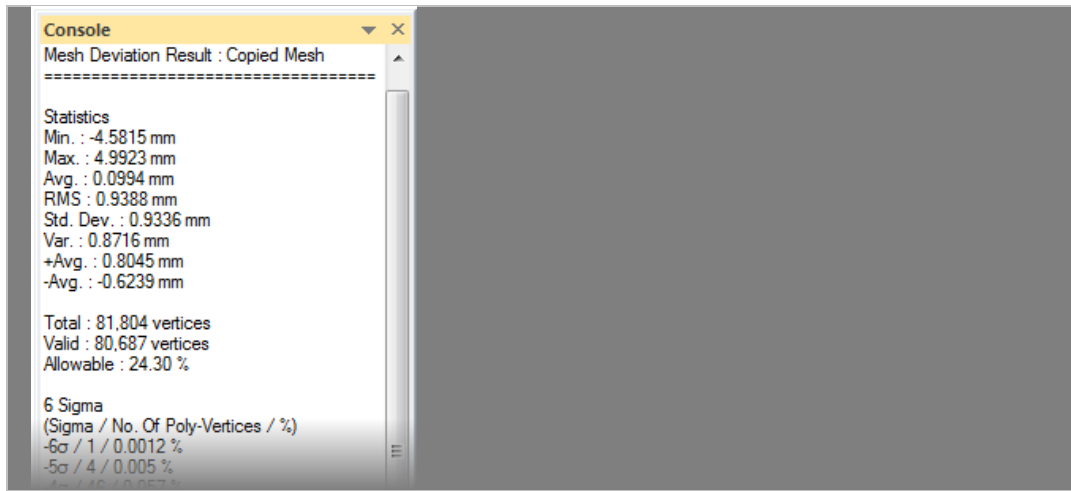
2. Select all meshes as **Target Entities**. Click the **Next Stage** button.

Note:

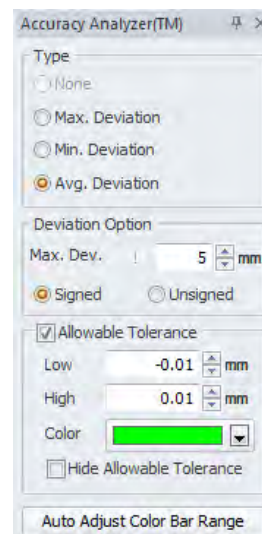
Histogram and **Entity Thumbnails View** windows will automatically open. Check the results in the windows.



The results will be recorded in the Console window. If needed, copy the results and paste it into other applications. For more information, see [Console](#) page.



3. In the **Accuracy Analyzer(TM)** panel check the **Avg. Deviation** option and uncheck the **Hide Allowable Tolerance** option.



4. Click **Cancel** to exit the command.

Options in Detail

Stages

This tool consists of 2 stages :

- **Target Entities and References**

Selects an entities and References for deviation analysis

- **Deviation Analysis**

Chooses a candidate coordinate. The ideal coordinate is listed at the top of the list box.

Stage 1/2 – Target Entities and References

Target Entities – Chooses target entities.

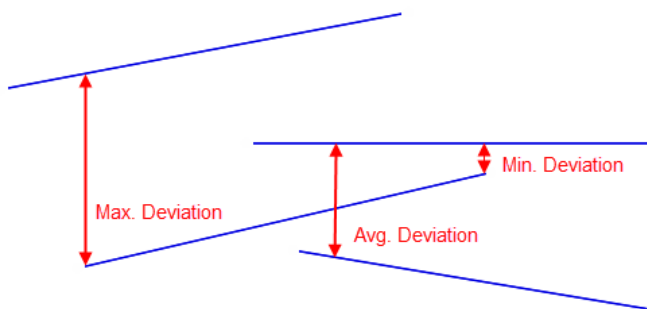
References – Chooses reference entities.

Stage 2/2 – Deviation Analysis

There are no options for this stage. However, deviations between target entities and target entities, and target entities and references can be analyzed by using the Accuracy Analyzer™ panel.

Accuracy Analyzer™ Panel

Type – Creates a color map will be calculated with Max. Deviation, Min. Deviation or Avg. Deviation when comparing three or more meshes.



None – No deviation will be analyzed.

Max. Deviation – Maximum allowed deviation will be analyzed.

Min. Deviation – Minimum allowed deviation will be analyzed.

Avg. Deviation – Averages of the max. and min. deviation will be analyzed.

Deviation Option – Sets options for deviation analysis.

Max. Dev. – Sets the Max. Dev. limit in mm.

Signed – Shows the deviation color map as positive and negative distances.

Unsigned – Shows the deviation color map as an absolute distance.

Allowable Tolerance – Sets the range of allowable deviation.

Low – Sets the minimum allowable deviation.

High – Sets the maximum allowable deviation.

Color – Sets the color bar range for deviations.

Hide Allowable Tolerance – Hides deviation that does not exceed the minimum or maximum allowable deviation.

Auto Adjust Color Bar Range – Sets the color bar range automatically.

See Also

[Docking Bars > Accuracy Analyzer](#)

7.7. View

The **View** menu features commands to change how a model is displayed, change and split viewpoints, as well as other viewing related functions.

The **View** menu provides tools used to:

Manage Visibility

- **[Visibility](#)**
Select different entity types to display or hide

Zoom

- **[Zoom](#)**
Zoom an area of interest, fit a model to a screen, or use the picture-in-picture (PIP) function

Manage Mesh Display Mode

- **[Mesh Display Mode](#)**
Set how meshes are displayed in the model view

Manage Body Display Mode

- **[Body Display Mode](#)**
Set how solid surfaces and bodies are displayed in the model view

Manage Viewpoints

- **[Viewpoint](#)**
Sets how a model is displayed in the model view

Manage Viewports

- Viewports

Set the number of viewing windows and how they are displayed

Manage Light Properties

- Light Properties

Set the global lighting properties such as color and point of origin within the model view

Clip a View

- View Clip

Cut away a part of a model using cutting planes or a box to view inside of it

Walk Through a Model

- Walk-through

Set the view to a first-person camera view to “walk-through” a model

7.7.1. View > Visibility

The **Visibility** menu features methods to show or hide entities in the Model View.

Global Control

- Show Selected Only

Display only selected entities

- Show All

Show all entities

- Hide All

Hide all entities

Individual Control

- Meshes

Show or hide meshes

- **Regions**

Show or hide regions

- **Point Clouds**

Show or hide point clouds

- **Surface Bodies**

Show or hide surface bodies

- **Sketches**

Show or hide sketches

- **3D Sketches**

Show or hide 3D sketches

- **Ref. Points**

Show or hide reference points

- **Ref. Vector**

Show or hide reference vectors

- **Ref. Planes**

Show or hide reference planes

- **Ref. Polylines**

Show or hide reference polylines

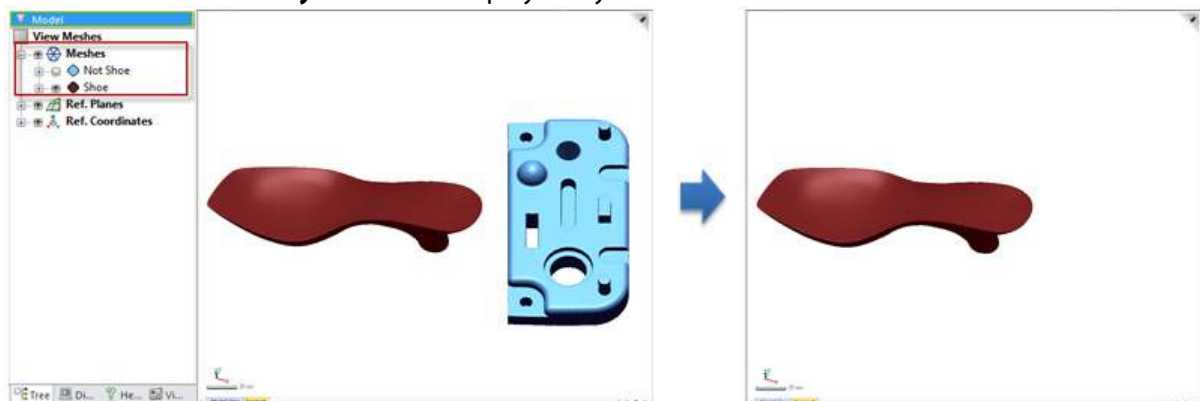
- **Ref. Coordinates**

Show or hide reference coordinates

View > Visibility > Show Selected Only

What

The **Show Selected Only** command displays only selected entities in the Model View.



Why

The **Show Selected Only** command is useful for:

- Focusing on only specific entities

Note:

This command will be unavailable and dimmed if no entities have been selected.

How to Use

How to show or hide selected entities

1. Select entities. Go to **View > Visibility > Show Selected Only** or click the icon in the toolbar.

See Also

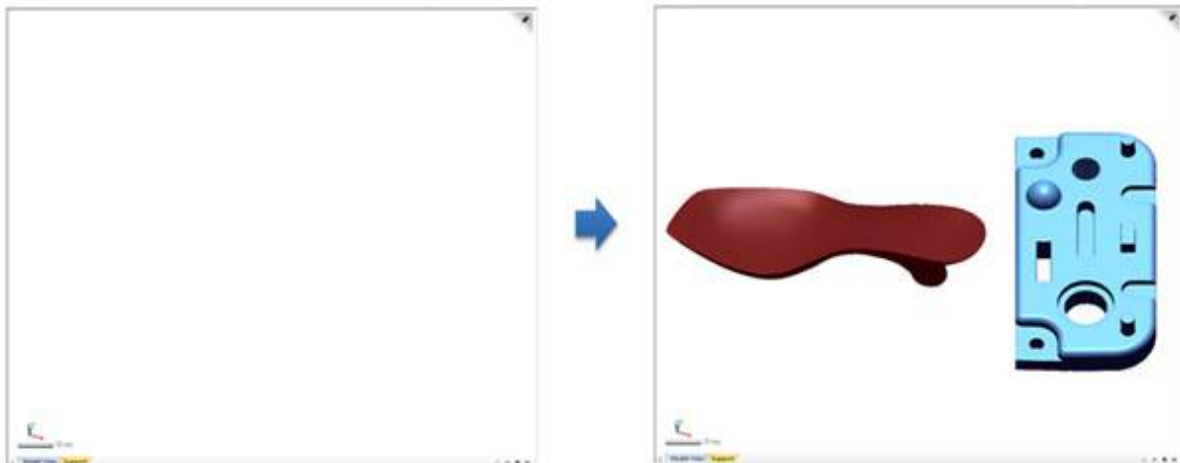
[View > Visibility > Show All](#)

[View > Visibility > Hide All](#)

View > Visibility > Show All

What

The **Show All** command shows all entities in the Model View.



Why

The **Show All** command is useful for:

- Quickly showing all hidden entities

How to Use

How to show all entities

1. Go to **View > Visibility > Show All** or click the icon in the toolbar.

See Also

[View > Visibility > Show Selected Only](#)

[View > Visibility > Hide All](#)

View > Visibility > Hide All

What

The **Hide All** command hides all entities in the Model View.



Why

The **Hide All** command is useful for:

- Quickly hiding all visible entities

How to Use

How to hide all entities

1. Go to **View > Visibility > Hide All** or click the icon in the toolbar.

See Also

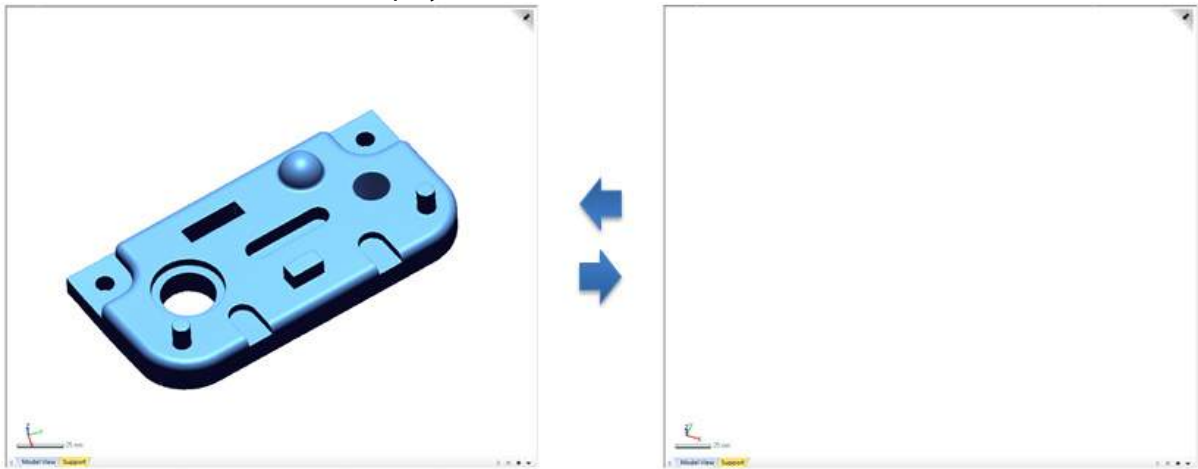
[View > Visibility > Show Selected Only](#)

[View > Visibility > Show All](#)

View > Visibility > Meshes

What

The **View Meshes** command displays or hides all meshes in the Model View.



Why

The **View Meshes** command is useful for:

- Viewing all meshes at one time
- Hiding all meshes at one time

How to Use

How to show or hide all meshes

1. Go to **View > Visibility > Meshes** or click the icon in the toolbar.

See Also

[View > Visibility > Show Selected Only](#)

[View > Visibility > Show All](#)

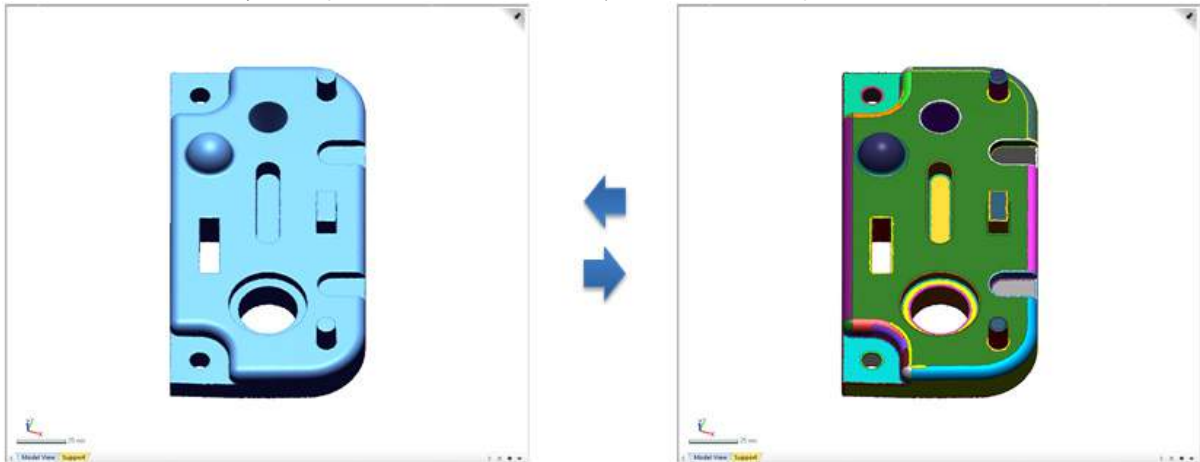
[View > Visibility > Hide All](#)

[View > Visibility > Regions](#)
[View > Visibility > Point Clouds](#)
[View > Visibility > Surface Bodies](#)
[View > Visibility > Solid Bodies](#)
[View > Visibility > Sketches](#)
[View > Visibility > 3D Sketches](#)

View > Visibility > Regions

What

The **View > Visibility > Regions** command displays or hides all regions in the Model view.



Why

The **View Regions** command is useful for:

- Viewing all regions at one time
- Hiding all regions at one time

How to Use

How to show or hide all regions

1. Go to **View > Visibility > Regions** or click the icon in the toolbar.

See Also

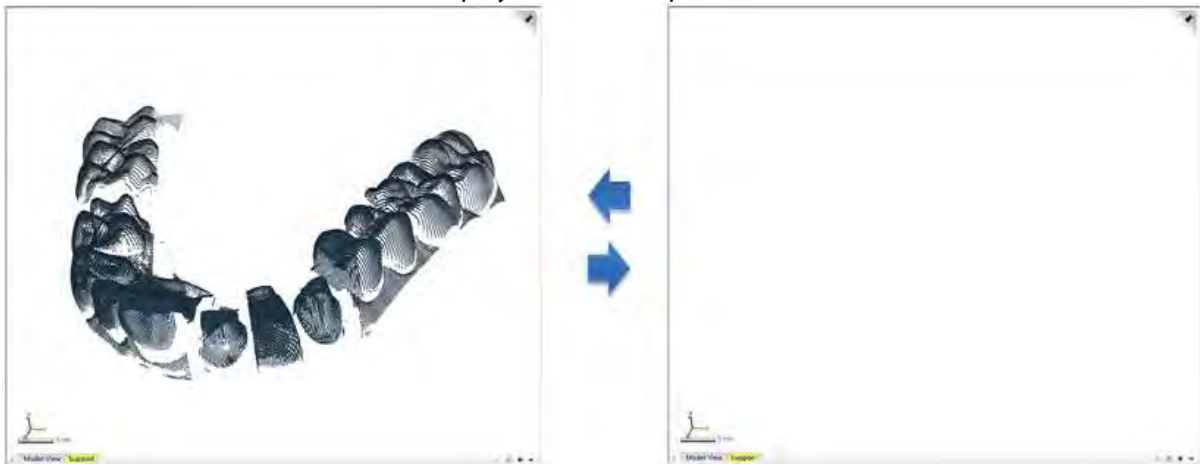
[View > Visibility > Show Selected Only](#)
[View > Visibility > Show All](#)
[View > Visibility > Hide All](#)

[View > Visibility > Meshes](#)
[View > Visibility > Point Clouds](#)
[View > Visibility > Surface Bodies](#)
[View > Visibility > Solid Bodies](#)
[View > Visibility > Sketches](#)
[View > Visibility > 3D Sketches](#)

View > Visibility > Point Clouds

What

The **View Point Clouds** command displays or hides all point clouds in the Model View.



Why

The **View Point Clouds** command is useful for:

- Viewing all point clouds at one time
- Hiding all point clouds at one time

How to Use

How show or hide all point clouds

1. Go to **View > Visibility > Point Clouds** or click the icon in the toolbar.

See Also

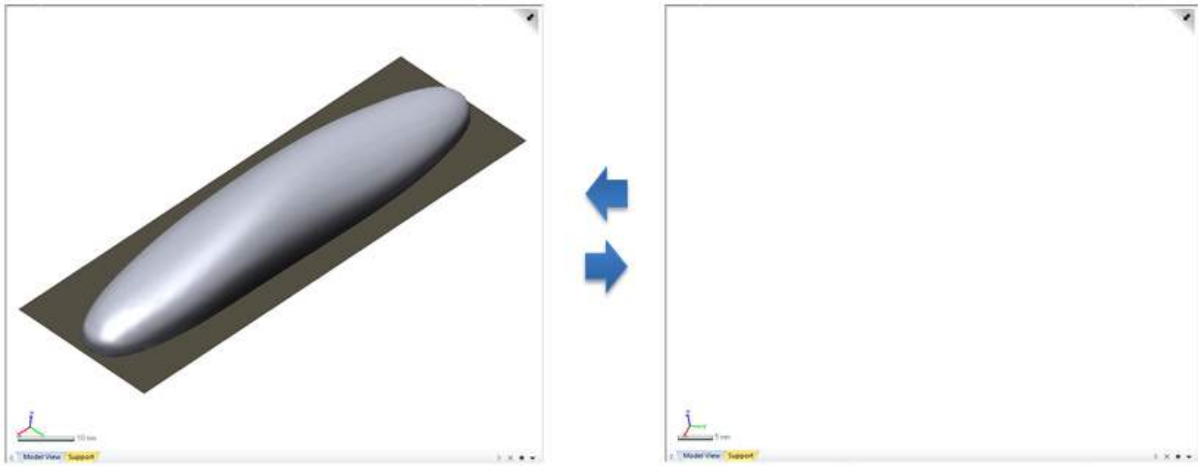
[View > Visibility > Show Selected Only](#)
[View > Visibility > Show All](#)
[View > Visibility > Hide All](#)

[View > Visibility > Meshes](#)
[View > Visibility > Regions](#)
[View > Visibility > Surface Bodies](#)
[View > Visibility > Solid Bodies](#)
[View > Visibility > Sketches](#)
[View > Visibility > 3D Sketches](#)

View > Visibility > Surface Bodies

What

The **View Surface Bodies** command displays or hides all Surface Bodies in the Model View.



Why

The **View Surface Bodies** command is useful for:

- Viewing all Surface Bodies clouds at one time
- Hiding all Surface Bodies at one time

How to Use

How show or hide all surface bodies

1. Go to **View > Visibility > Surface Bodies** or click the icon in the toolbar.

See Also

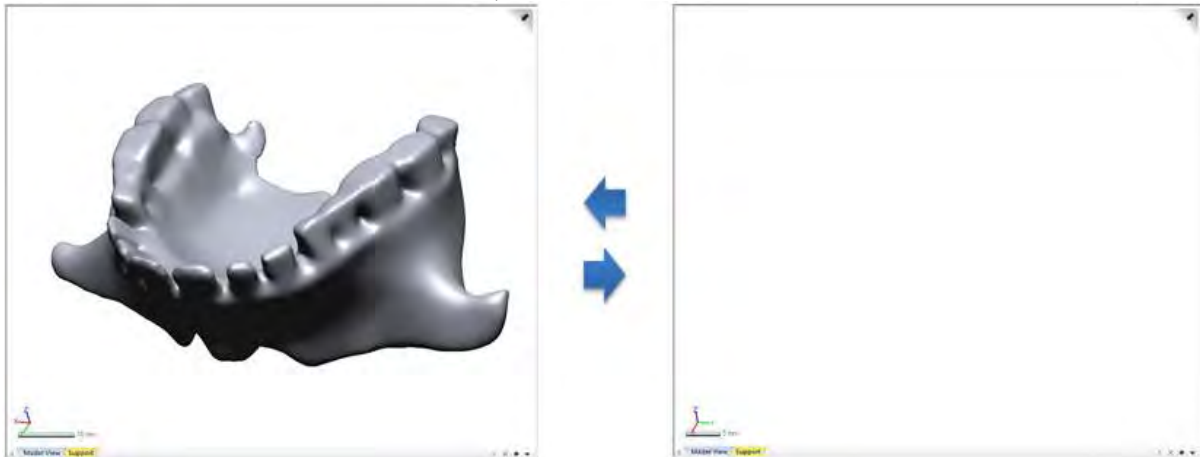
[View > Visibility > Show Selected Only](#)
[View > Visibility > Show All](#)
[View > Visibility > Hide All](#)
[View > Visibility > Meshes](#)

[View > Visibility > Regions](#)
[View > Visibility > Point Clouds](#)
[View > Visibility > Solid Bodies](#)
[View > Visibility > Sketches](#)
[View > Visibility > 3D Sketches](#)

View > Visibility > Solid Bodies

What

The **View Solid Bodies** command displays or hides all Solid Bodies in the Model View.



Why

The **View Solid Bodies** command is useful for:

- Viewing all Solid Bodies at one time
- Hiding all Solid Bodies at one time

How to Use

How show or hide all solid bodies

1. Go to **View > Visibility > Solid Bodies** or click the icon in the toolbar.

See Also

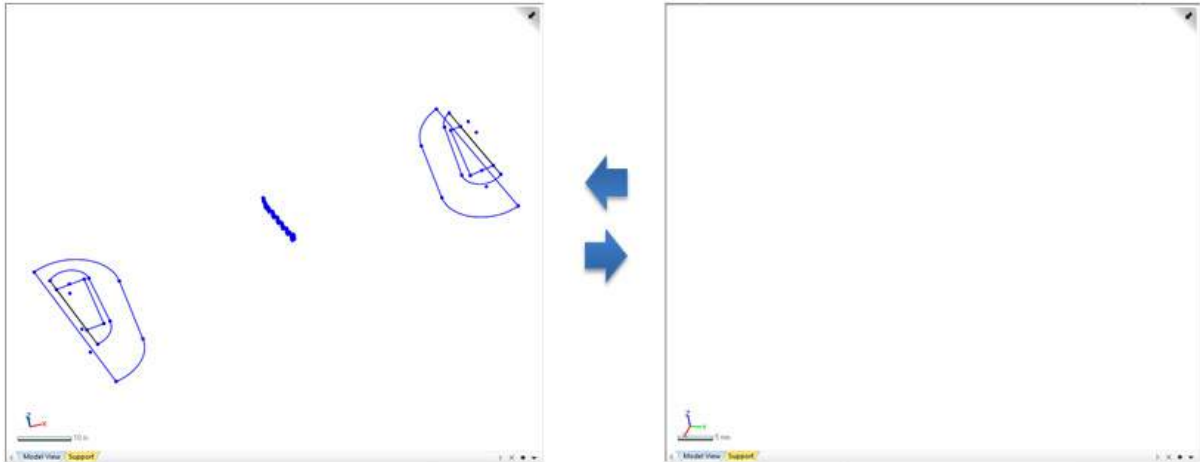
[View > Visibility > Show Selected Only](#)
[View > Visibility > Show All](#)
[View > Visibility > Hide All](#)
[View > Visibility > Meshes](#)
[View > Visibility > Regions](#)

[View > Visibility > Point Clouds](#)
[View > Visibility > Surface Bodies](#)
[View > Visibility > Sketches](#)
[View > Visibility > 3D Sketches](#)

View > Visibility > Sketches

What

The **View Sketches** command displays or hides all Sketches in the Model View.



Why

The **View Sketches** command is useful for:

- Viewing all Sketches at one time
- Hiding all Sketches at one time

How to Use

How show or hide all sketches

1. Go to **View > Visibility > Sketches** or click the icon in the toolbar.

See Also

[View > Visibility > Show Selected Only](#)
[View > Visibility > Show All](#)
[View > Visibility > Hide All](#)
[View > Visibility > Meshes](#)
[View > Visibility > Regions](#)
[View > Visibility > Point Clouds](#)

[View > Visibility > Surface Bodies](#)

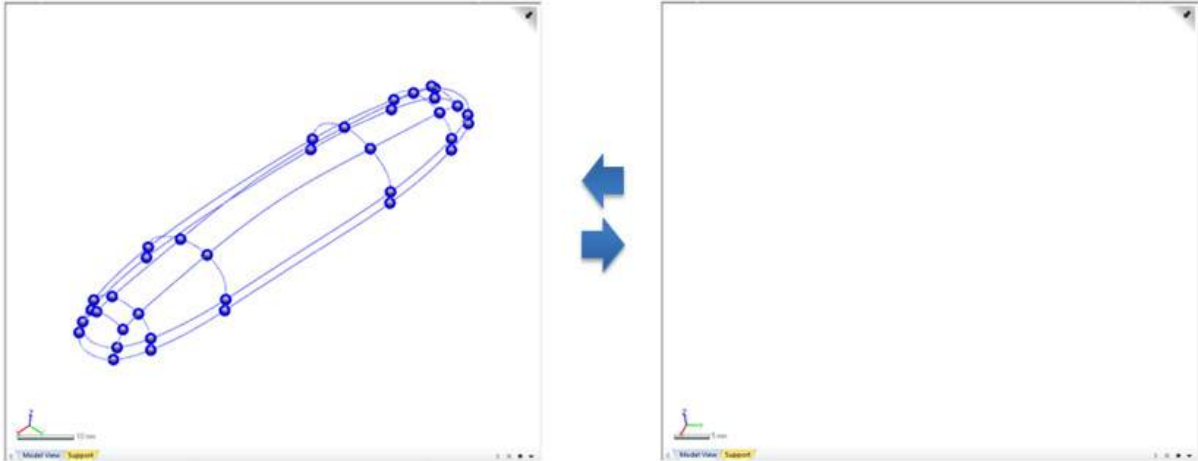
[View > Visibility > Solid Bodies](#)

[View > Visibility > 3D Sketches](#)

View > Visibility > 3D Sketches

What

The **View 3D Sketches** command displays or hides all 3D Sketches in the Model View.



Why

The **View 3D Sketches** command is useful for:

- Viewing all 3D Sketches at one time
- Hiding all 3D Sketches at one time

How to Use

How show or hide all 3D sketches

1. Go to **View > Visibility > 3D Sketches** or click the icon in the toolbar.

See Also

[View > Visibility > Show Selected Only](#)

[View > Visibility > Show All](#)

[View > Visibility > Hide All](#)

[View > Visibility > Meshes](#)

[View > Visibility > Regions](#)

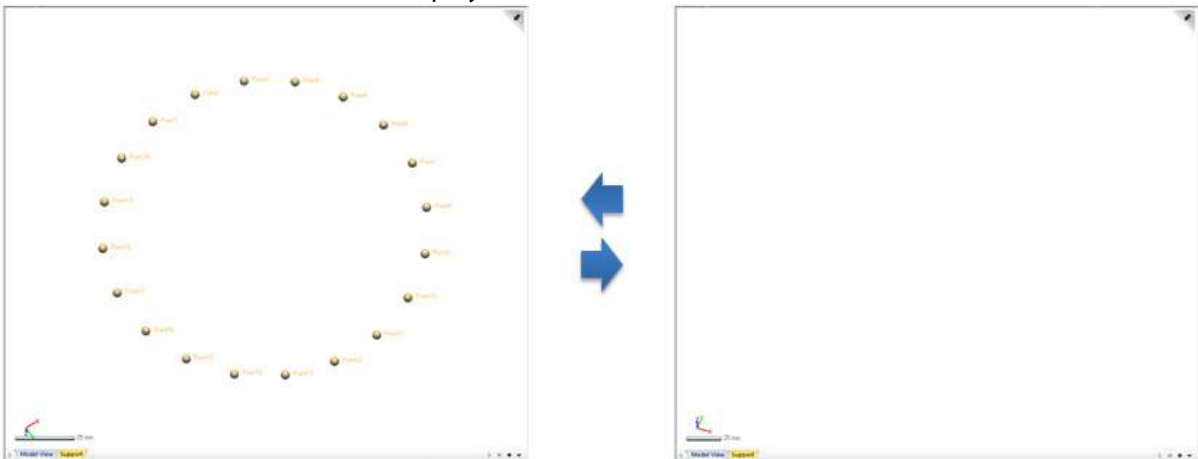
[View > Visibility > Point Clouds](#)

[View > Visibility > Surface Bodies](#)

View > Visibility > Ref. Points

What

The **View Ref. Points** command displays or hides all Ref. Points in the Model View.



Why

The **View Ref. Points** command is useful for:

- Viewing all Ref. Points at one time
- Hiding all Ref. Points at one time

How to Use

How show or hide all ref. points

1. Go to **View > Visibility > Ref. Points** or click the icon in the toolbar.

See Also

[View > Visibility > Show Selected Only](#)

[View > Visibility > Show All](#)

[View > Visibility > Hide All](#)

[View > Visibility > Ref.Vector](#)

[View > Visibility > Ref.Planes](#)

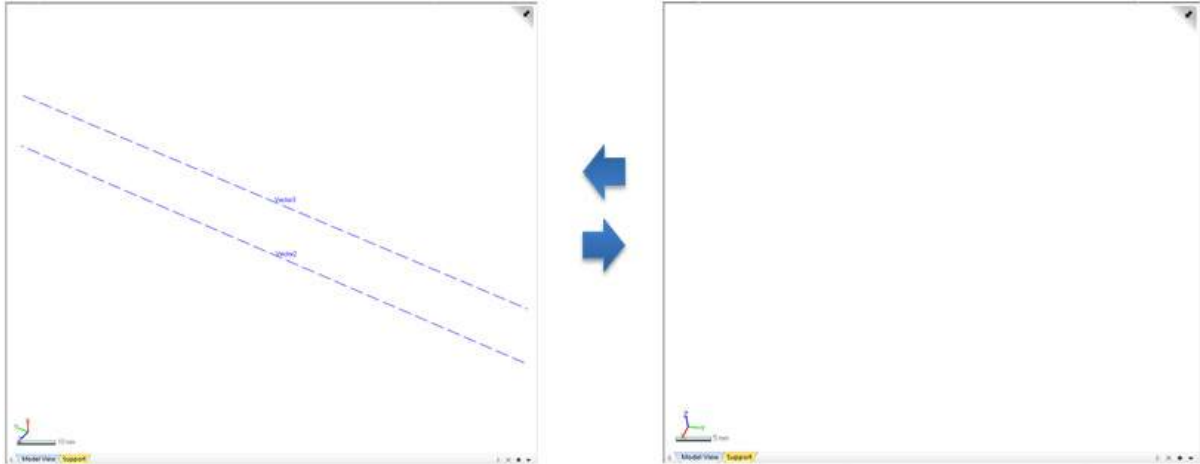
[View > Visibility > Ref.Polylines](#)

[View > Visibility > Ref.Coordinates](#)

View > Visibility > Ref. Vector

What

The **View Ref. Vector** command displays or hides all Ref. Vectors in the Model View.



Why

The **View Ref. Vector** command is useful for:

- Viewing all Ref. Vectors at one time
- Hiding all Ref. Vectors at one time

How to Use

How show or hide all ref. vectors

1. Go to **View > Visibility > Ref. Vector** or click on the icon in the toolbar.

See Also

[View > Visibility > Show Selected Only](#)

[View > Visibility > Show All](#)

[View > Visibility > Hide All](#)

[View > Visibility > Ref.Points](#)

[View > Visibility > Ref.Planes](#)

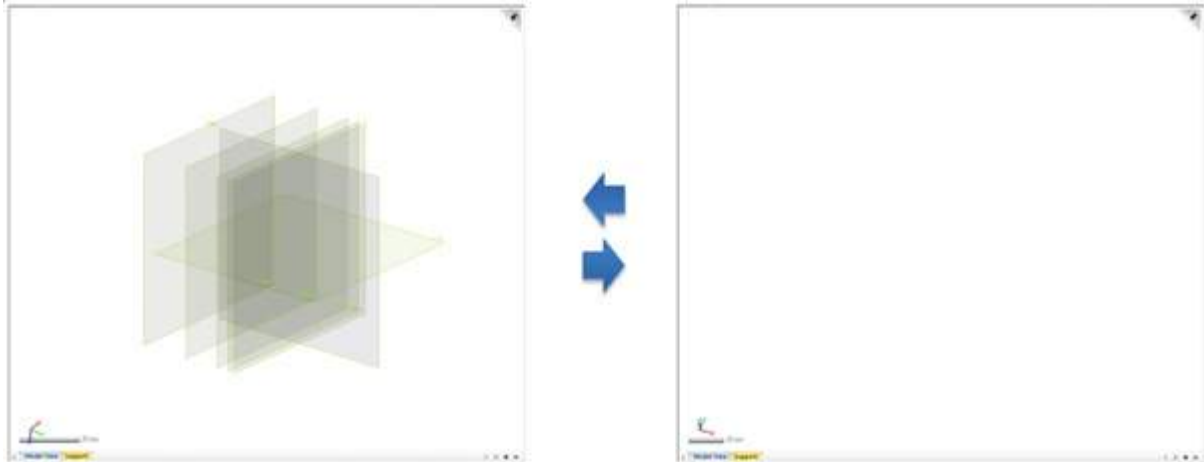
[View > Visibility > Ref.Polylines](#)

[View > Visibility > Ref.Coordinates](#)

View > Visibility > Ref. Planes

What

The **View Ref. Planes** command displays or hides all Ref. Planes in the Model View.



Why

The **View Ref. Planes** command is useful for:

- Viewing all Ref. Planes at one time
- Hiding all Ref. Planes at one time

How to Use

How show or hide all ref. planes

1. Go to **View > Visibility > Ref. Planes** or click on the icon in the toolbar.

See Also

[View > Visibility > Show Selected Only](#)

[View > Visibility > Show All](#)

[View > Visibility > Hide All](#)

[View > Visibility > Ref.Points](#)

[View > Visibility > Ref.Vector](#)

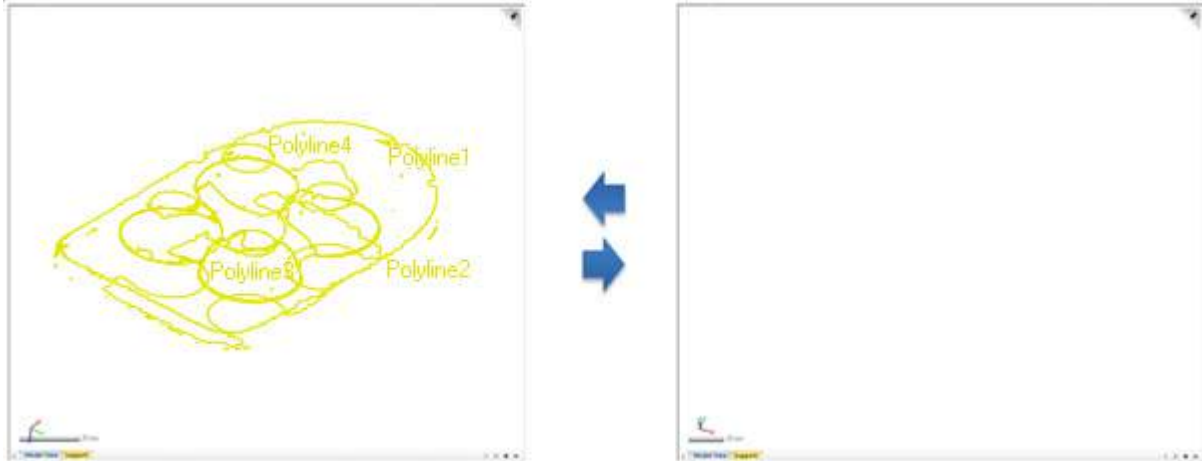
[View > Visibility > Ref.Polylines](#)

[View > Visibility > Ref.Coordinates](#)

View > Visibility > Ref. Polylines

What

The **View Ref. Polylines** command displays or hides all Ref. Polylines in the Model View.



Why

The **View Ref. Polylines** command is useful for:

- Viewing all Ref. Polylines at one time
- Hiding all Ref. Polylines at one time

How to Use

How show or hide all ref. polylines

1. Go to **View > Visibility > Ref. Polyline** or click on the icon in the toolbar.

See Also

[View > Visibility > Show Selected Only](#)

[View > Visibility > Show All](#)

[View > Visibility > Hide All](#)

[View > Visibility > Ref.Points](#)

[View > Visibility > Ref.Vector](#)

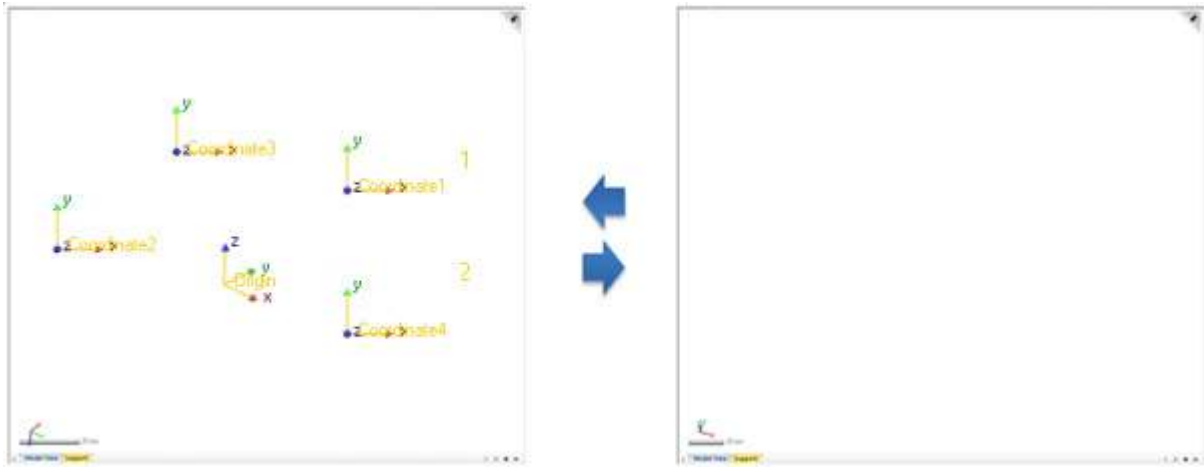
[View > Visibility > Ref.Planes](#)

[View > Visibility > Ref.Coordinates](#)

View > Visibility > Ref. Coordinates

What

The **View Ref. Coordinates** command displays or hides all Ref. Coordinates in the Model View.



Why

The **View Ref. Coordinates** command is useful for:

- Viewing all Ref. Coordinates at one time
- Hiding all Ref. Coordinates at one time

How to Use

How show or hide all ref. coordinates

1. Go to **View > Visibility > Ref. Coordinates** or click the icon in the toolbar.

See Also

[View > Visibility > Show Selected Only](#)

[View > Visibility > Show All](#)

[View > Visibility > Hide All](#)

[View > Visibility > Ref.Points](#)

[View > Visibility > Ref.Vector](#)

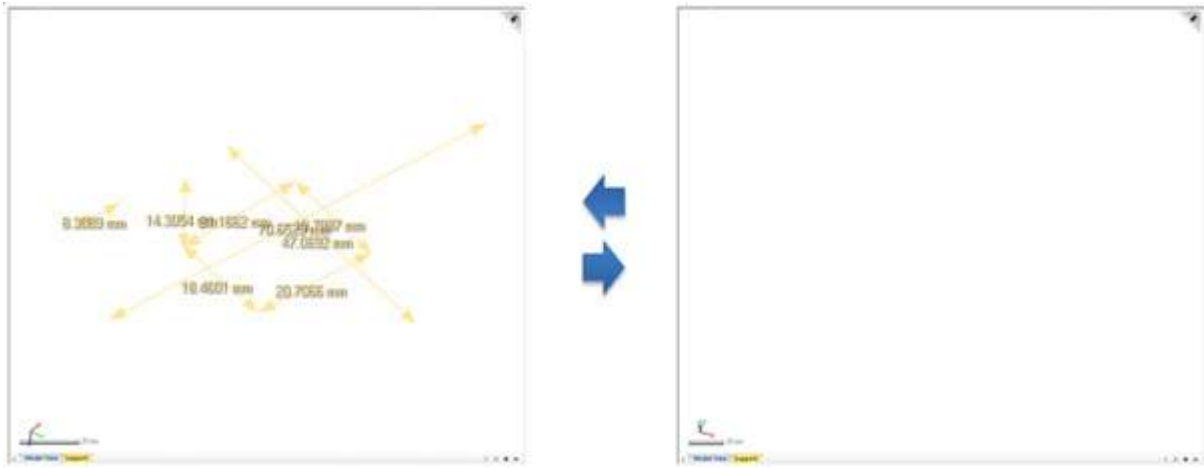
[View > Visibility > Ref.Planes](#)

[View > Visibility > Ref.Polylines](#)

View > Visibility > Measurements

What

The **Measurements** command displays or hides all measurement entities in the Model View.



Why

The **Measurements** command is useful for:

- Viewing all measurement entities at one time
- Hiding all measurement entities at one time

How to Use

How show or hide all measurements

1. Go to **View > Visibility > Measurements** or click the icon in the toolbar.

See Also

[View > Visibility > Show Selected Only](#)

[View > Visibility > Show All](#)

[View > Visibility > Hide All](#)

[View > Visibility > Ref.Points](#)

[View > Visibility > Ref.Vector](#)

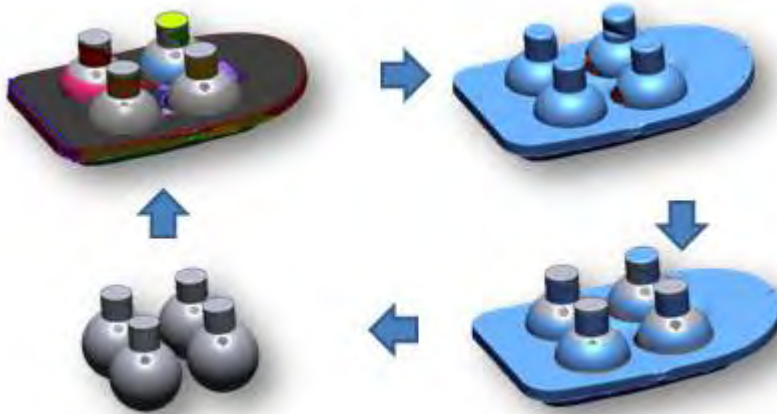
[View > Visibility > Ref.Planes](#)

[View > Visibility > Ref.Polylines](#)

View > Visibility > Smart Visibility

What

The **Smart Visibility** command intelligently offers a proper visibility method according to various modeling situations.



Within Mode

Smart Visibility will control the visibility of following entity types within each mode:

- Scan Data (Mesh / Point Cloud)
- Body (Solid / Surface)
- 2D Sketch
- 3D Sketch

Mode	Visibility	Remark
Default Mode	4 States: Current State / Scan Only / Scan Data + Body / Body Only	
Mesh Mode	3 States: Current State / Current Mesh Only / All Meshes	If poly-face does not exist in current mesh, Current Mesh Only state will be skipped.
Point Cloud Mode	2 States: Current State / Point Cloud Only	
Mesh Sketch Mode	4 States: Current State / 2D Sketch Only / Mesh + 2D Sketch / Body + 2D Sketch	Body for using Convert Entity command
2D Sketch Mode	4 States: Current State / 2D Sketch Only / Body + 2D Sketch / Mesh + 2D Sketch	Body for using Convert Entity command
3D Mesh Sketch Mode	2 States: Current State / Current Mesh + 3D Sketch	
3D Sketch Mode	4 States: Current State / Body + 3D Sketch / Mesh + Body + 3D Sketch / Mesh + 3D Sketch	

Note:

Region, Ref.Geometry, and Measurement entity types are excluded from this control, but Ref.Plane, Ref.Polyline, Ref.Coordinate and Measurement entity types can be controlled once by Smart Visibility in Mesh Sketch mode and 2D Sketch mode.

If you manually change the visibility as a specific entity type by using the Visibility tools other than using the Smart Visibility tool, then the Current State will be updated as the modified state and the visibility circulation will be restarted by the Smart Visibility tool. Plus, if the Current State is identical with any pre-

defined state, the Current State will be skipped. This rule will also be applied within modeling command and Modeling Wizard.

Within Modeling Command and Modeling Wizard

Smart Visibility will control the visibility of following entity types within each Modeling command and Modeling Wizard:

- Scan Data (Mesh / Point Cloud)
- Region (including Mesh Visibility)
- Body (Solid / Surface)
- 2D Sketch
- 3D Sketch

Mode	Visibility	Remark
Extrude, Revolve, Sweep, Loft	4 States: Current State / Preview Only / Scan Data + Preview / Body + Preview	
Primitives	2 States: Current State / Region Only or (In Preview stage) Region + Preview / Preview Only	Primitives only work with Regions.
Modeling Wizards	3 States (In 1st Stage): Current State / Region Only / Body + Region	
	3 States (In 2nd Stage): Region + Preview / Region + Body + Preview / Body + Preview	

Note:

The other commands not described in above will follow the behavior of Default Mode.

Why

The **Smart Visibility** command is useful for:

- Quickly hide other multiple entities
- Quickly show only desired entities in current modeling status

How to Use

How to use Smart Visibility

1. Go to **View > Visibility > Smart Visibility** or click the icon in the toolbar.
2. Check changes.
3. Go to **View > Visibility > Smart Visibility** or click the icon in the toolbar again to change

the visibility.

See Also

[View > Visibility > Show Selected Only](#)

[View > Visibility > Show All](#)

[View > Visibility > Hide All](#)

[View > Visibility > Ref.Points](#)

[View > Visibility > Ref.Vector](#)

[View > Visibility > Ref.Planes](#)

[View > Visibility > Ref.Polylines](#)

7.7.2. View > Zoom

The **Zoom** menu features methods to control zooming entities on Model View.

The **Zoom** menu is divided into 2 tool groups used to:

Zoom View

- **Area**
Zoom in on a specific area
- **Previous Area**
Display the previous area
- **Fit**
Fit all entities into the model view

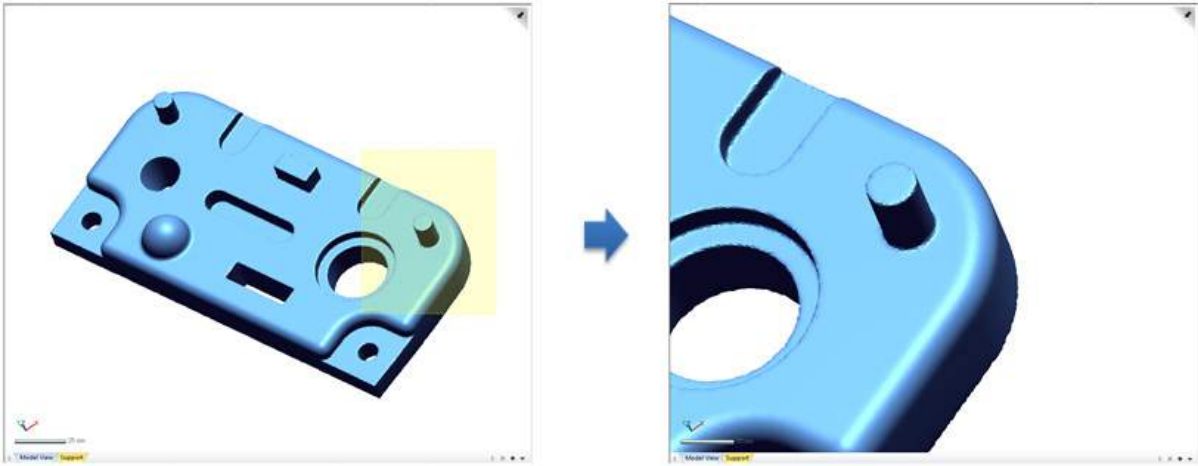
Create Picture-In-Picture

- **PIP**
Create a zoomed view within the model view

View > Zoom > Area

What

The **Zoom Area** command zooms into a manually selected rectangular area.



Why

The **Zoom Area** command is useful for:

- Zooming in on specific areas of a model

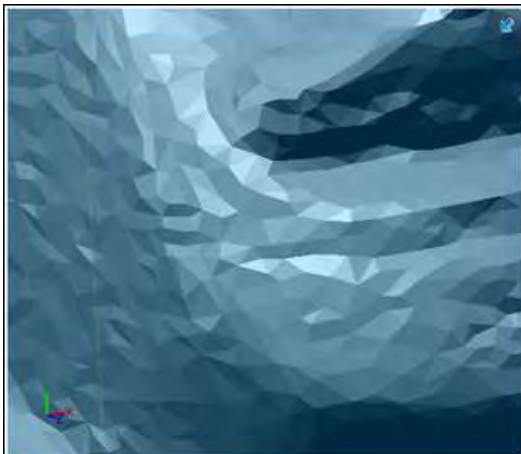
How to Use

How to zoom in on an area

1. Go to **View > Zoom > Area** or use the **Shift+Z** keyboard shortcut.
2. Drag-select a rectangular region over the model.

Note:

The **Zoom Area** command is active until another command runs or the command is ran again. To go back to a previous viewing area use the **View > Zoom > Previous Area** command.



See Also

[View > Zoom > Previous Area](#)

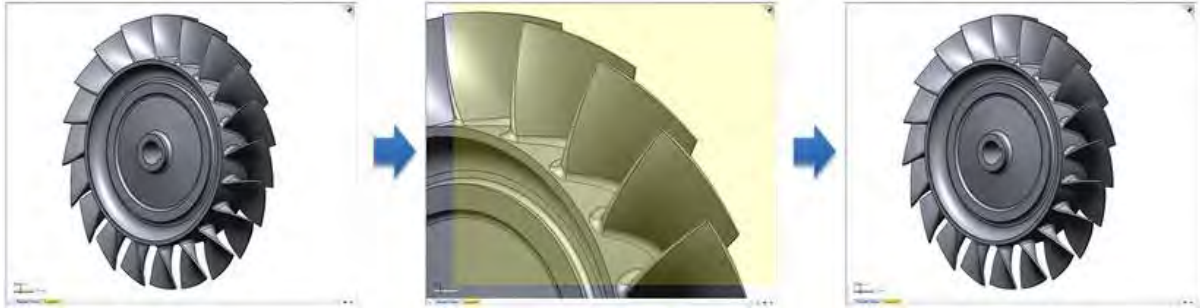
[View > Zoom > Fit](#)

[View > Zoom > PIP](#)

View > Zoom > Previous Area

What

The **Zoom Previous Area** command restores the viewing window to the previous view.



Why

The **Zoom Previous Area** command is useful for:

- Zooming to a previous view

How to Use

How to zoom to a previous area

1. Go to **View > Zoom > Previous Area** or click the icon in the toolbar. The **Shift + X** keyboard shortcut can also be used.

Note:

The history of the **Zoom Previous Area** command is stacked when the **Zoom Area** command is executed: however it is unavailable until the **Zoom Area** command has been executed first.

See Also

[View > Zoom > Area](#)

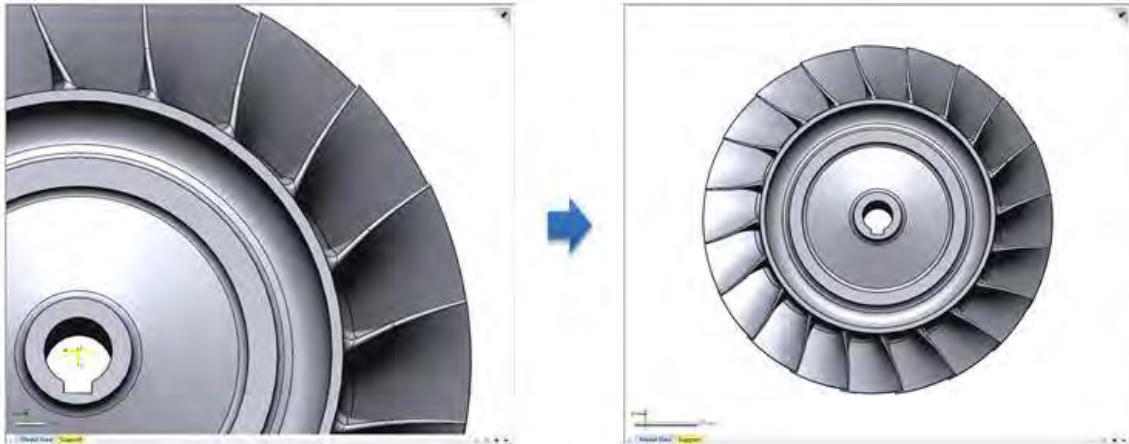
[View > Zoom > Fit](#)

[View > Zoom > PIP](#)

View > Zoom > Fit

What

The **Zoom Fit** command fits a model into the viewing window.



Why

The **Zoom Fit** command is useful for:

- Zooming a model to fit to the viewing window

How to Use

How to fit a model to the model view

1. Go to **View > Zoom > Fit** or click the icon in the toolbar. The **Ctrl + F** keyboard shortcut can also be used.

Note:

The history of the **Zoom Previous Area** command is stacked when the **Zoom Area** command is executed: however it is unavailable until the **Zoom Area** command has been executed first.

See Also

[View > Zoom > Area](#)

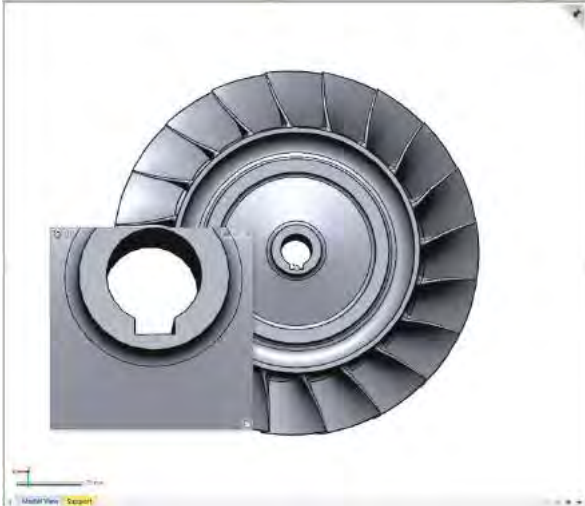
[View > Zoom > Previous Area](#)

[View > Zoom > PIP](#)

View > Zoom > PIP

What

The **Zoom PIP** command creates a pop-up window that zooms onto a specific position.



Event	Short Cut / Action
Window Toggle	Ctrl + W
Close Window	Esc
Move	Ctrl + Left Mouse Button
Window Size	Ctrl + Mouse Wheel
Magnification	Mouse Wheel

	Fix Center of Window
	Compensate Center (Click and drag the position that you want to set the center of the window)
	Close Window
	Edit Window Size

Why

The **Zoom Fit** command is useful for:

- Zooming into precise areas without zooming the entire entity

How to Use

How to zoom using the PIP function

1. Go to **View > Zoom > PIP** or click the icon in the toolbar. The **Ctrl + W** keyboard shortcut can also be used.

See Also

[View > Zoom > Area](#)

[View > Zoom > Previous Area](#)










[View > Zoom > Fit](#)

7.7.3. View > Mesh Display Mode

What

The **Mesh Display Mode** sets the display mode of a mesh by utilizing the following mode types:

- Point Set – Draws only poly vertices
- Wireframe – Draws only the edges of faces
- Shaded – Draws shaded faces
- Shaded With Edges – Draws shaded faces with edges
- Texture – Draws textures
- Curvature – Draws curvature flow on the mesh
- Region – Draws regions
- Geometry Type – Draws each geometry type (such as planes, spheres, revolutions, and freeforms, etc) in the same color
- Transparent – Draws transparent mesh

	Point Set	F5
	Wireframe	F6
	Shaded	F7
	Shaded With Edges	F8
	Texture	
	Curvature	
	Region	F9
	Geometry Type	
	Transparent	



Point Set: F5



Wireframe: F6



Shaded: F7



Shaded With Edges: F8



Texture



Curvature



Region: F9



Geometry Type



Transparent

Why

The **Mesh Display Mode** is useful for:

- Displaying mesh using one of the available display filters

How to Use

How to select a mesh display mode

1. Go to **View > Mesh Display Mode > Point Set / Wireframe / Shaded / Shaded With Edge** or click the icon in the toolbar.

See Also

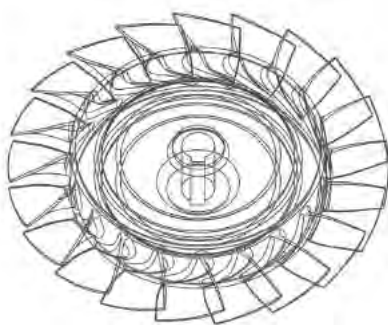
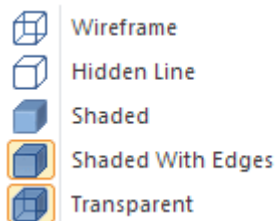
[Docking Bars > Display](#)

7.7.4. View > Body Display Mode

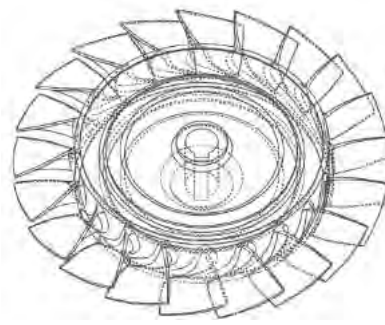
What

The **Body Display Mode** sets the display mode of a surface or solid body by utilizing the following mode types:

- Wireframe – Draws only the edges of bodies
- Hidden Line – Draws all hidden lines on a body as dotted lines
- Shaded – Draws all shaded faces
- Shaded With Edges – Draws shaded faces with edges
- Transparent – Draws transparent bodies



Wireframe



Hidden Line



Shaded



Shaded With Edges



Transparent

Why

The **Body Display Mode** is useful for:

- Displaying bodies using one of the available display filters

How to Use

How to select a body display mode

1. Go to **View > Body Display Mode > Wireframe / Hidden Line / Shaded / Shaded With Edge** or click the icon in the toolbar.








See Also




[Docking Bars > Display](#)


7.7.5. View > Viewpoint

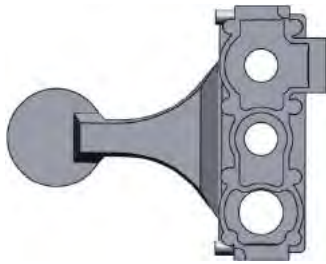
What

The **Viewpoint** commands set the view direction as one of seven pre-defined views and can also be used to rotate, flip and align a model's view.

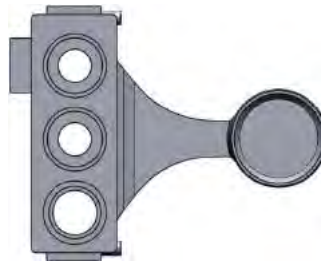
	Front	Alt+1
	Back	Alt+2
	Left	Alt+3
	Right	Alt+4
	Top	Alt+5
	Bottom	Alt+6
	Isometric	Alt+7

	Rotate CCW	
	Rotate CW	
	Flip	

	Align View	Ctrl+Shift+A



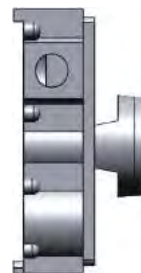
Front



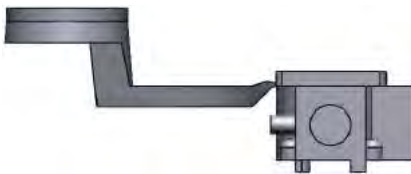
Back



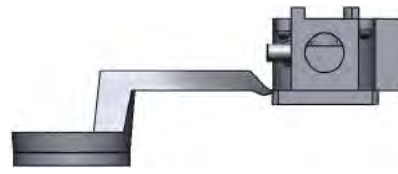
Left



Right



Top



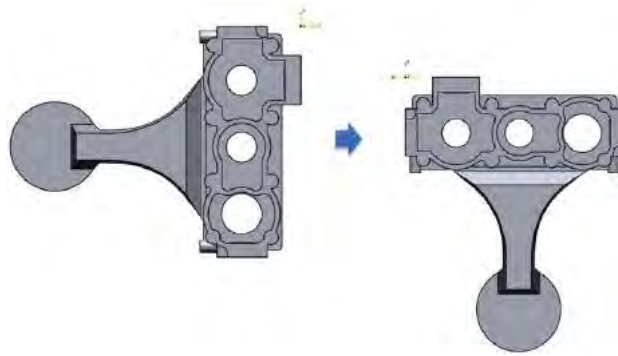
Bottom



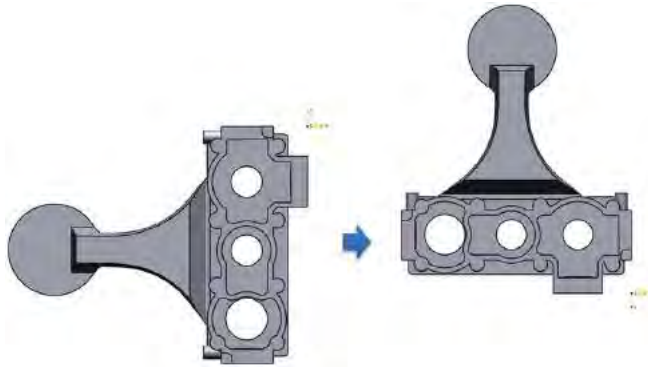
Isometric



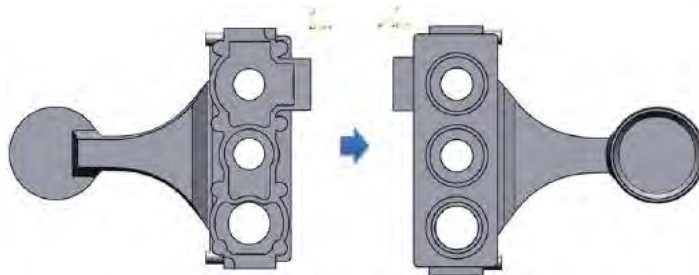
Bottom



Rotate CCW



Rotate CW



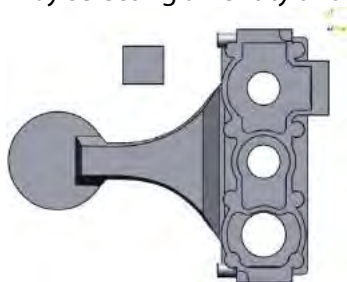
Flip

Align View

Sets the view direction in the **Model View** by selecting an entity and aligning to it.



Before Align View



After Align View

Note:

The Align View function can be reversed by holding the **Ctrl** button on the keyboard while using the function.

Why

The **Viewpoint** command is useful for:

- Setting the viewpoint of a model in the model view

How to Use

How to select a viewpoint

1. Go to **View > Viewpoint > Front / Back / Left / Right / Top / Bottom / Isometric** or click the icon in the toolbar.

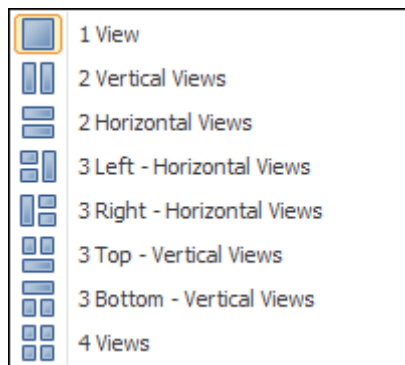
See Also

[Docking Bars > Display](#)

7.7.6. View > Viewport

What

The **Viewpoint** commands set the viewing window a pre-set view layout, each having its own properties.



1 View – Displays one view.



2 Vertical Views – Displays the view into two vertical panes.



2 Horizontal Views – Displays the view into two horizontal panes.



3 Left – Horizontal Views – Displays the view into three panes: two split panes on the left side and one full horizontal pane on the right side.



3 Right – Horizontal Views – Displays the view into three panes: two split panes on the right side and one full horizontal pane on the left side.



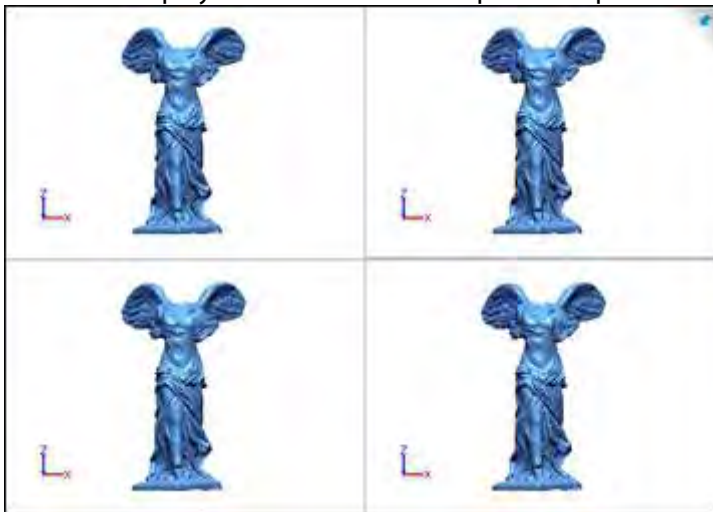
3 Top–Vertical Views – Displays the view into three panes: two split panes on the top half and one full horizontal pane on the bottom half.



3 Bottom–Vertical Views – Displays the view into three panes: two split panes on the bottom half and one full horizontal pane on the top half.



4 Views - Displays the view into four equal sized panes.



Why

The **Viewport** command is useful for:

- Setting the viewport window mode in the model view

How to Use

How to select a viewport

1. Go to **View > Viewport > 1 View / 2 Vertical Views / 2 Horizontal Views / 3 Left – Horizontal Views / 3 Right – Horizontal Views / 3 Top – Vertical Views / 3 Bottom – Vertical Views / 4 Views** or click the icon in the toolbar.

7.7.7. View > Light Properties

What

The **Light Properties** command enables changing of the direction or the color of a light source.



Why

The **Light Properties** command is useful for:

- Customizing the lighting conditions of the viewing area

How to Use

How to set light properties

1. Go to **View > Light Properties** or click on the icon in the toolbar.



2. Drag light sources to desired positions in the model view. The shadow will change while the light source is altered.



3. Click **OK** to complete the command.

Options in Detail

Methods

Light – Sets the color of each light source.

Light 1 – Sets the color of light 1.

Light 2 – Sets the color of light 2.

Light 3 – Sets the color of light 3.

Light 4 – Sets the color of light 4.

Reset – Resets all light settings back to default.

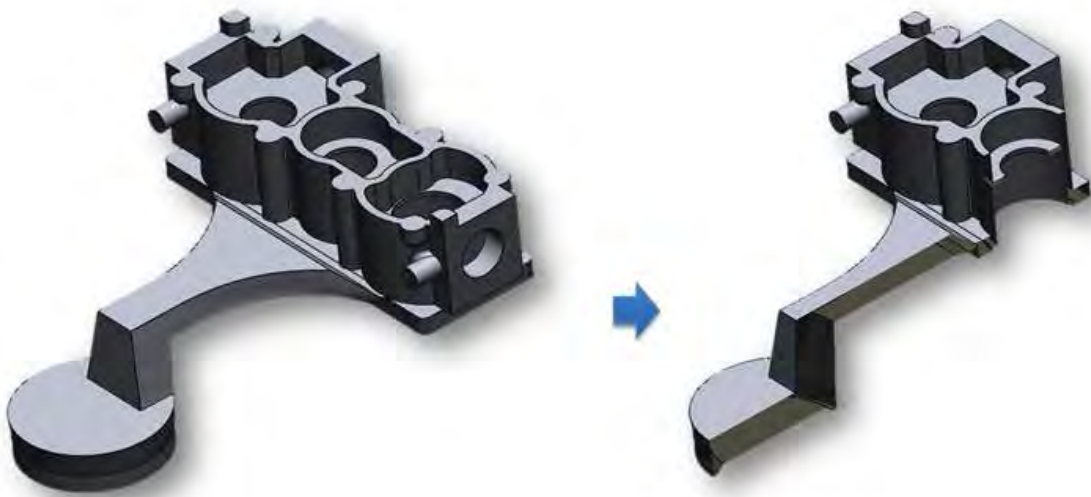
See Also

[Docking Bars > Display](#)

7.7.8. View > View Clip

What

The **View Clip** command enables viewing of a model by clipping away a part of it.



Why

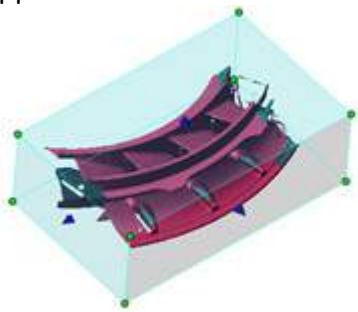
The **View Clip** command is useful for:

- Clipping away an area of a model to inspect it

How to Use

How to clip a view

1. Go to **View > View Clip** or click the icon in the toolbar.
2. Select the **Inside Box** option and a bounding box used to select the clipping area will appear.



3. The dimensions of the box can be adjusted by dragging the blue arrows or the green points.



4. Click **OK** to complete the command.

Options in Detail

Methods

Use View Clip – Toggles the view clip on or off.

Below Plane – Defines a cutting plane and clips all model data above it. The **Position** and **Normal Direction** options of the plane can be defined unless the **View Align** option is used, in which case the normal of the plane aligns to the screen direction.

Between Two Planes – Defines two cutting planes and clips all model data outside of the two planes. The **Position** or **Normal Direction** options of the plane can be defined unless the **View Align** option is used, in which case the normal of the plane aligns to the screen direction. The **Depth** option can be used to set the distance between two preview planes.

Inside Box – Defines a cutting box and clips all model data outside of the box. The box can be resized in the Model View.

Reset – Resets view clipping to default.

Plane Options

Position – Sets the position of the view clip plane.

X – Sets the X position.

Y – Sets the Y position.

Z – Sets the Z position.

Normal Direction – Sets the direction of the normal.

X – Sets the X position.

Y – Sets the Y position.

Z – Sets the Z position.

View Align – Aligns the normal direction to the global coordinates.

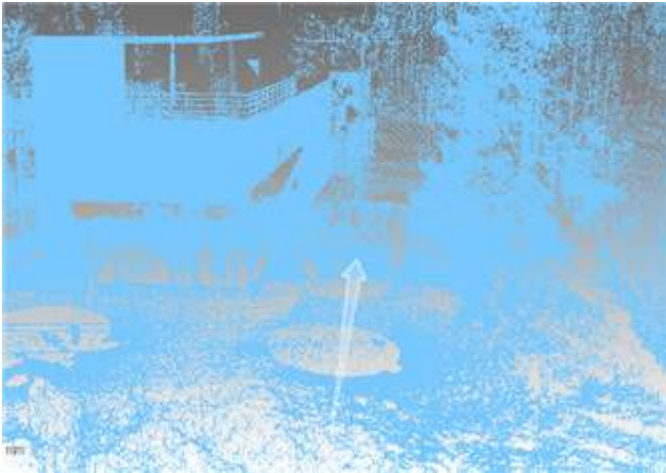
See Also

[Docking Bars > Display](#)

7.7.9. View > Walk Through

What

The **Walk-through** command enables the free movement of the viewpoint inside a model.



Why

The **Walk-through** command is useful for:

- Moving a virtual camera through a model
- Viewing the inside of a complex model like a scan of a building

How to Use

How to walk-through a model




1.

Note:

The **Walk-through** command will be dimmed if the current viewpoint is at the top or the bottom of a model. To enable the function the model must be rotated to a side view.

Upon executing the **Walk-through** command the **Projection Method** setting in the **Display** tab will automatically be changed to **Perspective**.

The **Point Shader** option can be set to the **Depth** method which hides points away from the viewpoint.

2. Controlling the camera is identical to controlling the **Pan** function in the default viewing mode and can be activated by using the **Ctrl + right mouse button** (or left and right mouse button together) in the selection mode . Alternatively, the **Ctrl + Left, Right, Up, Down** shortcut keys can be used.
3. Controlling the shift is identical to controlling the **Zoom** function in the default viewing mode and can be activated by using **Shift + right mouse button** in the selection mode . Alternatively, the **Shift + Up, Down** shortcut keys can be used.
4. Rotating the camera is identical to the **Rotate** function in the default viewing mode and can be activated by using the **right mouse button** in the selection mode . Alternatively, the **Left, Right, Up, and Down** keys on the keyboard can be used.

7.8. Add-Ins

The **Add-Ins** menu has various functions that are used to handle point clouds and polygons to generate complete polygon data.

The **Add-Ins** menu is divided into 2 tool groups used to:

Import and Register Scan Data

- **Import and Split**
Split scan data by the number of vertices or number of meshes while importing ASC, STL, OBJ, and PLY file formats
- **IGES Point Importer**
Import point cloud data formatted in the IGES file format
- **Register Target**
Align target scan data files automatically using alignment spheres

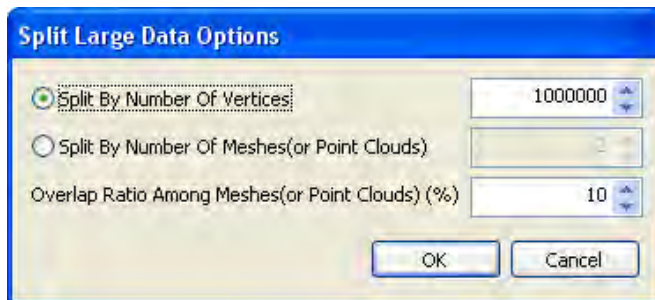
Modify Scan Data

- **Delete Poly-Faces**
Quickly remove poly-faces without entering the Mesh mode
- **Trim Mesh**
Cut off regions surrounded by reference curves on a mesh or align poly-edges along reference curves
- **Flatten/Deform**
Flatten and deform mesh to generate a new mesh in a desired shape

7.8.1. Add-Ins > Import & Split

What

The **Import & Split** command splits scan data by the number of vertices or number of meshes while importing ASC, STL, OBJ, and PLY file formats. If data files are too large to import at one time this command features the Suppress option in the Import dialog box which separates the file and loads them onto the hard disk drive one at a time.



Why

The **Import & Split** command is useful for:

- Importing large scan files

Options in Detail

Methods

Split By Number Of Vertices – Splits a file based on number of vertices.

Split By Number Of Meshes (or Point Clouds) – Splits a file based on number of meshes or point clouds.

Overlap Ratio Among Meshes (or Point Clouds) (%) – Sets the overlap ratio between meshes or point clouds.



See Also

[Insert > Import](#)

7.8.2. Add-Ins > IGES Point Importer

What

The **IGES Point Importer** command imports point data formatted in the IGES file format.

Why

The **IGES Point Importer** command is useful for:

- Importing IGES files
- Importing large IGES files instead of using the Insert > Import command

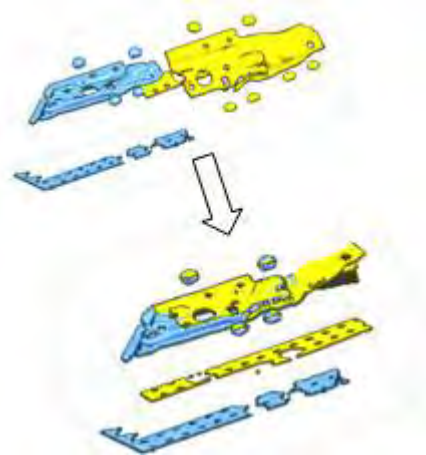
See Also

[Insert > Import](#)

7.8.3. Add-Ins > Register Target

What

The **Register Target** command roughly aligns meshes by matching target alignment spheres.



Why

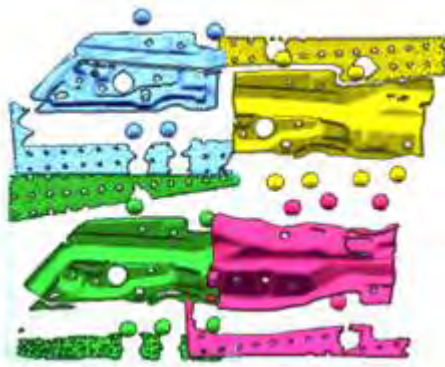
The **Register Target** command is useful for:

- Combining multiple scans that are too large for one scan
- Combining multiple scans that require the use of target alignment spheres

How to Use

How to align multiple scans by matching target alignment spheres

1. Go to **Add-Ins > Register Target** or click on the icon in the toolbar.
2. Select all meshes as **Meshes** targets.



3. Click the **Magic Wand** button to automatically get the ball size. When scan data is rough, the tolerance value can be increased. When scan data is smooth, the tolerance value can be decreased to help accurately find spheres.
4. Click the **OK** button to complete the command.



Options in Detail

Methods

Find Target Automatically – Automatically finds target spheres on scan data. This option is available only for meshes.

Set Target Manually – Directly defines targets by picking a seed point on a spherical area and registering each scan shot by matching extracted targets.

Selection Methods

Scan Data – Select meshes or point clouds.

Shared Scan Folder – Selects a folder to align files from.

Select Share Folder – Chooses a designated shared scan folder.

Ball Radius

Acceptable Tolerance – Specifies the allowable tolerance for the radius of target spheres.

Min. Size Of Cluster – Specifies the minimum number of poly-faces/vertices of a target sphere.

Max. Size Of Cluster – Specifies the maximum number of poly-faces/vertices of a target sphere.

More Options

Ignore Spike Faces – Ignores spiky faces on targets.

Use Curvature – Separates regions by curvature.

Note:

Target spheres must be disconnected from the rest of the scan to be found automatically, or manually located.

See Also

[Tools > Scan Tools > Align Between Scan Data](#)

7.8.4. Add-Ins > Delete Poly-Faces

What

The **Delete Poly-Faces** command quickly removes poly-faces without entering the **Mesh mode**.



Why

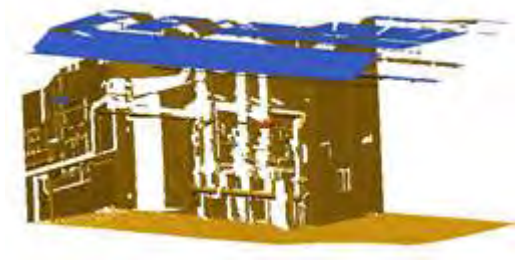
The **Delete Poly-Faces** command is useful for:

- Deleting individual irregular poly-faces from multiple meshes at a time
- Deleting extraneous poly-faces from scan data

How to Use

How to delete poly-faces from scan data

1. Go to **Add-Ins > Delete Poly-Faces** or click the icon in the toolbar.



2. Select poly-faces to remove.



3. Click the **Delete Selected Entities** button or press the **Delete** key.



4. Click the **OK** button to complete the command.

Options in Detail

Deleting

Select Poly-Faces – Selects poly-faces to delete.

See Also

[Edit > Delete](#)

7.8.5. Add-Ins > Trim Mesh

What

The **Trim Mesh** command cuts off regions surrounded by reference curves on a mesh or aligns poly-edges along reference curves. This function functions similarly as the **Tools > Mesh Tools > Trim** command, but can trim a mesh without entering the **Mesh mode**.



Why

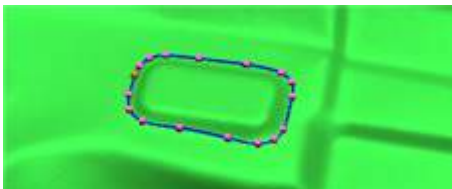
The **Trim Mesh** command is useful for:

- Deleting unwanted areas of mesh
- Deleting extraneous mesh from scan data
- Using child curve features to cut the mesh

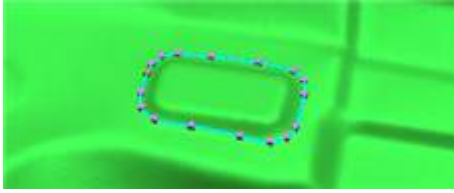
How to Use

A. How to trim meshes from scan data using the By Curve method

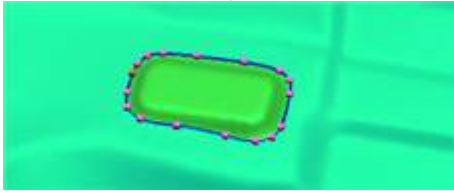
1. Go to **Add-Ins > Trim Mesh** or click the icon in the toolbar.



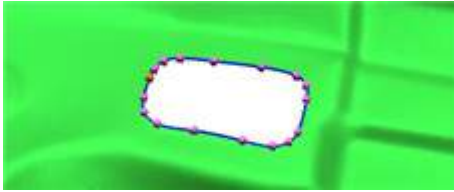
2. Select the **By Curve** method. Choose a **Mesh** and **Curves**. Set the **Method** to **Split Poly-Faces**.



3. Click the **Next Stage** button.

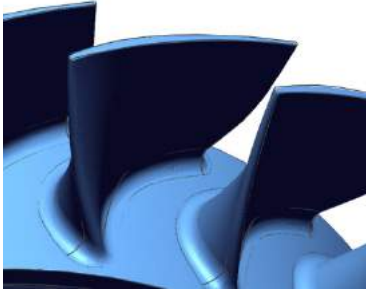


4. Click the **OK** button to complete the command.

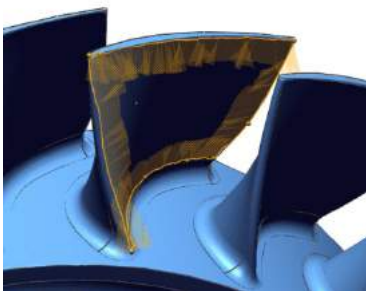


B. How to trim meshes from scan data using the By Custom Path method

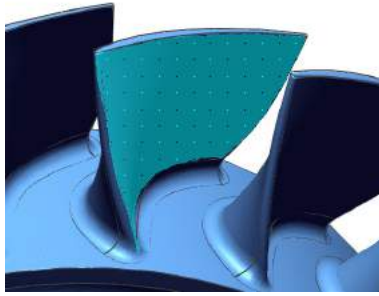
1. Go to **Add-Ins > Trim Mesh** or click the icon in the toolbar.



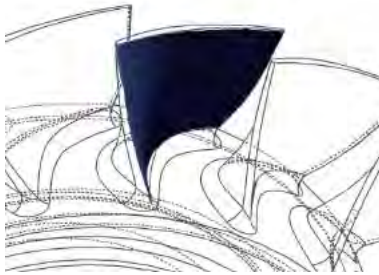
2. Select the **By Custom Path** method. Created the desired path.



3. Click the **Next Stage** button.



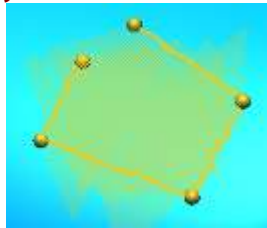
4. Select the remaining **Region** and click the **OK** button to complete the command.



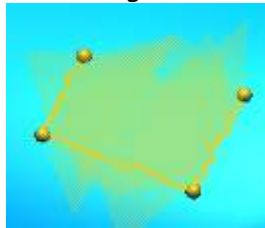
Options in Detail

Methods

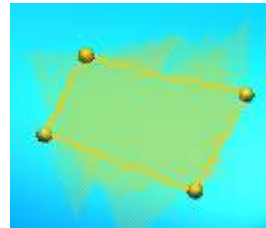
By Custom Path – Trims a mesh using a custom path.



Drawing Custom Path



Undo Path



Close Path

By Curve – You can trim a mesh with existing close curves.

Trim Path – Undo, close and discard paths.

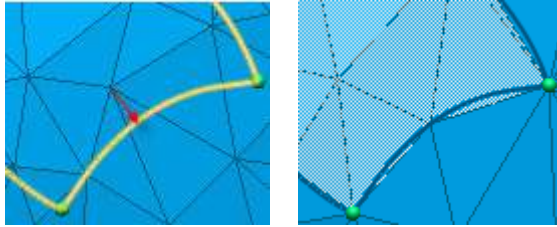
Curve Options

Mesh – Selects mesh.

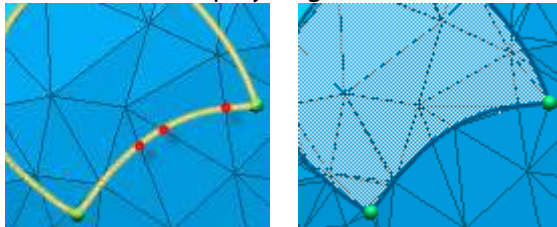
Curves – Selects Curves.

Method – Selects the method.

Rearrange Poly-Vertices – Trims by projecting nearby poly-vertices onto selected curves.



Split Poly-Faces – Trims by dividing poly-faces with selected curves at intersecting position between poly-edges and curves.



Max. Curve To Poly-Vertex Distance – Sets the maximum allowable distance from a curve to a poly vertex.

Max. Edge Splitting Ratio – Sets the maximum allowable splitting ratio of edges.

Remaining Region

Remaining Region – Selects what regions will remain.

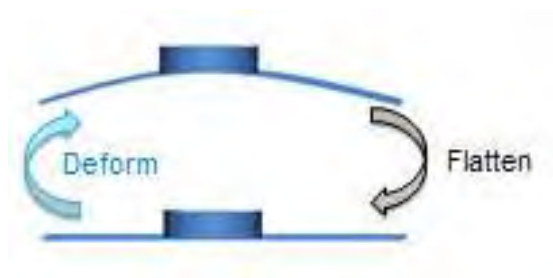
See Also

[Tools > Mesh Tools > Trim](#)

7.8.6. Add-Ins > Flatten / Deform

What

The **Flatten / Deform** command flattens and deforms mesh to generate a new mesh in a desired shape.



Why

The **Flatten / Deform** command is useful for:

- Flattening damaged mesh
- Flattening high curvature areas
- Flattening a bent shape
- Flattening to preserve a shape
- Deforming to create a desired shape

How to Use

How to flatten a mesh

1. Go to **Add-Ins > Flatten And Deform** or click on the icon in the toolbar.
2. Choose the **Flatten** method and select a mesh as the **Input Mesh**.



ISO View



Side View

3. Choose the **Select Seed Poly Faces** option under **Driving Surface**.
4. Click the **Select Poly-Faces** button and select a point on an area of curvature on the mesh.



5. Click the **Extend** button to automatically select the entire bent area.



6. Set the flatten direction. For this example the XY direction is used.



7. Click **OK** to complete the command.



Before

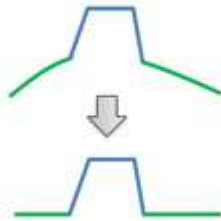


After

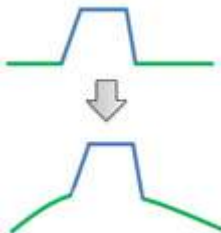
Options in Detail

Methods

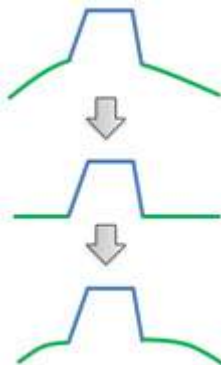
Flatten – Flattens a bent area while preserving features.



Deform – Deforms a flat area to a desired shape while preserving features.



Flexible Registration – First flattens then deforms a target area while preserving features.



Input Mesh

Select Mesh – Selects a mesh

Driving Surface

Select Initial Template – Chooses the Select Initial Template driving surface method.

Select Initial Template – Selects initial template to use.

Select Seed Poly-Faces – Chooses the Select Seed Poly-Faces driving surface method.

Angle – Sets the angle of selected seed poly-faces

Select Poly-Faces – Selects poly-faces for a driving surface.

Direction

Select Plane – Selects a plane to use as a directional plane

XY – Sets the direction as XY.

YZ – Sets the direction as YZ.

ZX – Sets the direction as ZX.

Offset – Offsets the directional plane by a set distance.

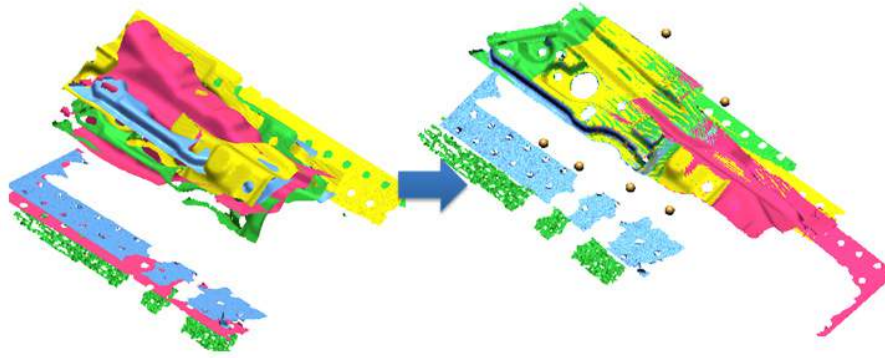
Target Template

Select Target Template – Selects a target template.

7.8.7. Add-Ins > Target AutoMatch BETA

What

The **Target AutoMatch BETA** command aligns scan data automatically based on the target information of scan shots. It checks a specified data folder for scan shots as they are created and imports and aligns data in real-time. Scan data can also be aligned to either nominal target points or with other scan data that has corresponding target points.



Why

The **Target AutoMatch BETA** command is useful for:

- Matching multiple scans of the same part using target scan balls
- Matching multiple scans in real time
- Using a single ASCII converter template to align multiple scans to each other

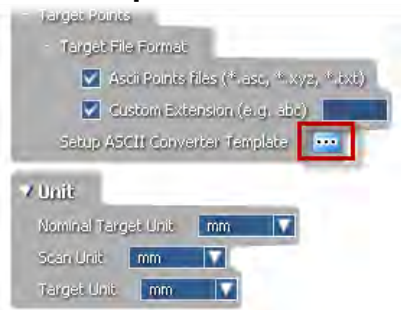
How to Use

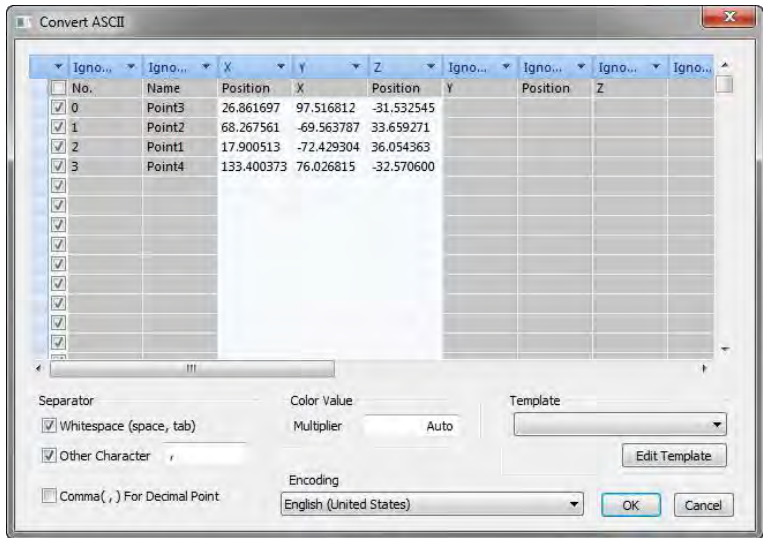
How to Target AutoMatch multiple scan files

1. Go to **Add-Ins > Target AutoMatch BETA**.
2. Click the **Open** button next to **Data Folder** and select the desired folder.



3. Click the **Open** button next to **Setup ASCII Converter Template** and choose a .txt file.

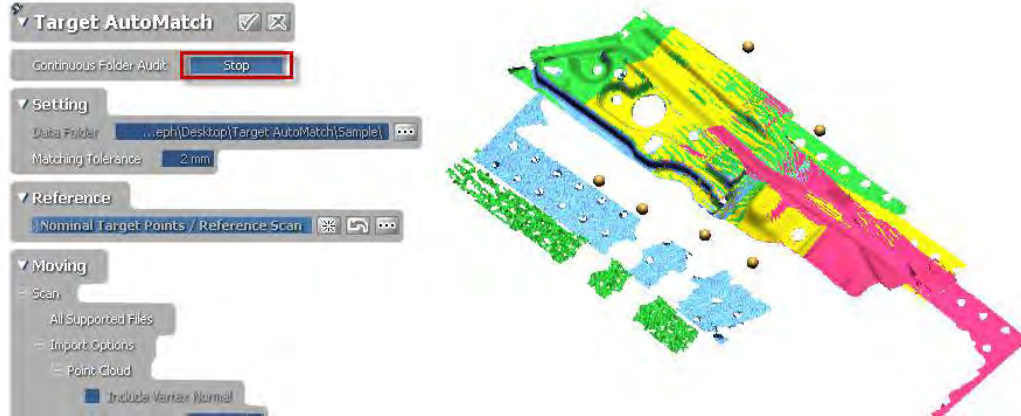




4. Click the **Start** button next to **Continuous Folder Audit** to begin processing in real-time.



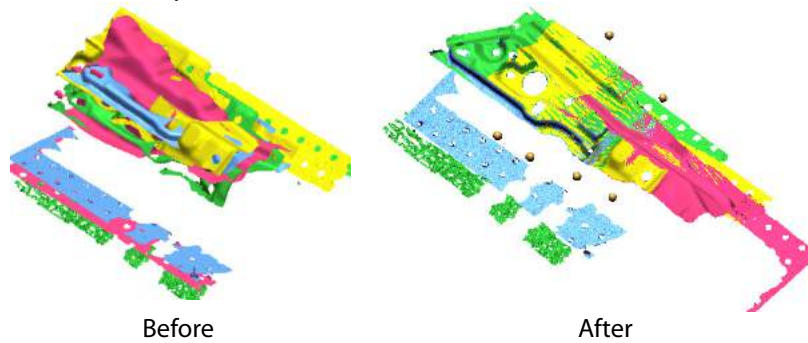
5. The scan data will be matched by their scan data balls. Click the **Stop** button to end real-time matching.



6. The Statistics for the final matched model will be displayed. Click the **OK** button.

Targets	X (Avg.)	Y (Avg.)	Z (Avg.)	Corresponding Targets Count	Avg. Error	Max. Error	RMS Error	Std. Dev. Error
Point1	26.841	97.523	-31.545	2	0.044	0.062	0.046	0.000
Point2	68.413	-69.516	33.641	3	0.144	0.206	0.150	0.002
Point3	17.864	-72.383	36.060	2	0.078	0.101	0.080	0.000
Point4	133.401	75.679	-33.014	3	0.842	1.678	0.970	0.233
Point5	143.769	-54.493	50.297	2	0.083	0.097	0.085	0.000
Point6	196.530	-49.020	45.569	1	0.024	0.024	0.024	0.000

7. Click **OK** to complete the command.



Options in Detail

Continuous Folder Audit

Start – Begins matching scan data from a selected target data folder.

Stop – Stops matching scan data from a selected target data folder.

Setting

Data Folder – Selects a target data folder

Matching Tolerance – Sets the tolerance threshold of target information on all scan data files to each other.

Reference

Nominal Target Points / Reference Scan – Selects nominal target points or reference scan data that other scan data files will be matched to.

Moving

Scan – Shows what file types are being moved

Import Options – Selects import options.

Point Cloud – Sets point cloud options.

Include Vertex Normal – Used to include the vertex normal information for scan data

Sampling Ratio – Sets the ratio for sampling of scan data from 1/1, 1/4, 1/9, 1/16, 1/25, 1/36, 1/49, and 1/64.

Mesh – Sets mesh options.

Suppress Mesh – Suppress any mesh in scan data files.

Common – Sets common options.

Include Texture – Sets texture options in scan data.

None – Excludes textures.

As It Is – Uses existing textures as set in scan data files.

As Color Points – Sets the existing textures as color points.

Target Points – Sets the target points to use for matching scan data files.

Target File Format – Selects the target file format for setting target points.

Ascii Points Files – Selects Ascii Points files (*.asc, *xyz and *.txt).

Custom Extension – Sets a custom extension.

Unit

Nominal Target Unit – Selects the unit for the nominal target.

Scan Unit – Selects the unit for scan data.

Target Unit – Selects the unit for the target data.

7.9. Help

Help > Context

The **Help > Context** menu loads the contents and index table of the help file into the Help window. When a specific item is double-clicked, the context will appear in the Help tab of the working window. To access documentation on a specific operation while a dialog tree is floating, press the F1 key while in that operation.

Help > Quick Start Guide

Going to **Help > Quick Start Guide** loads the quick start guide into the Help window. The purpose of the quick start guide is to give a “quick start” understanding of the modeling processes in the application. By working through the tutorial procedures, the fundamentals of the application can be learned.

Help > Tutorial

Going to **Help > Tutorial** loads the Tutorial that is installed with software.

Help > Release Note

Going to **Help > Release Note** loads the release note for that version of the application that shows the new and enhanced functions and fixed bugs in the current version and previous versions.

Help > Check For Update

Going to **Help > Check For Update** checks the version of the application installed on your machine and downloads the latest patch files if any exist.

Note:

There are two methods to update the application:

1. Letting the application automatically check when it is started.
2. Manually using the **Help > Check For Update** command.

Help > License Information

Going to **Help > License Information** displays the current license information, like license type and expiry date.

Help > Activate License

The **Help > Activate License** command activates a current license. Follow the on screen instructions to complete the activation.

Help > Deactivate License

The **Help > Deactivate License** command can be used to deactivate your current license. Follow the on screen instructions to complete the deactivation.

Help > Launch Performance Profiler

Going to **Help > Launch Performance Profiler** runs a utility which analyzes the graphic driver, memory, processor speed, etc. of a PC and sets the best values in the Hardware tab in the Preferences. After performing the Performance Profiler the resulting file will be saved as an .htm document in the installation folder with the name of the PC. This document contains the hardware specifications of your system.

Note:

The **Performance Profiler** can also be found in the Windows' **Start** menu, but this command doesn't save the results as an .htm file and its calculation speed is quicker than in the version found in the **Help** menu. If a system setting, for example a new graphic card has been installed, use the **Performance Profiler** in the Windows' **Start** menu to get the system information and register changed settings to the application.

When running the **Performance Profiler**, it is better to close all other applications for accurate results. The profiler takes about 2 or 3 minutes to complete the performance evaluation.

Help > Support

Going to **Help > Support** loads the Support tab which connects to the solution center. The Online Support Page in the Support tab can be searched for information. The support tab provides helpful information such as the latest tutorials, FAQs, and technical tips. As well, feedback for Geomagic products can be sent.

The Online Support Page can be accessed by going to:

<https://rapidform.zendesk.com/home>

Help > About

The **Help > About** menu displays a window including information about the application, such as the version of the product, the product ID and copyright information. Click the window to close it.

8. Glossary

This chapter explains the definition of terms used in Geomagic Design X. The following sections define many of the basic terms and concepts you should know in order to use Geomagic effectively.

A

ASME Y14.5M-1994

The national standard for dimensioning and tolerancing in the United States. ASME stands for The American Society of Mechanical Engineers. The Y14.5 is the standard number. "M" is to indicate the measurement standard is metric, and 1994 is the date the standard was officially approved.

Aspect Ratio

Ratio of width to height.

Axis

A straight virtual line that creates a basis for rotation. It can be used to create revolving features or patterns. An axis can be made in various ways. It can be created by drawing a line, extracting from a cylindrical feature, and by intersecting two planes.

B

Basic Dimension

A numerical value used to describe a theoretically exact size, true profile, orientation, or location of a feature or datum target.

Bitmap

A Bitmap is a pixel by pixel image.

Boundary

A series of poly-edges that form the borders of a model or an internal hole

Bounding Box

A bounding box is a rectangular box that fully encloses mesh data. It is oriented to the world coordinate system.

Boundary Edge

A boundary edge is a poly-edge of a polygon that is not shared with another polygon.

Base

The first solid feature of a part. Additional features are usually built from here.

Body

A 3D object. It is composed of one or more components. When a body contains more than one component, it is a disjointed body.

Boolean

A Boolean is operations that are used to combine bodies by uniting, subtracting and intersecting to create new bodies.

C

CAD

Computer Aided Design.

Control Points

A set of points that parameterize and control the shape of a NURBS curve or surface.

Curvature

This is a mathematically-defined term which refers to the amount of roundness located at a point on a curve or surface. If a curve is flat, then its curvature is zero. As the curve becomes more rounded, the radius of curvature goes down and the curvature goes up. Curvature $k = 1 / \rho$, where ρ is defined as the radius of curvature. These two values ('k' and 'rho') are inversely related.

Curve

A curve is a mathematical function that describes the path of a point moving through space. In the application, curves are displayed by sampling the mathematical description of the curve at specified intervals, then drawing line segments between these sample points.

Centerline

A centerline marks an axis of symmetry in a sketch or drawing in a phantom font.

Chamfer

Related to a blend. Instead of modifying an edge by smoothing it over, the edge is shaved with a flat cutter, creating two new sharp edges and an extra face, the surface of which is a plane, cone or B-surface.

Closed profile

A closed profile is a sketch or sketch entity with no exposed endpoints. It can be a circle, a polygon, or shape composed by lines and arcs.

Cut

To remove a part from a body or from scan data, surface bodies, planes, or lines can be used to trim off a target feature.

Continuity

The continuity between the patches of a B-surface or segments of a B-surface can be of various levels:

'C' Continuity for curves and surfaces:

C0 means that the positions are continuous in both magnitude and direction

C1 means that the positions and first derivatives are continuous in both magnitude and direction

C2 means that the positions and first and second derivatives are continuous in both magnitude and direction

'G' Continuity for curves:

G0 means that the positions are continuous in direction but not necessarily in magnitude

G1 means that the positions and first derivatives are continuous in direction but not necessarily in magnitude

G2 means that the positions and first and second derivatives are continuous in direction but not necessarily in magnitude

'G' Continuity for surfaces:

G0 means that the positions are continuous in direction but not necessarily in magnitude

G1 means that the positions and surface normals are continuous in direction but not necessarily in magnitude

G2 means that the positions, surface normals, and curvatures are continuous in direction but not necessarily in magnitude

D

Datum

A theoretical exact point, axis, or plane derived from the true geometric counterpart of a specified datum feature. A datum is the origin from which the location or geometric characteristics of the features of a part are established.

Situation features are used to define the position and/or orientation of a tolerance zone, or to define the position and/or orientation of the virtual conditions (in the case of complementary requirements, e.g. maximum material requirement).

Datum Feature

A feature of a part that is used to establish a datum.

Datum Reference Frame

A datum system is an ordered list of at least one datums and at most three datums, which may be single or common datums. They are a set of symbols and rules that communicate to the user who is drawing on how dimensional measurements are to be made.

Datum Target

A specified point, line, or area on a part used to establish a datum. It can also be a portion of an integral feature (surface) which can be a point, a line or an area.

Decimation

An operation that reduces the number of entities in an object. It attempts to preserve the object's overall shape and sharp features while simplifying it.

Derived Feature

A derived feature is the center point, median line or median surface from one or more integral features.

Dimension

A numerical value expressed in appropriate units of measurement and used to define the size, location, orientation, form, or other geometric characteristics of a part.

Drag-select

The selection of entities by moving the mouse while holding down the left mouse button.

Draft

Draft is incorporated into the wall of a mold so that the opening of the cavity is wider than its base. Draft allows for easier ejection of the part from the mold.

E

Extrude

An extrude is the shape created by moving a profile along a path that is line.

Entity

An entity is an item in the model which can be accessed by the application.

F

Features

A general term applied to a physical portion of a part, such as a surface, pin, tab, hole or slot.

Fillet

A fillet is an operation that rounds a corner of a profile or an edge of a solid / surface to a specified radius.

G

Guide curve

A guide curve is either a 2D or 3D curve. It is used to generate the outer shape of a sweep or loft feature.

Geometries

Geometries are the points, curves and surfaces which specify the shape and position of an element.

H

Healing

A series of functions whose purpose is to correct geometric abnormalities contained within a model. Common functions may include: surface simplification (mapping spline surfaces to analytic shapes such as planes, cylinders, cones, etc.), poly-face stitching, and poly-edge gap repair. More advanced functions may include the ability to merge coincident vertices, repair self-intersections, and reduce and/or redeploy curve and surface control nodes. Programs typically perform healing on geometry that you import through neutral formats such as IGES, STEP, and SAT with problems caused by conditions ranging from imprecise geometric definitions to poor entity mapping to poorly written translator code.

I

IGES

Initial Graphics Exchange Specification, a standard for exchanging geometric information

Interpolation

The process of defining a curve (or surface) that passes through a set of points.

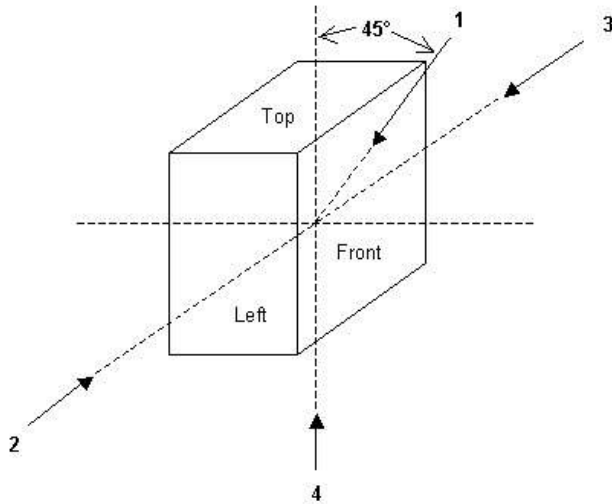
Iso-Lines

An isoline is a curve parameterized by one of the surface parameters (u,v) while the other surface parameter is fixed. Isolines provide a simple means to display a surface. By fixing the 'u' parameter of surface, the iso-u lines of the form $c(v) = s(u_0,v)$ are produced, and by fixing the v parameter, the iso-v lines of the form $c(u) = s(u,v_0)$ are produced.

L

Light

A source used to illuminate a model. In the application, there are 4 lights you can use. They are used to display mesh data with Shaded or Transparent rendering modes. The position of each light source is shown in the following picture.



LMB

Left Mouse Button

Loop

A fundamental item of topology representing one boundary of a face as a closed and connected set of edges.

Loft

The loft command connects several sectional profiles and then generates a smooth feature.

M

Mouse Drag

Moving the mouse while pressing and holding a mouse button.

Mirror

The mirror command generates symmetric features across a plane or planar face. In the sketch mode, a line is used instead of a plane.

N

Noise

Noise is erroneous data resulting from the scanning process, is usually represented as unnecessary points in 3D point cloud.

Normal

A surface normal at a point on a surface is a vector which is locally perpendicular to the surface. Normal describes a unit vector.

NURBS

Non-Uniform Rational B-Spline. This is a type of spline where curves pass close to, but not through control points or knots and is used in spline-based modeling. Also commonly used to describe NURBS surfaces; which are surfaces comprised of a number of splines in two directions (U and V).

P

Parallel Projection view

Parallel Projection view shows a model without any parallax.

Parallelism

Parallelism is the condition that results when a surface, axis or center plane is exactly parallel to a datum.

Path

A path is an element composed of lines or curves used to generate a sweep or loft feature.

Pattern

A pattern is a series of features from a selected seed feature. If the seed entity is changed, the other instances in the pattern update.

Plane

A plane is a flat construction geometry which can be used as the basis of 2D sketches, mirror planes, cutting a model, etc.

Point

A point is a single point construction geometry, which can be used to create other objects.

Poly-edge

A line connecting two adjacent vertices on a polygon.

Polygonal model

A model composed of polygonal faces.

Poly-face

A poly-face is a flat N-sided face composed of N vertices. The application only allows 3-sided faces which are three poly-edges connected by vertices.

Polyline

A collection of line segments, connected end to end.

Poly-Vertex

A point at which two or more lines or poly-edges intersect. Vertices can be selected for sketching, dimensioning, and many other operations.

Primitives

Pre-defined basic objects, either 2D or 3D, i.e., circle, square, arc, plane or sphere, cube, cylinder, cone, torus, and various polyhedrons.

Profile

A sketch entity or a group of edges on a body that is used to create a loft body, an extrude body and so on.

R

Region Groups

Region groups are user-defined geometric feature areas on a mesh. They can be used to extract design intent from a scanned object and to help users create surface or solid bodies based on geometric information from a target mesh.

RMB

Right Mouse Button.

Reference Geometry

Basic geometrical elements, such as planes, axes, and coordinate systems. Reference geometry is not a body but used to assist in creating features.

Revolve

A feature can be generated by revolving sketched profiles around a centerline.

Rollback

Enables a model to be returned to a previous point in a construction history. Changes made to a feature are reflected in subsequent features once Rolled Forward. This command is commonly used to change the parameters of previous features.

S

SAT

A file format for 3D solid geometry that is native for any ACIS modeling software developed by Spatial Technology Corp.

STL

Stereo Lithography file format. Contains polygonal mesh data defined by vertex, and connections between them. Can be written in ASCII (text) or binary.

Surface

A surface is used to describe the skin of a 3D geometric element.

Sweep

A feature is generated by moving a profile (section) along a path.

Spline

A 2D or 3D curve consisting of several control points.

Sketch

A group of sketch entities (lines, arcs, splines, etc) drawn on a plane or planar face. A 3D sketch is a non-planar entity and can be used as a path for a sweep or loft feature.

T

Tessellation

Tessellation is the process of subdividing a surface into smaller shapes. Tessellation is used to convert a Solid, Surface, or Reference object into a polygonal mesh. To create object surface patterns, tessellation breaks down the surface of an object into manageable polygons. Triangles are frequently used in drawing graphical objects because computer hardware can easily manipulate and calculate them.

3-2-1 Rule

A rule that defines the minimum number of points of contact required for a part datum feature with its primary, secondary, and tertiary datum planes.

Toggle

To switch from one setting to another. The term toggle implies that there are only two possible settings and that you are switching from the current setting to the other setting.

Tolerance

The value that features of a part are permitted to vary from the specified dimension.

Toolbar

Movable palette of tools that performs operations

Tangent arc

An arc that is tangent to another entity, such as a line and an arc.

U

UV

Instead of X and Y coordinates, the U and V coordinates are used as a base coordinate system for representing a 2D-based work space or entity.

V

Viewports

Windows that display 3D views of models.

W

Wireframe

One of the display methods for solid and surface bodies. In this mode only the body edges are displayed and the faces are hidden.

World Coordinates

In 3D space, World Coordinates are a rectangular Cartesian coordinate system used to define the location of models, lights, cameras, etc. It is independent of viewpoint and display. The origin of the world coordinate system is (0, 0, 0).

9. Index

3

3 Points Arc	757
3 Tangent Arc	758
3 Tangent Circle.....	751
3D Digital Scanner.....	535
3D Mesh Sketch.....	493
3D Mouse.....	135
3D Sketch.....	494
3D Sketches.....	912

A

Accuracy Analyzer	108
Accuracy Analyzer(TM)	82
Add Poly-Face.....	639
Add-Ins	941
Align	873
Align Between Scan Data	561
Align Wizard(TM)	77
All	223
Angle	890
Application Field of 3D Scan Data	52
Area.....	922
Auto Segment	680
Auto Sketch	80
Auto Surfacing.....	76, 349
AutoCAD.....	190
Average Meshes.....	582

B

Basic Entity Terminology	21
Basic Reverse Design Workflow	48
Basic Scan Data Processing Workflow.....	46
Batch Process	199
Best Fit	882
Body Display Mode	928
Body Tools	871
Boolean	332, 586
Boundary.....	831
Boundary Entities.....	227
Breuckmann	529
By Color	237

By Criteria	235
By Import	777

C

CAD Correct	477
CAD-Ready File.....	53
Capture Screen	158
Category of 3D Scan Data	55
Centerline	770
Centerpoint Arc.....	752
Chamfer	433, 783
Changing Units.....	42
Circle	746
Circular Pattern	444
Color Bar	116
Combine.....	571
Console	48, 123
Construct Mesh.....	704
Convert Body	676
Convert Color To Texture	728
Convert Current Selection	233
Convert Entities.....	786, 855
Convert Texture To Color	727
Coordinate	286
Copy	244
Copy Texture/Vertex Color.....	729
Creaform Portable 3D Scanners.....	514, 520
Create Curve Network	844
Curve Pattern	446
Custom Region.....	226
Cut.....	243, 330

D

Data Processing.....	41
Datum Match	884
Decimate.....	648
Decimate Meshes	573
Defeature	635
Delete.....	245
Delete Face	463
Delete Poly-Faces	946
Design	199
Dialog Tree	33, 126
Display.....	98

Distance.....	887
Docking Bar Convention.....	37
Docking Bars.....	94
Draft	435
Duplicate Body	474

E

Edit	240
Edit Boundaries.....	599
Edit Poly-Edge.....	619
Edit Texture	715, 720
Ellipse	760
Emboss.....	438
Enhance Shape.....	655
Enlarge	230, 691
Entities Around Edges/Faces.....	228
Error List.....	123
Esc Key.....	129
Exit.....	218
Export.....	148
Export Texture	726
Exporting a Model.....	50
Extend	381, 792, 860
Exterior Poly-Face.....	239
Extract Feature Curves.....	841
Extrude.....	305, 359
Extrusion Wizard	404

F

Feature	250
Features of Geomagic Design X	9
File.....	139
Fill Face.....	377
Fill Holes.....	630
Fillet.....	425, 781
Filter Noise	695
Find Defect	591, 871
Fit	924
Fit From Points	835
Fit Polyline.....	775
Fitting Surface Modeling	72
Fix Normal.....	597
Flatten And Deform.....	951

G

Getting Help	15, 85
Global Remesh	611
Glossary.....	961

H

Healing Wizard.....	594
Helix	452
Help.....	113, 958
Hide All.....	905
Hollow	328
Hybrid Modeling	73

I

ICF	154
IGES Point Importer	943
Image Folder	121
Import.....	252
Import & Split	942
Import Polyline	808
Importing Scan Data	44
Index	971
Insert.....	250, 661, 689
Insert Node	867
Installation	18
Interactive Alignment	876
Intersection.....	830
Introduction.....	1
Introduction to the Interface	28
Inventor.....	192
Inverse.....	225

K

Key Themes of Geomagic Design X.....	4
KODEN OCM-A	532
Konica Minolta VIVID	524

L

Light Properties.....	936
Line.....	737
Linear Pattern	441, 799
LiveScan(TM).....	496
LiveTransfer(TM)	84, 163
Loft.....	319, 369
Loft Wizard.....	416

M

Match.....	863
Match Color	730
Measure	887
Measurements.....	918
Menu.....	139

Menu Bar	87
Merge.....	567, 685, 866
Mesh	261
Mesh Buildup Wizard	541
Mesh Buildup Wizard(TM)	75
Mesh Deviations	897
Mesh Display Mode	926
Mesh Fit	336
Mesh Modeling	68
Mesh Sketch	483
Mesh Tools	589
Mesh vs. Point Cloud.....	63
Meshes.....	906
Mirror	449, 669, 795, 853
Mode.....	88, 220
Mode Convention	36
Modeling Feature.....	396, 424
Monitoring	129
Mouse Control	130
Move Face	461

N

Navigating Entities	38
New	139
None.....	224
Normal Information Manager	576

O

Offset.....	324, 375, 671, 702, 784, 849
Open.....	140
Optimize Mesh.....	614

P

Parabola	764
Parallelogram	742
Parametric Feature Modeling	70
Partial Ellipse.....	762
Paste.....	245
Pencil.....	824
Perimeter Circle.....	749
PIP	925
Pipe Wizard	419
Plane	268
Point	298
Point	772
Point Cloud.....	265
Point Cloud Tools.....	694
Point Clouds	908
Point Stream.....	150
Polygon.....	743

Polyline	290
Pop-up Menus.....	133
Power of 3D Scan Data.....	51
Powerful Tools for Reverse Modeling	74
Preferences	205
Previous Area	923
Primitives	397, 400
Print	160
Print Preview.....	161
Print Setup	162
Pro/E Wildfire	188
Profile View.....	124
Progress Bar	129
Project.....	851
Properties	115
Publish	150

Q

Quick Fit	880
Quick Modeling.....	136

R

Radius	891
RANGE5/RANGE 7.....	535
Rebuild	246
Recent Files.....	217
Rectangle	741
Redesign Assistant(TM)	78
Redo.....	242
Ref. Coordinates.....	917
Ref. Geometry	267
Ref. Planes.....	915
Ref. Points	913
Ref. Polylines.....	916
Ref. Vector.....	914
Refit.....	478, 657
Region Group	264
Region Tools.....	679
Regions.....	907
Register Target	943
Remove Body	476
Remove Constraint	870
Remove Marker	637
Remove Node	869
Remove Texture	724
Repeat Last Command.....	243
Replace Face	465
Resegment	683
Retriangulate	609
Reverse Modeling Strategy	66
Reverse Normal.....	395

Revolution Wizard.....	408
Revolve.....	311, 362
Rewrap.....	626
Roland LPX.....	532
Roll Forward.....	247
Roll To End.....	248
Rollback.....	247
Run.....	203

S

Sampling.....	697
Save.....	141
Save As.....	142, 143
Save As CATIA V4.....	196
Save As CATIA V5.....	198
Scan Tools.....	540
Scanner Direct Control.....	496
Section.....	826, 893
Select.....	218
Selection.....	35
Separate.....	690
Separate Point Cloud.....	712
Sew.....	390
Sew Boundaries.....	607
Shining Tech.....	536
Shortcut Keys.....	131
Show All.....	904
Show Selected Only.....	903
Shrink.....	232, 693
Siemens NX.....	186
Silhouette.....	788
Sketch.....	490
Sketch Entities.....	732, 818
Sketch Options.....	810
Sketch Tools.....	779, 848
Sketches.....	911
Slot.....	745
Smart Brush.....	643
Smart Dimension.....	815
Smart Visibility.....	919
Smartsnap.....	814
Smooth.....	641, 700, 857
Snap On Entities.....	810
Snap On Polylines.....	812
Solid.....	305
Solid Bodies.....	910
SolidWorks.....	182, 194
Spiral.....	457
Spline.....	765, 819
Split.....	663, 686, 794, 861
Split Face.....	467
Status Bar.....	129
Subdivide.....	651

Summary Info.....	216
Support.....	120
Suppress.....	249
Surface.....	334
Surface Bodies.....	909
Surface UV Curves.....	833
Surface> Boundary Fit.....	343
Sweep.....	315, 365
Sweep Wizard.....	412
System Requirement.....	15

T

Tangent Arc.....	755
Target AutoMatch.....	954
Text.....	773
Text To Splines.....	807
Texture Tools.....	714
Thicken.....	326, 673
Time Indicator.....	129
Title Bar.....	87
Toolbar.....	90
Tools.....	539
Trace Feature Line.....	839
Transform Body.....	470
Transform Scan Data.....	578
Tree.....	95
Triangulate.....	553
Trim.....	384, 666, 790, 858
Trim & Merge.....	389
Trim Mesh.....	947

U

Understanding Mesh.....	59
Understanding Point Clouds.....	56
Understanding Reverse Design.....	65
Understanding Scan Data.....	51
Understanding User Interface Philosophy.....	33
Undo.....	241
Unsuppress.....	249
Untrim.....	387
User Interface.....	87
User Interface Overview.....	28

V

Various Modeling Wizards(TM).....	80
Vector.....	278
View.....	900, 902
View Clip.....	938
Viewing Window.....	125
Viewpoint.....	114, 930

Viewport.....932
Virtual Rescan621

Why 3D Scanning? 51
Wizard 873

W

Walk Through940
What Can Geomagic Design X Do?20
What is Reverse Design?65

Z

Z Corporation ZScanner 700 522
Zoom 921