

# ANSYS 2025/R1

POWERING INNOVATION THAT DRIVES HUMAN ADVANCEMENT

© 2025 ANSYS, Inc. or its affiliated companies  
Unauthorized use, distribution, or duplication is prohibited.

## ANSYS TurboGrid User's Guide

---



ANSYS, Inc.  
Southpointe  
2600 Ansys Drive  
Canonsburg, PA 15317  
ansysinfo@ansys.com  
<http://www.ansys.com>  
(T) 724-746-3304  
(F) 724-514-9494

Release 2025 R1  
January 2025

ANSYS, Inc. and  
ANSYS Europe,  
Ltd. are UL  
registered ISO  
9001:2015  
companies.

---

## Copyright and Trademark Information

© 2025 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

ANSYS, Ansys Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners. FLEXIm and FLEXnet are trademarks of Flexera Software LLC.

## Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2015 companies.

## U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

## Third-Party Software

See the [legal information](#) in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, contact ANSYS, Inc.

Published in the U.S.A.

---

---

# Table of Contents

<b>1. Ansys TurboGrid Interface</b> .....	19
1.1. Introduction .....	19
1.2. Object Selector .....	20
1.2.1. Visibility Check Box .....	22
1.3. Object Editor .....	22
1.3.1. Common Options .....	23
1.3.2. Data Tabs .....	23
1.3.3. Color Tab .....	23
1.3.3.1. Mode .....	23
1.3.3.2. Variable .....	24
1.3.3.3. Range .....	24
1.3.3.4. Hybrid and Conservative Values .....	25
1.3.3.5. Undefined Color .....	25
1.3.4. Render Tab .....	25
1.3.4.1. Draw Faces .....	25
1.3.4.2. Draw Lines .....	26
1.3.4.3. Applying Instance Transforms to Objects .....	26
1.4. Mesh Workspace .....	27
1.5. Geometry Workspace .....	28
<b>2. File Menu</b> .....	29
2.1. Introduction .....	29
2.2. New Case Command .....	31
2.3. Load TurboGrid Init File Command .....	31
2.3.1. BladeGen (*.inf) File Details .....	32
2.3.2. TurboGrid Init (*.tginit) File Details .....	32
2.4. Load Profile Points Or CAD Command .....	32
2.5. Import NDF File Command .....	32
2.6. Update Geometry Command .....	33
2.7. Load State Command (Import State Command) .....	34
2.8. Save State Command (Export State Command) .....	35
2.9. Save State As Command .....	35
2.10. Save Project .....	36
2.11. Refresh .....	36
2.12. Save Submenu .....	36
2.12.1. Save Blade Command .....	36
2.12.2. Save Blade As Command .....	36
2.12.3. Save Periodic/Interface Surfaces Command .....	37
2.12.4. Save Inlet Command .....	37
2.12.5. Save Inlet As Command .....	37
2.12.6. Save Outlet Command .....	37
2.12.7. Save Outlet As Command .....	37
2.12.8. Save Topology Command .....	37
2.12.9. Save Topology As Command .....	37
2.13. Save Mesh Command .....	38
2.14. Save Mesh As Command .....	38
2.14.1. Save Mesh Dialog Box .....	38
2.14.1.1. File Type .....	38
2.14.1.2. Export Units .....	39
2.14.1.3. Write In/Out/Passage As .....	39

2.14.1.4. Write Single Precision Mesh .....	39
2.14.1.5. Write Turbo Coordinates Data .....	39
2.14.1.6. Region Name Prefix .....	40
2.14.2. Region Naming .....	41
2.15. Export Submenu .....	41
2.15.1. Export Geometry Command .....	41
2.15.2. Export Turbo Coordinates Data Command .....	41
2.16. Save Picture Command .....	42
2.17. Recent State Files Submenu .....	46
2.18. Recent Session Files Submenu .....	46
2.19. Quit Command .....	46
<b>3. Edit Menu .....</b>	<b>47</b>
3.1. Undo and Redo Commands .....	47
3.2. Options Command .....	47
3.2.1. TurboGrid Options .....	48
3.2.1.1. Viewer .....	49
3.2.1.1.1. Background .....	49
3.2.1.1.2. Text Color and Edge Color .....	49
3.2.1.1.3. Axis Visibility .....	49
3.2.1.1.4. Ruler Visibility .....	49
3.2.1.1.5. Stereo .....	50
3.2.1.2. Mesh Quality Measures .....	50
3.2.2. Common Options .....	50
3.2.2.1. Appearance .....	50
3.2.2.2. Threading .....	51
3.2.2.3. Units .....	51
3.2.2.4. Viewer Setup: General .....	51
3.2.2.4.1. Double Buffering .....	52
3.2.2.4.2. Unlimited Zoom .....	52
3.2.2.4.3. Use GPU Rendering for Printing .....	52
3.2.2.5. Viewer Setup: Mouse Mapping .....	52
<b>4. Session Menu .....</b>	<b>55</b>
4.1. Introduction .....	55
4.2. Play Session Command .....	56
4.3. New Session Command .....	57
4.4. Start Recording Command .....	58
4.5. Stop Recording Command .....	58
<b>5. Insert Menu .....</b>	<b>59</b>
5.1. Introduction .....	59
5.2. Mesh Command .....	59
5.3. User Defined Submenu .....	60
5.3.1. Point Command .....	60
5.3.1.1. Point: Geometry Tab .....	60
5.3.1.1.1. Point Definition .....	60
5.3.1.1.2. Symbol Definition .....	61
5.3.1.2. Point: Color Tab .....	61
5.3.1.3. Point: Render Tab .....	61
5.3.2. Line Command .....	61
5.3.2.1. Line: Geometry Tab .....	61
5.3.2.1.1. Domains .....	61
5.3.2.1.2. Line Definition .....	61

5.3.2.1.3. Line Type .....	62
5.3.2.2. Line: Color Tab .....	62
5.3.2.3. Line: Render Tab .....	62
5.3.3. Plane Command .....	62
5.3.3.1. Plane: Geometry Tab .....	62
5.3.3.1.1. Domains .....	62
5.3.3.1.2. Plane Definition .....	62
5.3.3.1.3. Plane Bounds .....	64
5.3.3.1.4. Plane Type .....	64
5.3.3.2. Plane: Color Tab .....	64
5.3.3.3. Plane: Render Tab .....	65
5.3.4. Turbo Surface Command .....	65
5.3.4.1. Turbo Surface: Geometry Tab .....	65
5.3.4.1.1. Domains .....	65
5.3.4.1.2. Turbo Surface Definition .....	65
5.3.4.2. Turbo Surface: Color Tab .....	65
5.3.4.3. Turbo Surface: Render Tab .....	65
5.3.5. Volume Command .....	65
5.3.5.1. Volume: Geometry Tab .....	65
5.3.5.1.1. Domains .....	65
5.3.5.1.2. Volume Definition .....	65
5.3.5.1.3. Hybrid/Conservative .....	66
5.3.5.2. Volume: Color Tab .....	67
5.3.5.3. Volume: Render Tab .....	67
5.3.6. Isosurface Command .....	67
5.3.6.1. Isosurface: Geometry Tab .....	67
5.3.6.1.1. Domains .....	67
5.3.6.1.2. Isosurface Definition .....	67
5.3.6.1.3. Hybrid/Conservative .....	67
5.3.6.2. Isosurface: Color Tab .....	67
5.3.6.3. Isosurface: Render Tab .....	67
5.3.7. Polyline Command .....	68
5.3.7.1. Polyline: Geometry Tab .....	68
5.3.7.2. Polyline: Color Tab .....	68
5.3.7.3. Polyline: Render Tab .....	69
5.3.8. Surface Command .....	69
5.3.8.1. User Surface: Geometry Tab .....	69
5.3.8.1.1. Method .....	69
5.3.8.1.2. Surface Data Format .....	69
5.3.8.2. User Surface: Color Tab .....	70
5.3.8.3. User Surface: Render Tab .....	70
5.3.8.4. Surface Groups .....	70
5.3.8.4.1. Surface Group: Definition Tab .....	71
5.3.8.4.2. Surface Group: Color Tab .....	71
5.3.8.4.3. Surface Group: Render Tab .....	71
5.3.9. Contour Command .....	71
5.3.9.1. Contour Plot: Definition Tab .....	72
5.3.9.1.1. Domains .....	72
5.3.9.1.2. Locations .....	72
5.3.9.1.3. Variable .....	72
5.3.9.1.4. Range .....	72

5.3.9.1.5. Hybrid/Conservative .....	72
5.3.9.1.6. Number of Contours .....	72
5.3.9.2. Contour Plot: Labels Tab .....	73
5.3.9.3. Contour Plot: Render Tab .....	73
5.3.10. Instance Transform Command .....	73
5.3.10.1. Instance Transform: Definition Tab .....	74
5.3.10.1.1. Number of Copies .....	74
5.3.10.1.2. CCL Editing .....	74
5.3.11. Legend Command .....	74
5.3.11.1. Legend: Definition Tab .....	74
5.3.11.1.1. Plot .....	74
5.3.11.1.2. Location .....	74
5.3.11.2. Legend: Appearance Tab .....	75
5.3.11.2.1. Sizing Parameters .....	75
5.3.11.2.2. Text Parameters .....	75
5.3.12. Text Command .....	75
5.3.12.1. Text: Definition Tab .....	75
5.3.12.1.1. Location .....	75
5.3.12.2. Text: Appearance Tab .....	76
5.3.12.2.1. Text Properties .....	76
<b>6. Display Menu .....</b>	<b>77</b>
6.1. Introduction .....	77
6.2. Display One Instance Command .....	77
6.3. Display Two Instances Command .....	77
6.4. Display All Instances Command .....	78
6.5. Hide/Unhide Geometry Objects Commands .....	78
6.6. Hide/Unhide Layers Commands .....	78
6.7. Hide/Unhide Mesh Objects Commands .....	78
6.8. Use Mesh for Low/High-Fidelity Geometry Commands .....	78
6.9. Blade-to-Blade View Submenu .....	78
<b>7. Viewer .....</b>	<b>81</b>
7.1. Introduction .....	81
7.2. Viewer Toolbar .....	81
7.3. Viewer Hotkeys .....	83
7.4. Multiple Viewports .....	84
7.4.1. Selecting, Adding, and Deleting Views .....	85
7.5. Selecting and Dragging Objects while in Viewing Mode .....	85
7.6. Stereo Viewer .....	86
<b>8. Tools Menu .....</b>	<b>87</b>
8.1. Calculator Command .....	87
8.1.1. Function Calculator Dialog Box .....	87
8.1.1.1. Function .....	87
8.1.1.1.1. area .....	88
8.1.1.1.2. areaAve .....	88
8.1.1.1.3. arealnt .....	88
8.1.1.1.4. ave .....	89
8.1.1.1.5. count .....	89
8.1.1.1.6. length .....	89
8.1.1.1.7. lengthAve .....	89
8.1.1.1.8. lengthInt .....	90
8.1.1.1.9. maxVal .....	90

8.1.1.1.10. minVal .....	90
8.1.1.1.11. probe .....	90
8.1.1.1.12. sum .....	90
8.1.1.1.13. volume .....	91
8.1.1.1.14. volumeAve .....	91
8.1.1.1.15. volumeInt .....	91
8.1.1.2. Location .....	91
8.1.1.3. Variable .....	91
8.1.1.4. Direction .....	91
8.1.1.5. Hybrid and Conservative Variables .....	92
8.2. Expressions Command .....	92
8.2.1. Expression Editor Dialog Box .....	92
8.2.1.1. Expression Editor Example .....	94
8.3. Variables Command .....	94
8.3.1. Variable Editor Dialog Box .....	94
8.3.1.1. Name .....	95
8.3.1.2. Type .....	95
8.3.1.3. Hybrid and Conservative Variable Values .....	96
8.3.1.4. Variable Editor Example .....	96
8.4. Command Editor Command .....	96
8.5. Reset Inlet/Outlet Points Command .....	97
<b>9. Help Menu .....</b>	<b>99</b>
<b>10. Ansys TurboGrid Workflow .....</b>	<b>101</b>
10.1. Introduction .....	101
10.2. Steps to Create a Mesh .....	101
10.3. Geometry .....	103
10.3.1. Defining Geometry from a CAD Source .....	103
10.3.1.1. Loading CAD Data From BladeEditor .....	104
10.3.1.1.1. Troubleshooting Blends (Fillets) from BladeEditor .....	105
10.3.1.2. Loading CAD Data From a CAD Input File .....	106
10.3.1.3. Objects in the Geometry Workspace (CAD Mode) .....	106
10.3.1.4. Geometry Browser Settings (CAD Mode) .....	107
10.3.1.4.1. CAD Input Definition .....	107
10.3.1.4.2. Geometry Setup (CAD Mode) .....	107
10.3.1.4.2.1. Rotation (CAD Mode) .....	107
10.3.1.4.2.2. # of Bladesets (CAD Mode) .....	108
10.3.1.5. Limitations of CAD Geometry Usage .....	108
10.3.2. Defining Geometry from Profile Points .....	108
10.3.2.1. Objects in the Geometry Workspace (Profile Points Mode) .....	110
10.3.2.2. Geometry Browser Settings (Profile Points Mode) .....	110
10.3.2.2.1. Point Data Definition .....	111
10.3.2.2.1.1. TurboGrid Curve Files .....	111
10.3.2.2.1.2. Coordinates and Units .....	111
10.3.2.2.2. Geometry Setup (Profile Points Mode) .....	111
10.3.2.2.2.1. Rotation (Profile Points Mode) .....	111
10.3.2.2.2.2. # of Bladesets (Profile Points Mode) .....	111
10.3.2.2.2.3. Leading/Trailing Edge Definition on the Blade (Profile Points Mode Only) ...	111
10.3.3. Defining Geometry from Profile Points and a CAD Input File .....	111
10.3.3.1. Geometry Browser Settings (Profile Points With Secondary CAD Mode) .....	112
10.3.4. Defining Geometry from Generated CAD: CAD From NDF .....	112
10.3.5. Defining Geometry from Generated CAD: CAD From Profile Points .....	112

10.3.5.1. Geometry Browser Settings (CAD From Profile Points Mode) .....	114
10.3.5.2. Limitations of CAD From Profile Points Input Mode .....	117
10.3.6. The Geometry Objects in the Mesh Workspace .....	118
10.3.6.1. The Machine Data Object .....	118
10.3.6.1.1. Data Tab .....	119
10.3.6.1.1.1. Pitch Angle .....	119
10.3.6.1.1.2. Rotation .....	119
10.3.6.1.1.3. Units .....	119
10.3.6.1.1.4. Machine Type .....	120
10.3.6.1.2. Rotation Axis Visibility .....	120
10.3.6.2. The Hub and Shroud Objects .....	120
10.3.6.2.1. Data Hub and Data Shroud Tabs .....	121
10.3.6.2.1.1. Coordinate System and Hub/Shroud File Definition .....	121
10.3.6.2.1.2. Geometric Representation .....	121
10.3.6.2.1.2.1. Curve Type .....	121
10.3.6.2.1.3. Curve or Surface Visibility .....	121
10.3.6.2.1.4. Draw Trimmed Portion Of Curve .....	122
10.3.6.2.1.5. Reread Button .....	122
10.3.6.2.2. Transform Tab .....	122
10.3.6.2.2.1. General Rotation .....	122
10.3.6.2.2.2. Translation .....	122
10.3.6.2.2.3. Axial Rotation .....	122
10.3.6.2.3. Hub Regions and Shroud Regions Tab .....	122
10.3.6.3. The Blade Set Object and Blade Objects .....	124
10.3.6.3.1. Blade Tab .....	125
10.3.6.3.1.1. Apply Settings To All Blades Check Box .....	125
10.3.6.3.1.2. Coordinate System and Blade File Definition .....	125
10.3.6.3.1.2.1. File Name .....	125
10.3.6.3.1.2.2. Coordinates, Angle Units, and Length Units .....	126
10.3.6.3.1.3. Geometric Representation .....	126
10.3.6.3.1.3.1. Method .....	126
10.3.6.3.1.3.2. Lofting .....	127
10.3.6.3.1.3.3. Curve Type .....	129
10.3.6.3.1.3.4. Surface Type .....	129
10.3.6.3.1.4. Leading and Trailing Edge Definitions .....	129
10.3.6.3.1.4.1. Cut-off or square .....	130
10.3.6.3.1.5. Curve or Surface Visibility .....	131
10.3.6.3.1.6. Reread Button .....	131
10.3.6.3.1.7. Save & Load Button .....	131
10.3.6.3.2. Transform Tab .....	131
10.3.6.3.2.1. General Rotation .....	131
10.3.6.3.2.2. Translation .....	131
10.3.6.3.2.3. Axial Rotation .....	131
10.3.6.3.3. Blade CAD Features .....	132
10.3.6.3.3.1. Blade Blends Tab .....	132
10.3.6.3.3.1.1. Hub Blend and Shroud Blend .....	133
10.3.6.3.3.1.2. Blade CAD Rotation Axis .....	134
10.3.6.3.3.1.3. Generated CAD Definition .....	134
10.3.6.3.3.2. Hub Partial Tip and Shroud Partial Tip tabs .....	134
10.3.6.3.3.2.1. Hub Tip and Shroud Tip .....	134
10.3.6.3.3.2.2. Blade CAD Rotation Axis .....	135

10.3.6.3.3.2.3. Generated CAD Definition .....	136
10.3.6.3.3.3. Blade Stagger tab .....	136
10.3.6.3.3.3.1. Blade Stagger .....	136
10.3.6.3.3.3.2. Blade CAD Rotation Axis .....	137
10.3.6.3.3.3.3. Generated CAD Definition .....	137
10.3.6.3.3.4. Common Settings .....	137
10.3.6.3.3.4.1. Blade CAD Rotation Axis .....	137
10.3.6.3.3.4.2. Generated CAD Definition .....	137
10.3.6.3.3.4.3. Apply And Generate CAD Button .....	137
10.3.6.3.3.4.4. Apply Button .....	138
10.3.6.3.4. Leading Edge .....	138
10.3.6.3.4.1. Leading Edge Tab .....	138
10.3.6.3.4.1.1. Curve .....	138
10.3.6.3.4.1.2. Save & Load Button .....	139
10.3.6.3.5. Trailing Edge .....	139
10.3.6.3.6. Complex Blade End Object .....	139
10.3.6.3.6.1. Complex Blade End Tab .....	139
10.3.6.4. The Hub Tip and Shroud Tip Objects .....	141
10.3.6.4.1. Hub Tip and Shroud Tip Tabs .....	141
10.3.6.4.1.1. Override Upstream Geometry Options .....	141
10.3.6.4.1.2. Clearance Type .....	141
10.3.6.5. The Low Periodic and High Periodic Objects .....	142
10.3.6.5.1. Data Tab .....	142
10.3.6.5.2. Rendering Properties .....	143
10.3.6.6. The Inlet and Outlet Objects .....	143
10.3.6.6.1. Inlet Tab or Outlet Tab .....	143
10.3.6.6.1.1. Interface Specification Method: Parametric .....	144
10.3.6.6.1.2. Interface Specification Method: Points .....	145
10.3.6.6.1.2.1. Automatically generate required intermediate points .....	145
10.3.6.6.1.2.2. Curve .....	145
10.3.6.6.1.2.3. How to Select and Move a Point .....	147
10.3.6.6.1.3. Interface Specification Method: Adjacent blade .....	147
10.3.6.6.1.4. Interface Specification Method: Fully extend .....	147
10.3.6.6.1.5. Interface Specification Method: Meridional splitter .....	147
10.3.6.6.1.5.1. Interface parametric locations .....	147
10.3.6.6.1.6. Point Visibility .....	147
10.3.6.6.1.7. Control Angle .....	147
10.3.6.6.2. Trim Inlet Tab or Trim Outlet Tab - Changing the Inlet or Outlet Boundary .....	148
10.3.6.6.3. Using Stage Interfaces .....	149
10.3.6.7. The Outline Object .....	149
10.4. Topology .....	149
10.4.1. About Topology .....	150
10.4.2. Basic Usage .....	150
10.4.3. The Topology Set Object .....	152
10.4.3.1. Definition Tab for the Topology Set Object .....	152
10.4.3.1.1. ATM Topology .....	153
10.4.3.1.1.1. Manually Selecting a Topology Family .....	153
10.4.3.1.1.2. Advanced Topology Control .....	154
10.4.3.1.2. Tip Topology Option .....	155
10.4.3.1.3. Split Mesh Regions .....	156
10.4.3.1.4. Mesh Generation Mode .....	157

10.4.3.2. Details Tab for the Topology Set Object .....	157
10.4.4. Using Splitter Blades with ATM .....	158
10.4.5. Using Tandem Vanes with ATM .....	159
10.4.5.1. Choosing the Appropriate TandemVaneAligned Template .....	160
10.4.5.2. The TandemVanCustomizedNr1 Template .....	163
10.4.6. Advanced Local Refinement Control .....	163
10.4.7. Span Location for Controlling Topology .....	164
10.4.8. ATM3D .....	165
10.4.9. ATM v2 .....	166
10.5. Mesh Data .....	166
10.5.1. The Mesh Data Objects .....	166
10.5.1.1. Mesh Size Tab .....	167
10.5.1.1.1. Lock Mesh Size Check Box .....	167
10.5.1.1.2. Method .....	167
10.5.1.1.3. Boundary Layer Refinement Control .....	168
10.5.1.1.3.1. Constant First Element Offset .....	170
10.5.1.1.3.2. Cutoff Edge To Boundary Layer .....	170
10.5.1.1.3.3. Cutoff Edge Split Factor .....	171
10.5.1.1.3.4. Target Maximum Expansion Rate .....	171
10.5.1.1.3.5. Near Wall Element Size Specification .....	171
10.5.1.1.3.5.1. Y Plus .....	172
10.5.1.1.3.5.2. Absolute .....	172
10.5.1.1.4. Five-Edge Vertex Mesh Size Reduction .....	172
10.5.1.1.5. Inlet Domain and Outlet Domain Check Boxes .....	173
10.5.1.2. Passage Tab .....	173
10.5.1.2.1. Spanwise Blade Distribution Parameters .....	173
10.5.1.3. Hub Tip/Complex Blade End and Shroud Tip/Complex Blade End Tabs .....	173
10.5.1.3.1. Hub Tip Distribution Parameters and Shroud Tip Distribution Parameters .....	174
10.5.1.3.2. Blade Tip Settings .....	174
10.5.1.3.2.1. Blades with two rounded edges .....	174
10.5.1.3.2.2. Blades with one rounded edge and one cut-off edge .....	175
10.5.1.3.2.3. Blades with two cut-off edges .....	175
10.5.1.3.3. Spanwise Distribution Parameters (for Complex Blade Ends) .....	175
10.5.1.4. Distribution Settings in General .....	176
10.5.1.5. Inlet and Outlet Tabs .....	178
10.5.1.5.1. Inlet Domain and Outlet Domain Settings .....	178
10.5.1.6. Mesh Around Blade Tab .....	182
10.5.1.6.1. Tip Centerline Location .....	182
10.5.1.7. Complex Blade End Mesh Data Settings .....	182
10.5.2. Changing the Number of Elements on a Selected Master Topology Edge .....	182
10.6. Layers .....	183
10.6.1. Adding Layers .....	183
10.6.2. Deleting Layers .....	183
10.6.3. Editing the Settings of Layers .....	183
10.6.4. Layer Visibility .....	183
10.6.5. The Layers Object .....	184
10.6.5.1. Layers Tab .....	184
10.6.5.1.1. Defining Intermediate Layers .....	184
10.6.5.1.2. Count .....	185
10.6.5.1.3. New Layer .....	185
10.6.5.1.4. Delete Selected Layers .....	185

10.6.5.1.5. Inserting Layers Automatically .....	185
10.6.5.1.6. Inserting Layer After Selected Layer .....	186
10.6.5.1.7. Span Location .....	186
10.6.5.1.8. Spanwise Mesh Interpolation Guide Curves .....	186
10.6.6. Layer Objects .....	187
10.6.6.1. Data Tab .....	187
10.6.6.1.1. Master Topology Visibility .....	187
10.6.6.1.2. Topology Visibility .....	187
10.6.6.1.3. Refined Mesh Visibility .....	187
10.7. 3D Mesh .....	188
10.7.1. The 3D Mesh Object .....	188
10.7.2. Surface Group and Turbo Surface Objects .....	189
10.7.2.1. 3D Mesh Turbo Surfaces .....	189
10.7.2.2. 3D Mesh Surface Groups .....	189
10.8. Mesh Analysis .....	189
10.8.1. Mesh Statistics .....	190
10.8.2. Mesh Limits .....	191
10.8.2.1. Maximum Face Angle .....	192
10.8.2.2. Minimum Face Angle .....	192
10.8.2.3. Connectivity Number .....	192
10.8.2.4. Element Volume Ratio .....	193
10.8.2.5. Minimum Volume .....	193
10.8.2.6. Edge Length Ratio .....	193
10.8.2.7. Skewness .....	193
10.8.2.8. Orthogonality Angle .....	195
10.8.3. Mesh Statistics Parameters - Order Of Importance .....	195
10.8.4. Volume .....	195
10.9. User Defined Objects .....	195
10.10. Default Instance Transform .....	195
10.11. Shortcut Menu Commands .....	196
10.11.1. Assign [Family   Families   Geometry Only] To Topology Instance .....	197
10.11.2. Auto Add Layers and Insert Layers Automatically Commands .....	197
10.11.3. Clear Selected Entities Command .....	197
10.11.4. Color Command .....	198
10.11.5. Create Entity and Assign Family .....	198
10.11.6. Create Mesh Command .....	198
10.11.7. Create New View Command .....	198
10.11.8. Delete Complex Blade End Command .....	198
10.11.9. Delete New View Command .....	198
10.11.10. Delete Secondary Passage Command .....	198
10.11.11. Delete Selected Entity Command .....	198
10.11.12. Edit Blade Topology Command .....	198
10.11.13. Edit Command .....	198
10.11.14. Edit Complex Blade End Topology Command .....	199
10.11.15. Edit in Command Editor Command .....	199
10.11.16. Edit Render Properties Command and Edit Render Properties For All Entities Command ...	199
10.11.17. Fit View Command .....	199
10.11.18. Hide Command .....	199
10.11.19. Insert Blade CAD Feature [feature object name] ( <b>Mesh</b> Workspace) .....	199
10.11.20. Insert Blade Command ( <b>Mesh</b> Workspace) .....	200
10.11.21. Insert Blade Command ( <b>Geometry</b> Workspace) .....	200

10.11.22. Insert Complex Blade End .....	200
10.11.23. Insert Edge Split Control Command .....	200
10.11.24. Insert Hub Complex Blade End Command .....	200
10.11.25. Insert Hub Tip Line .....	200
10.11.26. Insert Layer After and Insert Layer After Selected Layer Commands .....	200
10.11.27. Insert Layer Automatically Command .....	200
10.11.28. Insert Secondary Passage Boundary Command .....	201
10.11.29. Insert Secondary Passage Command .....	201
10.11.30. Insert Shroud Complex Blade End Command .....	201
10.11.31. Insert Shroud Tip Line .....	201
10.11.32. Insert USER DEFINED Object Command .....	201
10.11.33. Predefined Camera Commands .....	201
10.11.34. Projection Commands .....	201
10.11.34.1. Orthographic Command .....	201
10.11.34.2. Perspective Command .....	201
10.11.35. Render Properties Edit Options Command .....	201
10.11.36. Render Properties Show Curves Command .....	202
10.11.37. Render Properties Show Surfaces Command .....	203
10.11.38. Render Properties Topology and Refined Mesh Visibility Commands .....	204
10.11.39. Save Picture Command .....	204
10.11.40. Show and Hide All Siblings Command .....	204
10.11.41. Show Free Edges .....	204
10.11.42. Show Object and Show Commands .....	204
10.11.43. Show Surface Mesh .....	204
10.11.44. Show Tip Shell Curves .....	204
10.11.45. Show Tip Shell Mesh .....	205
10.11.46. Span Curve Visibility Commands .....	205
10.11.47. Suspend Object Updates Command .....	206
10.11.48. Toggle Axis Visibility Command .....	206
10.11.49. Toggle Ruler Visibility Command .....	206
10.11.50. Transformation Commands and Coordinate Systems .....	206
10.11.50.1. Cartesian .....	206
10.11.50.2. Blade-to-Blade .....	207
10.11.50.3. Meridional .....	207
10.11.50.4. 3D Turbo .....	207
10.11.51. Unassign Selected Geometries Command .....	207
10.11.52. Viewer Options Command .....	208
<b>11. Secondary Flow Paths .....</b>	<b>209</b>
11.1. Creating Geometry for Input to TurboGrid .....	209
11.1.1. Source of Geometry .....	210
11.1.2. Interfaces with the Main Passage .....	210
11.2. Specifying Secondary Flow Path Geometry in TurboGrid's Geometry Workspace .....	210
11.2.1. Associating CAD Objects in TurboGrid's Geometry Workspace .....	211
11.3. Specifying Secondary Flow Path Details in TurboGrid's Mesh Workspace .....	212
11.3.1. Geometry Branch .....	212
11.3.2. Mesh Data Branch .....	213
11.3.2.1. Setting Descriptions for the Secondary Flow Path Object .....	213
11.3.2.2. Setting Descriptions for the Secondary Flow Path's Boundary Object .....	214
11.3.3. 3D Mesh Branch .....	215
11.4. Specifying Secondary Flow Paths By Importing NDF Files .....	215
<b>12. Meshing High-Fidelity Geometry .....</b>	<b>219</b>

---

12.1. General Procedure for Meshing of Low-Fidelity and High-Fidelity Geometry .....	221
12.2. Sources of High-Fidelity Geometry for TurboGrid .....	221
12.3. Associating CAD Objects in TurboGrid's Geometry Workspace .....	222
12.3.1. Complex Blade End .....	223
12.4. Using ATM3D or ATM v2 .....	224
12.5. Working with Meshes for High-Fidelity Geometry in TurboGrid's Mesh Workspace .....	224
12.5.1. Geometry Branch .....	225
12.5.2. Mesh Data Branch .....	225
12.5.2.1. Setting Descriptions for Complex Blade End Mesh Data Objects .....	225
12.5.3. 3D Mesh Branch .....	228
12.6. Analyzing the Mesh .....	229
12.7. Saving the Mesh .....	229



---

## List of Figures

1.1. TurboGrid Interface .....	19
10.1. Object Selector .....	102
10.2. Part of Toolbar .....	103
10.3. Rotation Axis .....	120
10.4. Spanwise Lofting versus Streamwise Lofting .....	128
10.5. "Cut-off or square" Trailing Edge .....	130
10.6. Approximate Blend versus True Blend .....	133
10.7. Blade with Partial Tips .....	135
10.8. Master Topology and Refined Mesh on the Hub Layer for an Axial Compressor Blade .....	152
10.9. Conformal Tip Not Enabled .....	155
10.10. Conformal Tip Enabled .....	156
10.11. Example Template for Dual Splitter Blades .....	159
10.12. Theta Definition Requirement .....	161
10.13. Choosing the Tandem Vane Template .....	162
10.14. Variations in boundary layer region thickness .....	168
10.15. Variables that control the blade boundary layer region distribution .....	169
10.16. Inlet Domain Segments .....	181
10.17. Refined Mesh Showing Areas of Unacceptable Minimum Face Angle .....	188
10.18. Ideal and Skewed Triangles and Quadrilaterals .....	194
10.19. Hub Curve With Direction Arrows .....	202
10.20. Blade Profiles With Direction Arrows .....	203
10.21. Span Curves .....	205
11.1. Secondary Flow Path for a Compressor .....	209
12.1. Inlet Guide Vane with Partial Tip Clearance .....	219
12.2. Turbine Rotor with Squealer Tip .....	220
12.3. Typical Mesh Statistics when Meshing High-Fidelity Geometry .....	229



---

# List of Tables

- 1.1. Icon Overlays and Text Styles ..... 20
- 8.1. Expression Editor Shortcut Menu Commands ..... 92
- 10.1. Topology Template Naming Conventions ..... 154
- 10.2. Mesh Generation Mode Comparison ..... 157
- 10.3. Tandem Vane Templates ..... 159
- 10.4. Skewness Ranges and Element Quality ..... 194



# Chapter 1: Ansys TurboGrid Interface

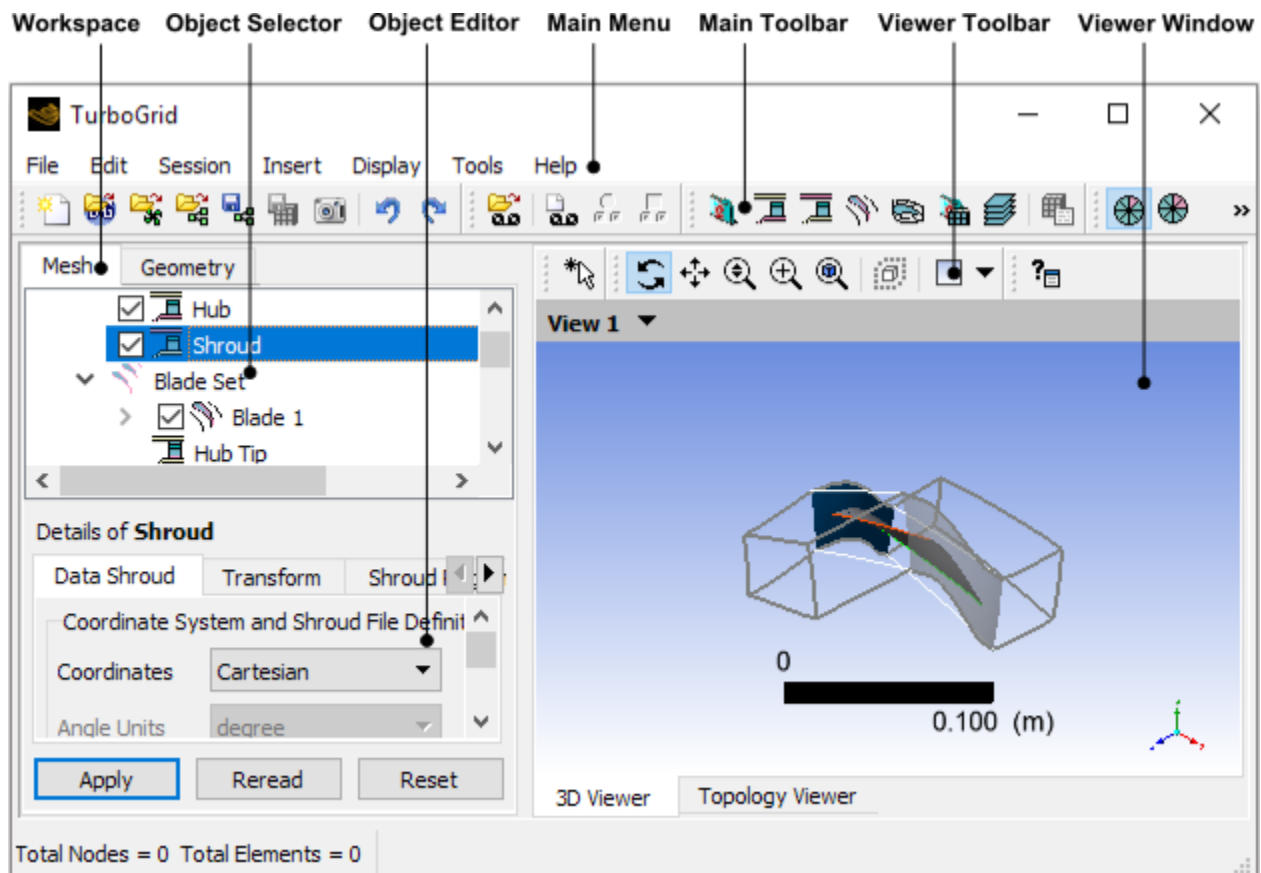
This chapter describes:

- 1.1. Introduction
- 1.2. Object Selector
- 1.3. Object Editor
- 1.4. Mesh Workspace
- 1.5. Geometry Workspace

## 1.1. Introduction

The Ansys TurboGrid interface is divided into several parts, as shown in [Figure 1.1: TurboGrid Interface](#) (p. 19). This chapter describes two main parts of the Ansys TurboGrid interface: the *object selector* and the *object editor*. It also introduces the two workspaces: the **Mesh** workspace and the **Geometry** workspace.

**Figure 1.1: TurboGrid Interface**

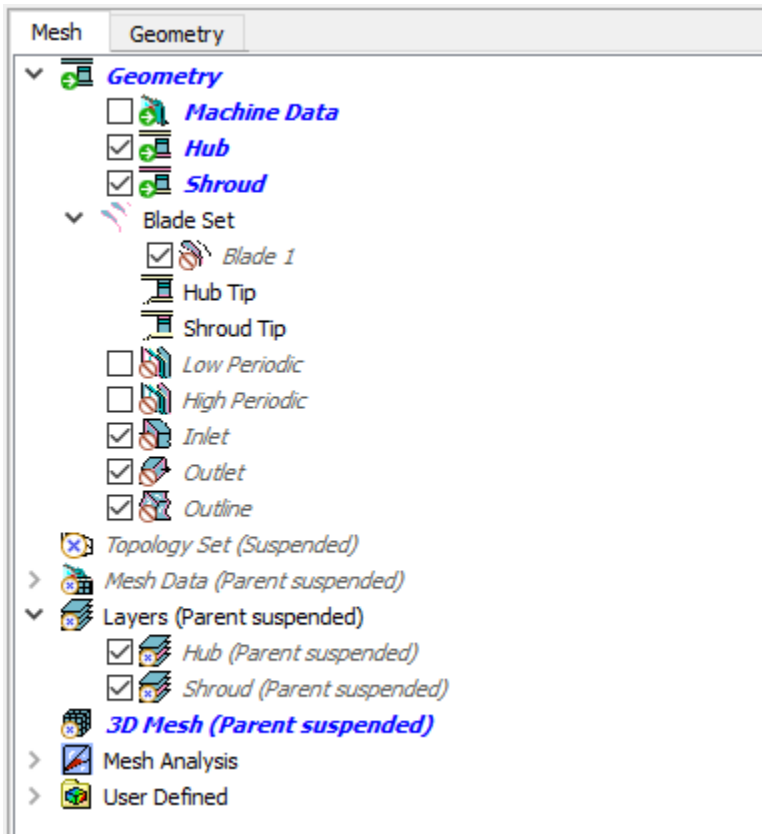


Information about main menu options and the viewer is given in subsequent chapters in this guide.

[Ansys TurboGrid Workflow \(p. 101\)](#) shows how to use Ansys TurboGrid.

## 1.2. Object Selector



The **Mesh** and **Geometry** workspaces each contain an object selector, which is a tree that lists objects in Ansys TurboGrid.






An object selector initially contains some objects in a tree format. *Objects* are data items used to drive aspects of mesh generation, visualization, and calculation.

The objects in an object selector are listed with special icons and text fonts that carry meanings as described in [Table 1.1: Icon Overlays and Text Styles \(p. 20\)](#).

**Table 1.1: Icon Overlays and Text Styles**

Icon Overlay	Font	Appended Phrase	Description
	Grey Italic		The object cannot be processed. Some other object(s) must be defined before this object can be processed.
	Blue Bold Italic		The object is ready to be defined, if applicable, and then processed. The object must be

Icon Overlay	Font	Appended Phrase	Description
			processed before a mesh can be created.
	Black		The object is complete and requires no more information before a mesh can be generated. The object can, however, be edited.
	Red Bold Italic	(Error)	The object has a problem. In the case of the <code>Mesh Analysis</code> object, a red font indicates that at least one mesh statistic falls outside the limits set in the <code>Mesh Limits</code> object.
		(Suspended)	The object will not be processed because it is suspended. You can control whether such an object is suspended from the shortcut menu. For details, see <a href="#">Suspend Object Updates Command</a> (p. 206).
		(Parent suspended)	The object is suspended because a parent object is suspended. For example, whenever the <code>Topology Set</code> object is suspended, the <code>Mesh Data</code> object will also be suspended.

The object selector reflects the structure of the object definitions. For example, in the **Mesh** workspace, there is a `Hub` object in both the `Geometry` and `3D Mesh` branches. To select the geometry `Hub` object, select the `Hub` object from the `Geometry` branch.

You can open an object editor for any object by:

- Double-clicking it.

You may need to expand a tree branch to reach a particular object. This is accomplished by clicking on the plus symbol at the root of the branch.

- Right-clicking the object and using the shortcut menu.

Shortcut menu items will be available according to the type of object. All shortcut menu commands are described in [Shortcut Menu Commands](#) (p. 196).

An alternative way to edit an object is by using the **Command Editor** dialog box. The **Edit in Command Editor** menu item is available by right-clicking an object in the object selector. This operation opens


the **Command Editor** dialog box and displays the definition of the object and its parameter settings. Edit the CCL to change the object. For further details, see [Command Editor Command \(p. 96\)](#).

### 1.2.1. Visibility Check Box

In an object selector, each object that can be displayed in the viewer window has a check box to the left of it. The check box controls the visibility of the object in the viewer. Selecting the check box turns on visibility for that object, while clearing the check box turns off the visibility. For details, see [Common Options \(p. 23\)](#).

## 1.3. Object Editor

An object editor is used to define or edit the properties of an object. It contains a set of one or more tabs that depend on the type of object being edited.

Many properties can be set via a CEL expression. To enter an expression, click in the box for a property, then click the *Enter Expression*  icon that appears beside the field, then do one of the following:


- Enter an expression definition directly.
- Type the name of an existing expression.


(This is a special case of the first point.)


You must ensure that the expression evaluates to a value having appropriate units for the property that uses the expression.


For details on CEL expressions, see [Ansys CFX Expression Language in the TurboGrid Reference Guide](#).



When using Ansys TurboGrid in Ansys Workbench, you can set any property that accepts a CEL expression via a parameterized CEL expression with the parameterized CEL expression being defined by a Workbench input parameter. You can do this in either of the following ways:

- Create an expression and parameterize it using the **Expression Editor** dialog box, then use that expression as the value of the property. For details, see [Expression Editor Dialog Box \(p. 92\)](#).
- In the object editor, click the *Set to Workbench input parameter*  icon (which appears to the right of the property) and then, in the **Enter Parameter Name** dialog box, either enter a unique name for a new Workbench input parameter (a suggested name is entered by default) or select an existing Workbench input parameter.

If you make changes in the object editor, be sure to apply them (click **Apply**) before clicking the *Set to Workbench input parameter*  icon, otherwise your changes will be lost.

If you cannot see the *Set to Workbench input parameter*  icon, click the user interface away from the property that you are trying to parameterize.

You can click *Set to value*  to stop a property from being specified by a Workbench input parameter. However, the corresponding CEL expression persists, and can be managed by the **Expression Editor** dialog box. For details, see [Expression Editor Dialog Box \(p. 92\)](#).

A property that accepts only an integer value is also capable of being controlled by a Workbench input parameter. For such a property, the *Set to Workbench input parameter*  and *Set to value*  icons are used in the same way as for a property that accepts a CEL expression. Integer-value properties cannot take expressions. (No CEL expression is involved in transferring the Workbench input parameter's value from Ansys Workbench to Ansys TurboGrid, for a property that accepts only an integer value.)

### 1.3.1. Common Options

The options located at the bottom of the object editor are available from any of the tabs, and are common to most objects. A description of each option follows:

- The **Apply** button saves the changes made to all the tabs and, if applicable, updates the object in the viewer. Objects that depend on the edited object are also updated.
- The **Reset** button returns the settings for an object to those stored in a database for all the tabs. The settings are stored in the database each time you click **Apply**.
- The **Defaults** button restores the system default settings for all the tabs of an object. The system defaults are stored for each parameter, without regard for the types of object(s) in which the parameter may be used. For this reason, using the **Defaults** button may result in unsuitable changes to an object's settings, and is not recommended.


### 1.3.2. Data Tabs

Most objects have one or more tabs that define the object. Such tabs display the definition of the object currently being edited.


### 1.3.3. Color Tab

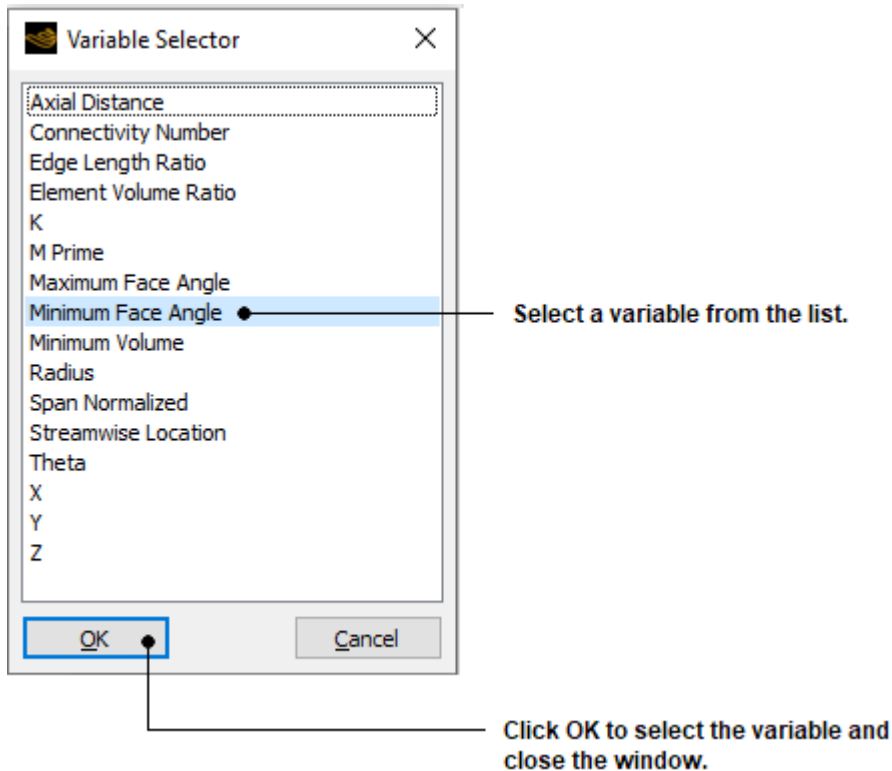
#### 1.3.3.1. Mode

The **Color** tab controls the color of objects in the viewer. The color can be either constant or based on a variable. Select one of the two options for **Mode**.


- Select **Constant** to specify a single color for an object. To choose a color, click the  icon to the right of the **Color** box.
- Select **Variable** to plot a variable on an object (maximum face angle on a plane, for example).
- Select **Use Plot Variable** (available for some plots, including an isosurface) to color an object by the same variable used to define it.



### 1.3.3.2. Variable

Choose the variable to plot from the **Variable** drop-down list. The drop-down list of variables contains the most commonly used variables. For a full list of variables, click the  icon.



### 1.3.3.3. Range


Click  next to the **Range** box to see the available methods for defining the range of the variable used to define the plot. This affects the variation of color used when plotting the object in the viewer. The lowest values of a variable in the selected range are shown in blue in the viewer, the highest values are shown in red.

- **Global** uses the range of the variable over all domains (regardless of the domains selected on the **Geometry** tab) to determine the minimum and maximum values.
- **Local** uses the range of the variable over the current object to determine the minimum and maximum values. This option is useful for utilizing the full color range on the object.
- Using **User Specified**, enter the minimum and maximum values for the contours. This option is useful to concentrate the full color range in a specific variable range. The variable values can be typed in, set using the embedded slider or, by clicking the  icon to the right of the **Units** box, entered as an expression. Click  in the box to the right of the variable value to see the available units for the variable(s).

### 1.3.3.4. Hybrid and Conservative Values

Select whether the plotted object is based on hybrid or conservative values of the variable used for coloring.

### 1.3.3.5. Undefined Color

Any areas in which the variable is not defined (when a section of an object lies outside of the computational domain, for example) use the color specified in the **Undef. Color** box. Click the  icon to the right of this box, to change the undefined color.

## 1.3.4. Render Tab

The exact appearance of the **Render** tab depends on the type of object plotted in the viewer window.

### 1.3.4.1. Draw Faces

If the **Draw Faces** check box is selected, the faces that make up an object are drawn. The faces are colored using the settings on the **Color** tab.

To change the transparency of an object, type in the transparency value or use the embedded slider (which has a maximum value of 1 and a minimum value of 0). A transparency of 0 means the shading is opaque (or having no transparency) and a transparency of 1 means the shading is invisible (completely transparent).

**Shading** properties can be changed to either `None`, `Flat Shading` or `Smooth Shading`.

- Select `None` so that no shading is applied to the object; it appears black.
- Select `Flat Shading` so that each rendered element is colored a constant color. Color interpolation is not used across or between rendered elements.
- Select `Smooth Shading` so that color interpolation is applied that results in color variation across a rendered element based on the color of surrounding rendered elements.

Lighting can be turned on and off by selecting/clearing the **Lighting** check box.

Specular lighting can be turned on and off by selecting/clearing the **Specular** check box. When selected, objects appear to reflect light.

Face culling turns off visibility of rendered element faces of objects that either face the viewer or point away from the viewer. Domain boundaries always have a normal vector that points out of the domain. **Face Culling** options are:

- `Front Faces`

Selecting `Front Faces` turns off visibility of all outward-facing rendered element faces (the faces on the same side as the normal vector). This would, for example, turn off visibility of one side of a plane or the outward facing rendered elements of a cylinder locator. When applied to a volume object, the first layer of rendered element faces that point outwards are rendered invisible.


- `Back Faces`

Selecting `Back Faces` turns off visibility of inward-facing rendered element faces (the faces on the opposite side to the normal vector). When applied to volume objects, the effect of back culling is not always visible in the viewer, since the object-rendered elements that face the outward direction obscure the culled faces. It can, however, reduce the render time when further actions are performed on the object. The effect of this would be most noticeable for large volume objects. In the same way as for front face culling, it turns off visibility of one side of surface locators.

- `No Culling`

`No Culling` turns on the visibility of all rendered element faces.

### 1.3.4.2. Draw Lines

If the **Draw Lines** check box is selected, the lines that make up the object's surface are drawn. To change the line width, type in the line-width value, increase or decrease the value by 1 by clicking the up and down arrows, or use the embedded slider (which has a maximum value of 10 and a minimum value of 1). Line color can be changed by clicking on the  icon to the right of the **Color** box.

The **Edge Angle** setting is used to limit the number of visible edges in a plot. The edge angle is considered to be the angle between two faceted faces of a surface that are connected by an edge. If the angle between two adjacent faces is greater than the **Edge Angle** setting, then the edge shared by the faces is drawn. If the edge angle is 0°, the entire surface is drawn. If the edge angle is large, then only the most significant corner edges of the surface are drawn.

A sensible setting for **Edge Angle** depends on the geometry. Experiment to get a value that clearly shows where the surface is located, without displaying too much of the surface mesh. Too many lines can make it confusing when more objects are added to the geometry.

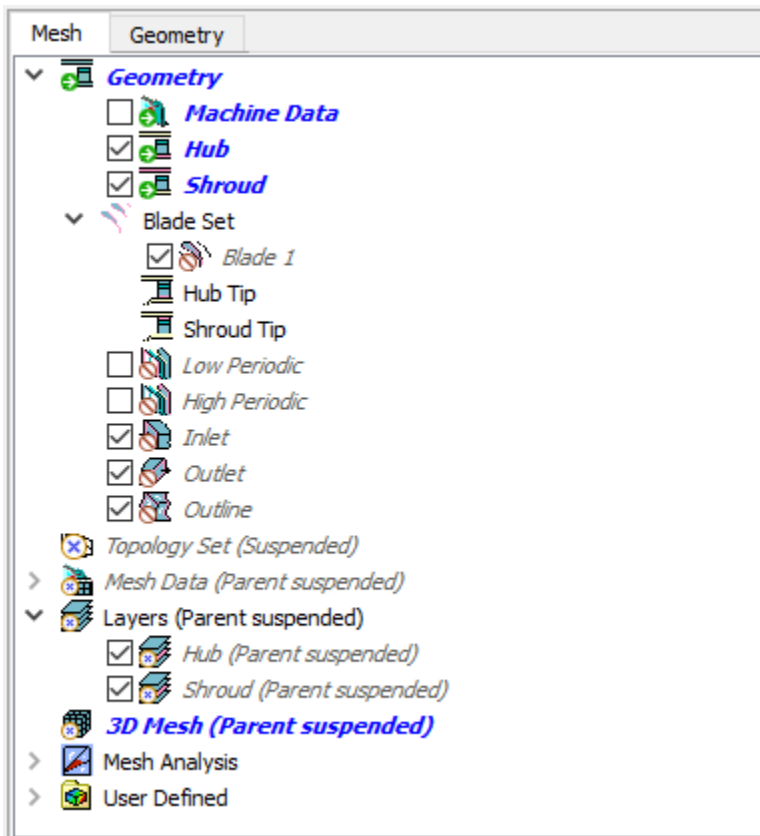
Setting an edge angle defines a minimum angle for drawing parts of the surface mesh. For example, if an edge angle of 30° is chosen, any edges shared by faces with an angle between them of 30° or more are drawn.

Reducing the edge angle shows more of the surface mesh in the viewer. When the edge angle is 0°, all of the surface mesh is shown.

### 1.3.4.3. Applying Instance Transforms to Objects

Instance transforms are created separately using **Insert > User Defined > Instance Transform**. For details, see [Instance Transform Command \(p. 73\)](#). The **Apply Instancing** check box is selected by default, and `Default Transform` is selected. To apply a different transform, it must be created and then selected from the list of existing instance transforms.

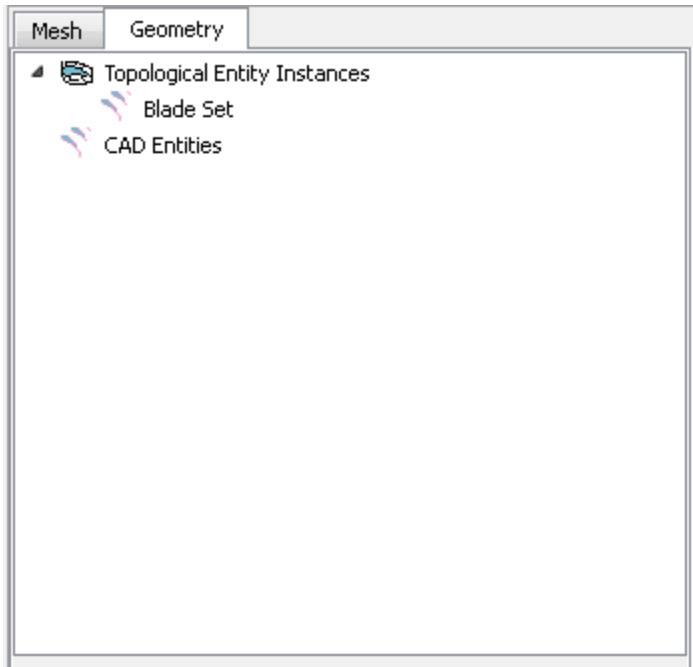
## 1.4. Mesh Workspace



The **Mesh** workspace holds the settings that TurboGrid needs to create a mesh.

In the **Mesh** workspace, the object selector shows objects that are used to drive aspects of mesh generation. These objects must be defined in a certain order. For details, see [Steps to Create a Mesh \(p. 101\)](#).

## 1.5. Geometry Workspace



The **Geometry** workspace can be used to:

- Select between using CAD geometry or profile point data as a geometry source. This selection is made by the **Input Mode** setting in the geometry browser, which is the main object editor of the **Geometry** workspace. To access the geometry browser, double-click a high-level object (either the **Topological Entity Instances** object or the **CAD Families** object) in the object selector.
- Using the object selector, object editors, and viewer, verify the import of CAD geometry from an upstream (BladeEditor) Geometry cell in Workbench, or manually import CAD data from another source. For details, see [Defining Geometry from a CAD Source \(p. 103\)](#).
- Specify a source of profile point geometry data (for example, an \*.inf file with associated curve files). For details, see [Defining Geometry from Profile Points \(p. 108\)](#).

---

# Chapter 2: File Menu

---

The following sections describe the commands available in the **File** menu:

- 2.1. Introduction
- 2.2. New Case Command
- 2.3. Load TurboGrid Init File Command
- 2.4. Load Profile Points Or CAD Command
- 2.5. Import NDF File Command
- 2.6. Update Geometry Command
- 2.7. Load State Command (Import State Command)
- 2.8. Save State Command (Export State Command)
- 2.9. Save State As Command
- 2.10. Save Project
- 2.11. Refresh
- 2.12. Save Submenu
- 2.13. Save Mesh Command
- 2.14. Save Mesh As Command
- 2.15. Export Submenu
- 2.16. Save Picture Command
- 2.17. Recent State Files Submenu
- 2.18. Recent Session Files Submenu
- 2.19. Quit Command

## 2.1. Introduction

---

Ansys TurboGrid uses and produces the following file types:

### Session File

.tse session files are produced by TurboGrid and contain CCL commands. Session files record the commands executed to a file for playback at a later date. Use session files to run TurboGrid in batch mode. See [Session Menu \(p. 55\)](#) and [Batch Mode in the TurboGrid Reference Guide](#) for details.

---

#### Note:

Because the session file is a text file of CCL commands, you can write your own session files using a text editor.

---

## State File

.tst state files are produced by TurboGrid and contain CCL commands. They differ from session files in that only a snap-shot of the current state is saved to a file. Using state files, you can close TurboGrid and continue working later from the same point. See [Load State Command \(Import State Command\) \(p. 34\)](#) and [Save State Command \(Export State Command\) \(p. 35\)](#) for details.

---

**Note:**

Because the state file is a text file of CCL commands, you can write your own state files using a text editor.

---

---

**Note:**

State files previously had the extension .cst.

---

## Topology File

.tgt topology files are produced by TurboGrid and define the topology. Using topology files, you can use the same topology for various cases without having to redefine it each time.

---

**Note:**

Because the topology file is a text file, you can write your own topology files using a text editor.

---

## Mesh File

.gtm mesh files are produced by TurboGrid and contain the mesh in a format which can be read by Ansys CFX. For details, see [Save Mesh Command \(p. 38\)](#).

## BladeGen.inf File

BladeGen.inf information files contain:

- Machine data (the rotation axis, number of blade sets, length unit)
- Names of geometry definition curve files for the hub, shroud, and blades
- Leading and trailing edge settings

The \*.inf file is structured as follows:

```
!===== CFX-BladeGen Export =====
Axis of Rotation: Z
Number of Blade Sets: 9
Number of Blades Per Set: 2
Geometry Units: MM <---- Unknown|IN|MM|FT|MI|M|KM|MIL|UM|CM|UIN
Blade 0 LE: EllipseEnd <---- EllipseEnd|CutoffEnd|SquareEnd
Blade 0 TE: CutOffEnd
Blade 1 LE: EllipseEnd
```

```
Blade 1 TE: CutOffEnd
Hub Data File: hub3.curve
Shroud Data File: shroud3.curve
Profile Data File: profile3.curve
```

(The statements following "<----" are comments that show possible values. They are not part of the format.)

For information about loading \*.inf files, see [Load TurboGrid Init File Command \(p. 31\)](#).

## Curve File

.crv and .curve curve files are used by TurboGrid to define machine geometry. These files contain points in free-format ASCII style and can be created:

- Using a text editor
- Using Ansys BladeGen
- Using the ExportPoints feature of BladeEditor with feature property "Export to file" = "Yes"
- By saving a modified blade geometry in TurboGrid.


## Tetin File

.tin Tetin files are produced by TurboGrid and describe the geometry in a format which can be read by ICEM CFD products. For details, see [Export Geometry Command \(p. 41\)](#).


## Picture Files

PNG (.png), AVZ (3D) (.avz), JPEG (.jpg), PPM (.ppm), PostScript (.ps), and Encapsulated PS (.eps) files can be saved. For details, see [Save Picture Command \(p. 42\)](#).

## 2.2. New Case Command

To display the object selector and begin a new TurboGrid case, select **File > New Case** from the main menu or click *New Case* .

## 2.3. Load TurboGrid Init File Command

To load a BladeGen \*.inf file or TurboGrid \*.tginit initialization file, select **File > Load TurboGrid Init File** or click *Load TurboGrid Init File* . The **Open TurboGrid Initialization File** dialog box is displayed.

For more information, see:

[2.3.1. BladeGen \(\\*.inf\) File Details](#)

[2.3.2. TurboGrid Init \(\\*.tginit\) File Details](#)

### 2.3.1. BladeGen (\*.inf) File Details

If the `inf` file does not specify the curve files, you may turn on an option in the dialog box (called **Guess missing curve files**) in order to guess the names of the missing curve files. The algorithm for guessing curve files looks for files in the working directory with a `.curve` or `.crv` extension, with "hub", "shroud", or "profile" in the name (using a case-insensitive search). If the curve filenames are unusual, you should verify that the correct curve files were selected by opening the appropriate Geometry object (for example, the Hub object) in the **Mesh** workspace.

If a `BladeGen.inf` file specifies multiple blades in the blade set, multiple blades will be generated.

After loading a `BladeGen.inf` file, the **Curve Type** settings for the Hub and Shroud geometry objects (in the **Mesh** workspace) are set to `Piece-wise Linear` instead of the default: `Bspline`.

### 2.3.2. TurboGrid Init (\*.tginit) File Details

When opening a TurboGrid Initialization file using the **Open TurboGrid Initialization File** dialog box:

- Select a blade row to load under **Select a Blade Row**.
- If in stand-alone mode, select zero or more secondary flow paths to load under **Secondary Flowpaths to Import**.

## 2.4. Load Profile Points Or CAD Command

---

To load new geometry curves, select **File > Load Profile Points Or CAD** from the main menu or click

*Load Profile Points Or CAD* . The geometry browser is displayed with **Input Mode** set to `Profile Points`. For details, see [Defining Geometry from Profile Points \(p. 108\)](#).

## 2.5. Import NDF File Command

---

To use an NDF file as a geometry source, use the **Import NDF File** command.

---

#### Note:

The **Import NDF File** command requires:

- A BladeModeler or BladeBuilder license, depending on the selected **CAD Generator**.
- 

To use the **Import NDF File** command:

1. Select **File > Import NDF File**.

The **Import NDF** dialog box appears.

2. Ensure that **CAD Generator** is set appropriately.

The **CAD Generator** setting specifies which tool is used to generate CAD geometry for TurboGrid. The default selection is controlled by preference **Default CAD Generation Tool** (see [TurboGrid Options](#) (p. 48)).

If the NDF file cannot be opened using the selected **CAD Generator**, a notice will appear and the **OK** and **Apply** buttons will be disabled.

3. Set **NDF File Name** to the name of the NDF file containing the geometry.

Optionally click the *Browse*  icon beside **NDF File Name** to use the **Import NDF File** dialog box to select an NDF file.

4. Optionally change **Generated CAD Definition > File Name**, which specifies the planned name of the CAD definition file (Parasolid file with extension ".x\_b") to generate.

If the specified CAD definition file already exists, it will be overwritten by the CAD definition that is generated upon importing the specified NDF file.

5. Under **Select a Blade Row**, select a blade row to load from the CAD definition file after the latter is generated.
6. Click **OK** or **Apply** to generate the CAD definition based on the specified NDF file and then load the selected blade row.

Generation (or regeneration) of the CAD geometry involves running BladeEditor or BladeBuilder temporarily in the background to generate the CAD file and associated TurboGrid Initialization file, then loading those two files into TurboGrid, resulting in new/updated CAD objects and (in accordance with the contents of the TurboGrid Initialization file) new/updated topological entities in the **Geometry** workspace tree.

After importing an NDF file, the selected blade row, as well as the flow path and any secondary flow paths, are loaded into the current session of TurboGrid. You can load into subsequent sessions of TurboGrid any other blade rows from the same generated CAD definition file by using the **Load TurboGrid Init File** command to load the `.tginit` file. For details, see [Load TurboGrid Init File Command](#) (p. 31).

For more information, see [Defining Geometry from Generated CAD: CAD From NDF](#) (p. 112).

## 2.6. Update Geometry Command

When running outside of Workbench, you can re-load the geometry from its source, or regenerate it from its input data, by selecting **File > Update Geometry**.

Geometry sources include:

- Profile curves
- CAD data

- Blade blend specifications (not available in Workbench) in the **Mesh** workspace tree, under the **Geometry** branch.

---


**Note:**

If you are running in Workbench, for example, if you are updating the files in Ansys Blade-Gen/BladeModeler, and you want to re-load the updated files into TurboGrid, refresh the geometry using Workbench or the **File > Refresh** command in TurboGrid.

---

## 2.7. Load State Command (Import State Command)

---

A previously saved state file can be loaded into TurboGrid. To load a state file in stand-alone mode, select **File > Load State** from the main menu or click *Load State*  on the toolbar; when running in Ansys Workbench, select **File > Import State**. The **Load State File** dialog box is displayed.

If any file specified in a state or session file cannot be found, TurboGrid will automatically search the state or session file directory for a file of the same name. If this search fails, the current working directory will be searched. As a result, state and session files will not have to be edited to change the path when state, session and curve files are moved from one directory to another.

Select the **Load as new case** option button to delete all existing objects and create new objects that are defined in the state file.

Select the **Append to current case** option button to add all objects defined in the state file to the existing objects. Existing objects are not removed unless they have the same name as an object in the state file, in which case they are replaced. Loading a state file in this way allows the use of a number of state files as building blocks for commonly used objects.

---

**Note:**

If you are running TurboGrid in Ansys Workbench, and there is a geometry provided to TurboGrid from an upstream cell, then that geometry is used and the geometry data in the state file is ignored.

---

---

**Note:**

Because the `Topology Set` object is now suspended by default, session and state files from version 11 or earlier might not play or load correctly. To support older session and state files, the **Play Session File** and **Load State File** dialog boxes have an option named **Unsuspend TOPOLOGY SET before loading**. Selecting this option causes the `Topology Set` object to be unsuspended before playing/loading a session/state file. When starting TurboGrid from the command line, adding the command line parameter `-u` causes the `Topology Set` object to be initially unsuspended.

---


## 2.8. Save State Command (Export State Command)

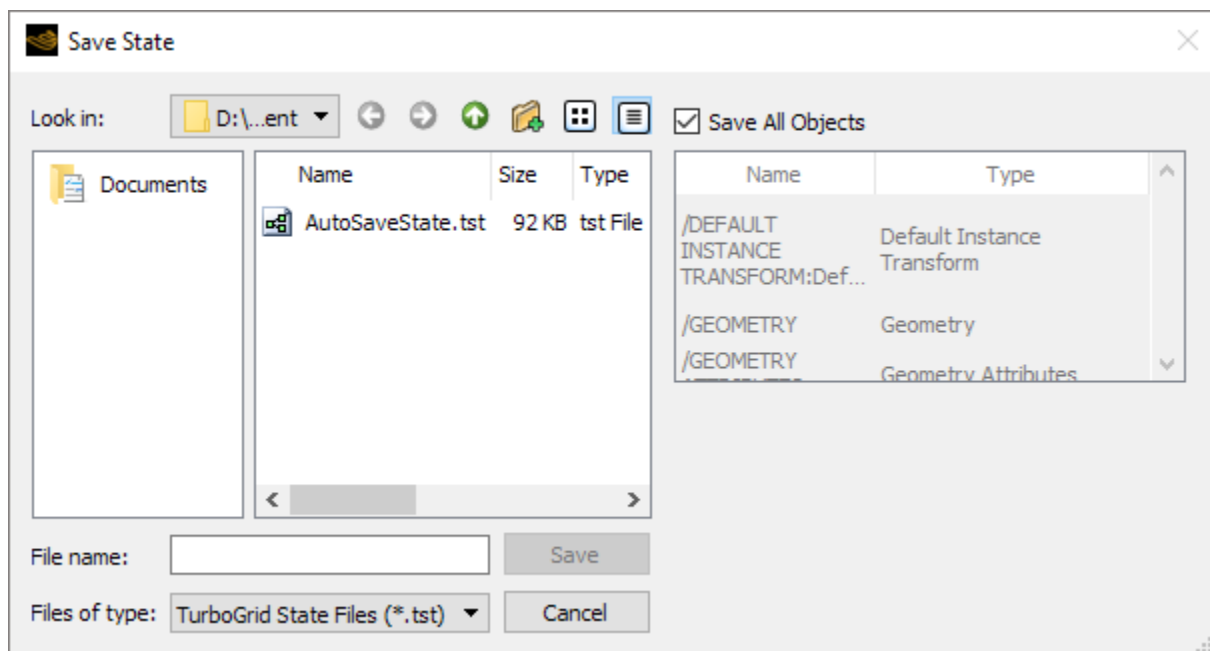
If you have not saved a state file during the current TurboGrid session, selecting **File > Save State** from the main menu (**File > Export State** when running in Ansys Workbench) opens the **Save State** dialog box where you can type a filename for your state file. For details, see [Save State As Command \(p. 35\)](#).

If you have already saved a state file during the current TurboGrid session, selecting **File > Save State** from the main menu overwrites that file. To save a state to a different filename, select **File > Save State As...** from the main menu. For details, see [Save State As Command \(p. 35\)](#).

## 2.9. Save State As Command

Saving a state file produces a text file containing CCL commands for the current TurboGrid state. To save a state to a new file, select **File > Save State As...** from the main menu or click *Save State As...*

. The **Save State** dialog box is displayed.



Set **Look in** to the directory in which you want to create the state file.

In the list on the right side, select the objects to include in the state file, or select **Save All Objects**. If **Save All Objects** is selected, the current state of all objects is written to the state file. If **Save All Objects** is cleared, select the objects to save to the state file by clicking on each object. The current state of all selected objects is written to the state file.

Enter (or select) a filename, then click **Save** to save the state file.

A state file is linked to the geometry files from which it was created by an absolute path; therefore, the location of the geometry files should not be changed. This also applies to topology files if the **From File** option is selected for a Topology Set object.

State files are automatically saved with a `.tst` file extension.

## 2.10. Save Project

---

When running in Ansys Workbench, you can select **File > Save Project** to save the entire Ansys Workbench project.

## 2.11. Refresh

---

When running in Ansys Workbench, you can select **File > Refresh** to refresh the associated Turbo Mesh cell.

## 2.12. Save Submenu

---

Many objects that can be saved or saved under a different name are listed under the **Save** submenu.

[2.12.1. Save Blade Command](#)

[2.12.2. Save Blade As Command](#)

[2.12.3. Save Periodic/Interface Surfaces Command](#)

[2.12.4. Save Inlet Command](#)

[2.12.5. Save Inlet As Command](#)

[2.12.6. Save Outlet Command](#)

[2.12.7. Save Outlet As Command](#)

[2.12.8. Save Topology Command](#)

[2.12.9. Save Topology As Command](#)

### 2.12.1. Save Blade Command

If you have not saved a blade file during the current TurboGrid session, selecting **File > Save > Blade** from the main menu opens the **Save Blade** dialog box where you can type a filename for your blade file. For details, see [Save Blade As Command \(p. 36\)](#).

If you have already saved a blade file during the current TurboGrid session, selecting **File > Save > Blade** from the main menu overwrites that file. To save a blade to a different filename, select **File > Save > Blade As...** from the main menu. For details, see [Save Blade As Command \(p. 36\)](#).

### 2.12.2. Save Blade As Command

Saving a blade file produces a text file defining the current blade, which can then be used to define the blade for future meshes. If the same changes to the blade would otherwise be repeated, this eliminates wasted time. To save a blade to a new file, select **File > Save > Blade As** from the main menu. The **Save Blade** dialog box is displayed.

Blade files are saved with a `.crv` file extension if the file type is All Blade Files (`*.crv` or `*.curve`).

### 2.12.3. Save Periodic/Interface Surfaces Command

Saving the periodic/interface surfaces produces a text file describing the location and shape of the interfaces between adjacent blades. You can optionally set a base name for constructing the data filename. You can optionally change the units in which to save the data.

### 2.12.4. Save Inlet Command

Saving the inlet produces a text file describing the location and shape of the inlet region. This includes any added inlet points.

Inlet files are saved with a `.crv` file extension if the file type is All Inlet Files (`*.crv *.curve`).

### 2.12.5. Save Inlet As Command

Save an inlet (see above) to an alternative filename.

### 2.12.6. Save Outlet Command

Saving the outlet produces a text file describing the location and shape of the outlet region. This includes any added inlet points.

Outlet files are saved with a `.crv` file extension if the file type is All Outlet Files (`*.crv *.curve`).

### 2.12.7. Save Outlet As Command

Save an outlet (see above) to an alternative filename.

### 2.12.8. Save Topology Command

If you have not saved a topology file during the current TurboGrid session, selecting **File > Save > Save Topology** from the main menu opens the Save Topology dialog box where you can type a filename for the topology file. For details, see [Save Topology As Command \(p. 37\)](#).

If you have already saved a topology file during the current TurboGrid session, selecting **File > Save > Save Topology** from the main menu overwrites that file. To save a topology to a different filename, select **File > Save > Save Topology As** from the main menu. For details, see [Save Topology As Command \(p. 37\)](#).

### 2.12.9. Save Topology As Command

Saving a topology file produces a text file defining the current topology. This file can then be used to define the topology for other geometries, which may be necessary for certain analyses. To save a topology to a new file, select **File > Save > Save Topology As** from the main menu or click *Save*

*Topology As* . The **Save Topology** window is displayed.


Topology files are saved with a `.tgt` file extension automatically.

## 2.13. Save Mesh Command

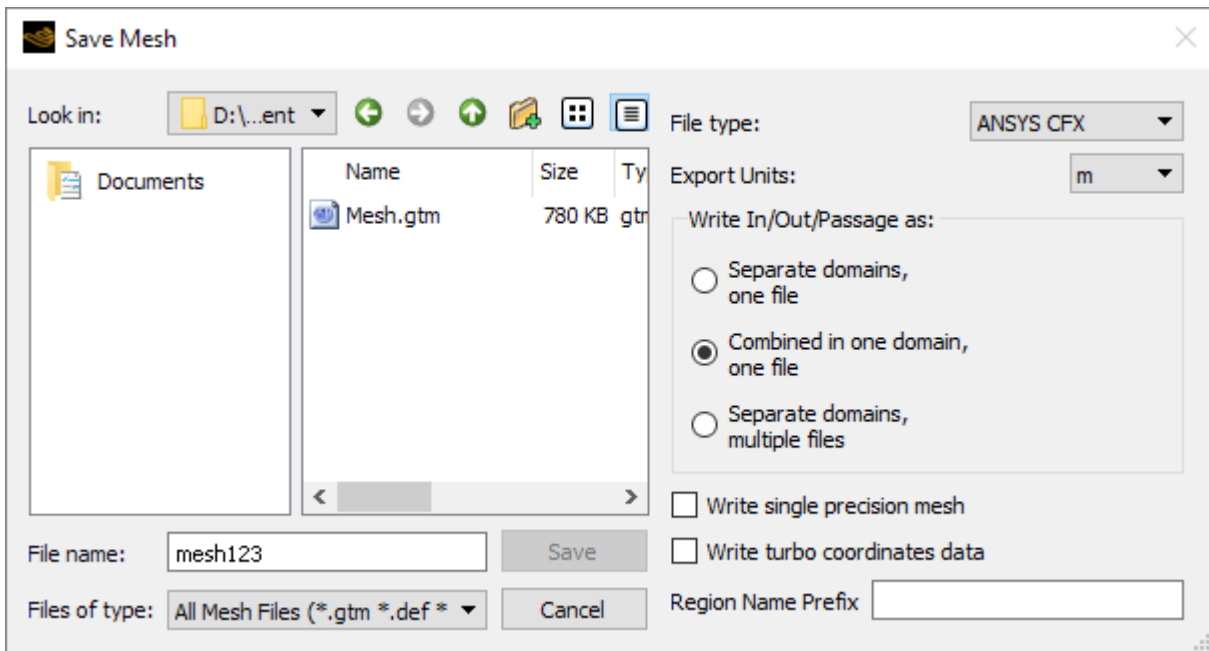
If you have not saved a mesh during the current TurboGrid session, selecting **File > Save Mesh** from the main menu opens the **Save Mesh** dialog box where you can type a filename for the mesh file(s). For details, see [Save Mesh As Command \(p. 38\)](#).

If you have already saved a mesh during the current TurboGrid session, selecting **File > Save Mesh** from the main menu overwrites the file(s). To save a mesh to a different name, select **File > Save Mesh As** from the main menu. For details, see [Save Mesh As Command \(p. 38\)](#).

## 2.14. Save Mesh As Command

Saving a mesh file produces input files for Ansys CFX or produces a CGNS file. To save a mesh to a new file, select **File > Save > Save Mesh As** from the main menu or click *Save Mesh As* . The **Save Mesh** dialog box is displayed.

### 2.14.1. Save Mesh Dialog Box



Set **Look in** to the directory in which you want to create the mesh file.

#### 2.14.1.1. File Type

If **File type** is set to `Ansys CFX`, a file of type `.gtm` is saved by default (that is, if no extension is specified). You can also save a file of type `.def` by adding the `.def` extension to the specified filename. Both `.gtm` and `.def` files contain regions that can be used in CFX-Pre to set up a CFD problem.

If **File type** is set to `CGNS`, TurboGrid saves a CGNS (CFD General Notation System) file (extension `.cgns`) in HDF5 format. CGNS files can be used by Ansys software such as CFX-Pre, CFD-Post, Fluent, and by third party software that supports:

- The features that TurboGrid writes,
- CGNS Version 4.3.0,
- HDF5 1.12.2.

If you change **File type**, any existing file extension (for example, `.def` or `.cgns`) is changed automatically in the specified filename.

### 2.14.1.2. Export Units

Set **Export Units** to the length unit for the exported mesh.

### 2.14.1.3. Write In/Out/Passage As

If **File type** is set to `Ansys CFX`, the following options are available:

- **Separate domains, one file**

The inlet and outlet domains remain separate from the passage domain. Three separate assemblies appear in CFX-Pre. This choice is ideal when you want to place the inlet and outlet domains in a different frame of reference from the passage.

- **Combined in one domain, one file**

The inlet and outlet domains are combined with (merged with) the passage domain. One combined assembly appears in CFX-Pre. This choice is ideal when you want to keep the inlet and outlet domains in the same frame of reference as the passage.

- **Separate domains, multiple files**

Up to three separate files are written: `<basename>.gtm` for the passage domain, `<basename>_Inlet.gtm` for the inlet domain (if applicable), and `<basename>_Outlet.gtm` for the outlet domain (if applicable). Here, `<basename>` represents the specified name and `.gtm` is an example file extension.

### 2.14.1.4. Write Single Precision Mesh

If **File type** is set to `Ansys CFX`, you can select the **Write single precision mesh** check box to cause a single-precision mesh file to be written instead of a double-precision file. The default is double-precision. There is little benefit to using single-precision other than to reduce the size of the mesh file.

### 2.14.1.5. Write Turbo Coordinates Data

If **File type** is set to `Ansys CFX`, you can select the **Write turbo coordinates data** check box to include turbo coordinates data in the saved mesh file.

Turbo coordinates data includes the following coordinates on all blade surfaces:

- U,V (surface coordinates on the blade)
- Streamwise Location, Span Normalized
- Axial Distance, Radius, Theta

When imported into CFX-Pre, turbo coordinates data can be used to define a custom coordinate system. Such a coordinate system can be used to specify coordinates for user locations using turbo coordinates rather than Cartesian or cylindrical coordinates. See [Coordinate System Definition in the CFX-Pre User's Guide](#).

Turbo coordinates data can be extracted from the saved mesh file using `cfx5dfile`. For details, see [Extracting Profile Data from Results Files in the CFX-Solver Modeling Guide](#).

---

**Note:**

- The blade surfaces on which the turbo coordinates are defined usually extend beyond the actual blade.
  - You can export turbo coordinates data to one or more `.csv` files. For details, see [Export Turbo Coordinates Data Command \(p. 41\)](#).
- 

### 2.14.1.6. Region Name Prefix

This property specifies a string of characters that is prefixed to all mesh region names when the mesh is written to file. This property is blank by default.

For information on how CFX-Pre handles duplicate mesh region names, see [Importing Multiple Meshes](#).

---

**Note:**

CGNS mesh region names have a 32-character limit. This limit sometimes causes the mesh region name to be truncated or replaced with a unique name (if necessary) when saved in TurboGrid.

For example,

ABCDEF0123456789ABCDEF0123456789regionA would be truncated to

ABCDEF0123456789ABCDEF0123456789, and

ABCDEF0123456789ABCDEF0123456789regionB would be renamed Boundary 3.

If your mesh region names have been truncated or renamed, you may want to manually rename the affected mesh regions in CFX-Pre or Fluent.

---

## 2.14.2. Region Naming

The regions saved to the `.gtm` and `.def` files are BLADE, PER1, PER2, HUB, SHROUD, INFLOW and OUTFLOW.

If the mesh has an inlet or outlet 3D region, additional regions will be created in the files for these domains with the suffix **Inlet** or **Outlet**. For example, the regions for a case with an inlet 3D region are PER1 Inlet, PER2 Inlet, HUB Inlet, SHROUD Inlet, INFLOW and OUTFLOW Inlet. The same applies for an outlet 3D region and for the Passage Region. For the parts specific to a region (BLADE for Passage, INFLOW for Inlet, and OUTFLOW for Outlet), no suffix is added.

## 2.15. Export Submenu

Items that can be exported are listed under the **Export** submenu.

[2.15.1. Export Geometry Command](#)

[2.15.2. Export Turbo Coordinates Data Command](#)

### 2.15.1. Export Geometry Command

The geometry in TurboGrid can be exported to a tetin file format which can then be read by ICEM CFD products. To export a geometry to a tetin file, select **File > Export > Geometry** from the main menu. The **Export Geometry** dialog box is displayed.

Geometry files are saved with a `.tin` file extension automatically.

### 2.15.2. Export Turbo Coordinates Data Command

You can export turbo coordinates data to one or more `.csv` files.


Turbo coordinates data is described in [Write Turbo Coordinates Data \(p. 39\)](#).

To export turbo coordinates data to one or more `.csv` files:

1. Select **File > Export > Turbo Coordinates Data** from the main menu.
2. Configure the settings in the **Export Turbo Coordinates Data** dialog box.

**File Option** can be set to:


- `One File Per Surface` — Exports one `.csv` file per surface.

Set **Output Folder** to the name of an output folder to which to export. To help set the output folder name, you can click *Browse* .

Set **File Prefix** to provide a prefix to use in naming each exported `.csv` file. You can optionally specify no prefix by leaving the field blank. The name of each exported file always includes the name of the applicable surface.

Select **Overwrite Existing Files** to enable overwriting any existing files of the same name.

- All Surfaces In One File — Exports one .csv file for all surfaces.

Set **File** to the name of the .csv file to export. To help set the file name, you can click *Browse*  and use the **Export** dialog box.

3. Click **Export** to export the file(s).

You can click **Cancel** to cancel the export.

You can view the .csv files as user surfaces in CFD-Post.


---

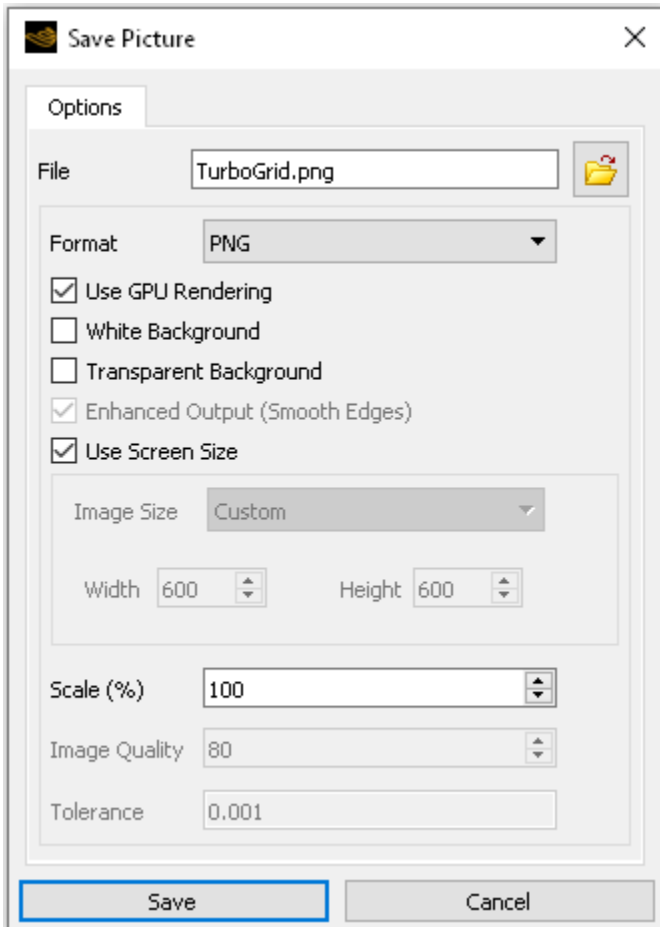
### Note:

You can include turbo coordinates data in a saved mesh file. For details, see [Write Turbo Coordinates Data \(p. 39\)](#).

---

## 2.16. Save Picture Command


To save the current contents of the viewer window to a file, select **File > Save Picture** from the main menu or click *Save Picture* . The **Save Picture** dialog box is displayed.



## Options Tab


The **Options** tab has the following settings:

### File

Enables you to specify the filename of the file. You may enter the filename and path into the **File** field, or click the *Browse*  icon and search for the directory in which the file is to be saved.

Files are always saved with the file extension corresponding to the selected graphics format.

### Format

To choose a file format, click . When creating a new image file, the file format you choose affects the quality of the image:

#### PNG

Portable Network Graphics is a raster file format (\*.png) that supports lossless image compression.

#### AVZ (3D)

Ansys Viewer Format is a file format (\*.avz) used to present interactive three-dimensional views. It can be displayed using the Ansys Viewer.

#### JPEG

A compressed file format (\*.jpg) developed for compressing raw digital information. File sizes are relatively small. Due to compression artifacts, this format is not recommended for line drawings.

#### PPM

Portable Pixel Map is a file format (\*.ppm) with uncompressed data and is not recommended for large images.

#### PostScript and Encapsulated PS

PostScript (\*.ps) and Encapsulated PS (\*.eps) are generally recommended for output to a printer or line drawings. However, some graphics objects and features can cause the PS/EPS to output as a very large bitmap file, in which case a PNG file would be a more efficient alternative. Note that the Ansys logo and the axis do not cause the PS/EPS output to become a bitmap.

---

### Note:

Existing CFD-Viewer State (\*.cvf) files previously generated can be converted by using an application supplied as part of the CFX, TurboGrid and CFD-Post installations:

```
<CFXROOT>\bin\cfx5cvfconvert <cvf-file>
```

### Use GPU Rendering Check Box

If your graphics hardware is compatible with GPU rendering, selecting this option is strongly recommended. You can set the default value for this option using the following preference: **Edit > Options > Common > Viewer Setup > Use GPU Rendering for Printing**.

If **Use GPU Rendering** is selected, GPU rendering is used unless compatible GPU hardware is not found, in which case all saved pictures are instead software rendered (by the CPU).

When you save a picture with GPU rendering, your graphics hardware renders an image that closely matches what is shown in the viewer.

If GPU rendering is not used, software rendering is used instead. Software rendering is relatively slow and does not always render as shown in the viewer. One benefit of software rendering is that it has no graphics hardware requirements.

### White Background Check Box

You can save the current image with a white background by selecting **White Background**.

When **White Background** is selected, certain white objects may be colored black and certain black objects may be colored white in the image file. Objects that are not affected can usually be manually colored by editing them.

---

**Note:**

You can adjust the viewer background colors in the preferences (**Edit > Options > TurboGrid > Viewer**).

---

### Transparent Background Check Box

**Transparent Background** specifies a transparent background for a PNG image.

---

**Note:**

GPU rendering is required to support transparent backgrounds. If playing a session file in batch mode to save PNG images using TurboGrid (or CFX-Pre), transparent backgrounds are disabled because GPU rendering is not available in batch mode (software rendering is used).

---

### Enhanced Output (Smooth Edges) Check Box

---

**Note:**

This setting is available only when using software rendering. Note that software rendering can be slower than GPU rendering and does not correctly render some graphics objects (that is, OpenGL shader graphics). It is recommended that you use GPU rendering whenever possible. You can turn GPU rendering on and off from the preferences (**Edit > Options > Common > Viewer Setup > Use GPU Rendering for Printing**).

---

When **Enhanced Output (Smooth Edges)** is selected, the image is processed by antialiasing.

## Use Screen Size Check Box

When **Use Screen Size** is selected, the output has the same width and height, measured in pixels, as shown in the viewer. You can clear the check box to specify the width and height using other options.

## Image Size

With **Use Screen Size** not selected, you can set **Image Size** to one of the following options:

- HD: 1280 pixels wide and 720 pixels high
- FHD: 1920 pixels wide and 1080 pixels high
- QHD: 2560 pixels wide and 1440 pixels high
- 4k: 3840 pixels wide and 2160 pixels high
- 8k: 7680 pixels wide and 4320 pixels high
- Custom: Set **Width** and **Height** in pixels.

## Width/Height

With **Use Screen Size** not selected, and with **Image Size** set to `Custom`, you can specify the width and height of the image in pixels by entering values for **Width** and **Height**.

## Scale (%)

**Scale (%)** is used to scale the size of bitmap images to a fraction (in percent) of the current viewer window size. This option is available when **Use Screen Size** is selected.

## Image Quality

**Image Quality** is available only for the JPEG format. A value of 100 specifies the highest image quality; a value of 1 specifies the lowest image quality.

## Tolerance

---

### Note:

This setting is available only when using software rendering. Note that software rendering can be slower than GPU rendering and does not correctly render some graphics objects (that is, OpenGL shader graphics). It is recommended that you use GPU rendering whenever possible. You can turn GPU rendering on and off from the preferences (**Edit** > **Options** > **Common** > **Viewer Setup** > **Use GPU Rendering for Printing**).

---

**Tolerance** is a non-dimensional value used in face sorting when generating pictures. Larger values result in faster generation times, but may cause defects in the resulting output.

Click **Save** to save the current viewer contents to an image file.

## 2.17. Recent State Files Submenu

---

TurboGrid saves the file paths of the last several state files opened. To re-open a recently used state file, select **File > Recent State Files** from the main menu and then select the file from the **Recent State Files** submenu.

## 2.18. Recent Session Files Submenu

---

TurboGrid saves the file paths of the last several session files opened. To re-open a recently used session file, select **File > Recent Session Files** from the main menu and then select the file from the **Recent Session Files** submenu.

## 2.19. Quit Command

---

To exit from TurboGrid select **Quit** from the file menu. Objects created during the TurboGrid session are not automatically saved. If you want to save the objects before quitting, create a state file. For details, see [Save State Command \(Export State Command\)](#) (p. 35).

---

# Chapter 3: Edit Menu

---

The following sections describe the commands available in the **Edit** menu:

## 3.1. Undo and Redo Commands

### 3.2. Options Command


---

## 3.1. Undo and Redo Commands


Ansys TurboGrid includes an infinite Undo feature, limited only by the available memory on the machine and a few other restrictions described below. Select **Edit > Undo** from the main menu or click *Undo*




on the toolbar to return to the state immediately prior to when the last **Apply** action was executed.

Select **Edit > Redo** from the main menu or click *Redo*  on the toolbar to reapply changes that were undone. A **Redo** command must follow an undo or Redo command.

The undo function has a few limitations:

- When a mesh is created, the undo stack is cleared, meaning that the mesh creation process itself, and all commands before it, cannot be undone using the **Undo** command.
- While creating a session file, the undo/redo commands are not available.
- Any action that does not affect the state cannot be undone. For example, creating a mesh cannot be undone, nor can saving a topology file because neither of these actions changes the state.
- The undo function cannot return to the initial state of a default object. For example, after defining the `Hub` object for the first time, you cannot click *Undo* to return to the undefined state of the `Hub` object.
- Undo also reverses geometry manipulation when the named view icons located at the top of the Viewer window have been used. Rotation, zoom and translation actions performed using the mouse are not affected by selecting **Edit > Undo** from the main menu or by clicking on the  icon on the toolbar. You can, however, undo some viewer manipulations. For details, see [Viewer Hotkeys \(p. 83\)](#).

The redo feature is used to reverse an undo action. Selecting **Edit > Redo** from the main menu or clicking on the  icon on the toolbar can be done repeatedly to reverse as many undo actions as have been applied.

---

## 3.2. Options Command

The **Options** dialog box enables you to set various general preferences. Settings are retained per user.

1. Select **Edit > Options**.

The **Options** dialog box appears.

2. Set options as required. For descriptions of the available options, see:

- [TurboGrid Options \(p. 48\)](#)
- [Common Options \(p. 50\)](#)

If desired, you can use the **CFX Defaults** or the **Workbench Defaults** buttons at the bottom of the dialog box to quickly set TurboGrid to have the standard appearance and operation of CFX or Workbench respectively. The TurboGrid settings that are affected by these buttons are:

- **TurboGrid > Viewer > Background**
- **TurboGrid > Viewer > ANSYS Logo/Text Color/Edge Color**
- **Common > Viewer Setup > Mouse Mapping**

3. Click **OK**.

### 3.2.1. TurboGrid Options

The TurboGrid options are:

- **Enable Advanced Features**

Some advanced features are hidden in the user interface. You can select this option to unhide those advanced features.

- **Enable Beta Features**

Some Beta features are hidden in the user interface. You can select this option to "unhide" those features. When selected, such features are identified by "(Beta)" in the user interface. Note that Beta features are unofficial and not well tested.

- **Enable regions based on high and low blade geometry**

This preference is on by default. Turning off this preference prevents writing `GEO HIGH` and `GEO LOW` versions of the `HIGHBLADE` and `LOWBLADE` surface groups, enabling Fluent in Workbench to read meshes from TurboGrid via CGNS files; Fluent does not currently support reading the `GEO HIGH`- and `GEO LOW`-based surface groups. For details on surface groups, see [Surface Groups in the TurboGrid User's Guide \(p. 70\)](#).

- **Use ATM3D Meshing By Default**

Turning on this preference causes new cases to use the ATM3D meshing approach by default. This preference takes effect the next time you start TurboGrid.

You can override the default setting for the current case by setting **Mesh Generation Mode** to `ATM3D` in the `Topology Set` object. For details, see [Mesh Generation Mode \(p. 157\)](#).

- **Default CAD Generation Tool**

Specifies the default CAD Generation tool to use for importing NDF files (see [Import NDF File Command](#) (p. 32)) and for generating CAD from profile points (see [Geometry Browser Settings \(CAD From Profile Points Mode\)](#) (p. 114)).

- **Opening Region Tolerance**

This value is multiplied by the hub curve length (in a given case) in order to generate the default value for the **Hub Region Proximity Tolerance** setting. Similarly, this value is multiplied by the shroud curve length in order to generate the default value for the **Shroud Region Proximity Tolerance** setting. For details, see [Hub Regions and Shroud Regions Tab](#) (p. 122).

- **ATM Five Star Vertex Mesh Size Factor**

This value affects mesh smoothness. A value anywhere from 0.7 to 0.8 is recommended. A value much higher than 1.0 is not recommended.

### 3.2.1.1. Viewer

Highlight Type controls how an object is highlighted in the viewer window while in picking mode when highlighting is on. For details, see [Viewer Toolbar](#) (p. 81).

- If `Bounding Box` is selected, the object is always highlighted with a red box surrounding the Object.
- If `Wireframe` is selected, the object is traced with a red line if the object contains surfaces.

#### 3.2.1.1.1. Background

The following background options are available:

- `Color`: A constant color can be chosen.
- `Image`: One of a list of `Predefined` images or a `Custom` image can be selected. When setting a custom image, you must choose an image file and a type of mapping. The image types that are supported are: `jpg`, `png`, `ppm`. Mapping options are `Flat` and `Spherical`.

#### 3.2.1.1.2. Text Color and Edge Color

In the stand-alone version, set text color and edge color as appropriate.

#### 3.2.1.1.3. Axis Visibility

If the **Axis Visibility** check box is selected, the axis appears in the lower left corner of the viewer window. The axis is useful for reference when the geometry is rotated. The axis labels change when the viewer coordinates are transformed.

#### 3.2.1.1.4. Ruler Visibility

If the **Ruler Visibility** check box is selected, a ruler appears in the viewer to show the length scale.

### 3.2.1.1.5. Stereo

See [Stereo Viewer](#) (p. 86).

### 3.2.1.2. Mesh Quality Measures

The `Mesh Quality Measures` branch contains settings for the mesh analysis variables.

For each mesh analysis variable:

- There is a check box of the same name that controls:
  - Whether the variable is calculated during mesh generation and reported in the **Mesh Statistics** dialog box, and
  - Whether the variable appears in the **Mesh Limits** object.
- The settings below the check box serve as defaults for the **Mesh Limits** object.

For details, see:

- [Mesh Statistics](#) (p. 190)
- [Mesh Limits](#) (p. 191)

## 3.2.2. Common Options

Ansys TurboGrid includes an auto-save function that backs up work at set time intervals by saving a state file. Select the frequency of the auto save by picking a value from the drop-down list. **Auto Save** can be disabled by selecting `Never` from the list.

To change the directory in which auto-saved state files are saved, type a filename and path into the **Temporary Dir** box, or click the browse icon beside the **Temporary Dir** box to open the **Temporary Directory** dialog box.

### 3.2.2.1. Appearance

The appearance of the user interface can be controlled from the **Appearance** options. The default user interface style will be set to that of your machine. For example, on Windows, the user interface has a Windows look to it. If, for example, a Motif appearance to the user interface is preferred, select to use this instead of the Windows style.

1. Under **GUI Style**, select the user interface style to use.
2. For **Font** and **Formatted Font**, specify the fonts to use in the application.

---

**Note:**

It is important not to set the font size too high (over 24 pt. is not recommended) or the dialog boxes might become difficult to read. Setting the font size too small might cause some portions of the text to not be visible on monitors set at low res-

---

olutions. It is also important not to set the font to a family such as Webdings, Wingdings, Symbols, or similar type faces, or the dialog boxes become illegible.

---

### 3.2.2.2. Threading

Select **Enable Threading** and set **Hardware Capacity Fraction** to a value greater than 0 to make use of multiple CPU cores to improve the general performance of applications that share this common setting, at the possible expense of CPU availability for other applications. The maximum allowed value of 1 (default) uses all available CPU cores.

### 3.2.2.3. Units

Click `Units` in the options box to control the units that are presented in the user interface for each Quantity Type.

Select a pre-defined Units System such as `SI`, `English Engineering` or `British Technical`. The predefined Units for any quantity types in these units systems cannot be changed. Select the **Custom** units system to specify any valid units for each quantity type. For example, you may want to display **Length** in `mm` (an SI unit), but **Angle** in `radians` (not an SI unit). Click **More Quantity Types** to display the **Custom Quantity Types** form and set custom units for more quantity types.

The units set on this panel define the units used in the user interface for all quantity types. The units you select for a quantity type appear wherever that quantity type is used. For example, if you choose `mm` as the unit of **Length** and create a plot in Ansys TurboGrid colored by **Length**, you must specify a user specified length range in units of `mm`. If you created a Legend for the plot, the values on the Legend would be in units of `mm`.

If the **Always convert units to Preferred Units** check box is selected, all quantities entered in the user interface are converted into the preferred units.

---

**Note:**

Setting units in the variable editor will override the actual setting for that quantity type.

---

### 3.2.2.4. Viewer Setup: General

Viewer options under the `Common` branch are `Double Buffering` and `Unlimited Zoom` toggles.

---

**Note:**

Using `Unlimited Zoom` will allow the exceeding of the depth-buffer accuracy, resulting in rendering artifacts.

---

### 3.2.2.4.1. Double Buffering

`Double Buffering` is a feature supported by most OpenGL implementations. It provides two complete color buffers that swap between each other to animate graphics smoothly. If your implementation of OpenGL does not support double buffering, you can clear this check box.

### 3.2.2.4.2. Unlimited Zoom

By default, zoom is restricted to prevent graphics problems related to depth sorting. Selecting `Unlimited Zoom` allows an unrestricted zoom.

### 3.2.2.4.3. Use GPU Rendering for Printing

If your graphics hardware is compatible with GPU rendering, selecting this preference is strongly recommended.

When selected, this preference causes saved pictures in CFD-Post to use GPU rendering and saved pictures in CFX-Pre and TurboGrid to (by controlling the default value of a setting in the **Save Picture** dialog box) use GPU rendering by default.

When you save a picture with GPU rendering, your graphics hardware renders an image that closely matches what is shown in the viewer.

If GPU rendering is not set or cannot be used (for example, if compatible GPU hardware is not found), software rendering is used instead. Software rendering is relatively slow and does not always render as shown in the viewer. One benefit of software rendering is that it has no graphics hardware requirements.

---

**Note:**

Batch mode printing in CFX-Pre and TurboGrid does not support GPU rendering.

---

### 3.2.2.5. Viewer Setup: Mouse Mapping

The mouse mapping options enable you to assign viewer actions to mouse actions and keyboard/mouse combined actions.

A description of each action follows.

- Rotate: rotate the view about the screen X and Y axes.
- Object Zoom: drag the mouse up to zoom out and down to zoom in.
- Camera Zoom: drag the mouse up to zoom in and down to zoom out.
- Translate: drag the mouse to translate the view in the plane of the screen.
- Zoom Box: drag a rectangle around the area of interest. The selected area will fill the viewer when the mouse button is released.
- Zoom In: click the mouse button to zoom in step-by-step centered on the location of the mouse pointer.

- Zoom Out: click the mouse button to zoom out step-by-step centered on the location of the mouse pointer.
- Rotate Z: drag the mouse up to rotate the view clockwise about the screen Z axis, and down to rotate the view counterclockwise.
- Set Pivot Point: click an object to set the point about which the Rotate and Rotate Z actions pivot.
- Move Light: drag to move the angle of the virtual light source in the viewer. Drag the mouse left or right to move the horizontal lighting source and up or down to move the vertical lighting source. The lighting angle holds two angular values, each between 0° and 180°.



---

# Chapter 4: Session Menu

---

The following sections describe the commands available in the **Session** menu:

- 4.1. Introduction
- 4.2. Play Session Command
- 4.3. New Session Command
- 4.4. Start Recording Command
- 4.5. Stop Recording Command

## 4.1. Introduction

---

Session files contain a record of the commands issued during a TurboGrid session. Actions that cause commands to be written to a session file include:

- Creation of new objects and changes to existing objects committed by clicking **Apply** in the object editor.
- Creation of a mesh.
- Commands issued in the **Command Editor** dialog box.
- Calculations performed in the built-in calculator.
- Viewer manipulation performed using the icons located at the top of the viewer window. (Viewer manipulation performed using the mouse and keyboard are not recorded to a session file.)
- Creation of new cameras and selecting a camera view.
- All actions available from the **File** menu.
- Creation of expressions and user variables.
- Creation of chart lines and viewing charts.

Session files can be used to run TurboGrid in batch mode. For details, see [Batch Mode in the TurboGrid Reference Guide](#).

---

**Note:**

Since the session file is a text file of CCL commands, you can write your own session files using a text editor.

---

**Note:**


Any session file played from the **Session** menu should not contain the `>quit` command.

---

## 4.2. Play Session Command

---

A previously recorded session file can be played in TurboGrid. To play a session file, select **Session** >

**Play Session** from the main menu or click *Play Session*  on the toolbar. The **Play Session File** dialog box is displayed.

The commands listed in the selected session file are run. Existing objects with the same name as objects defined in the session file are replaced by those in the session file.

For any file specified in a state or session file, if the file cannot be found TurboGrid will automatically search the state or session file directory for a file of the same name. If this procedure fails, the current working directory will be searched. As a result, state and session files will not have to be edited to change the path when state, session and curve files are moved from one directory to another.

---

**Note:**

While recording a session file, any preference changes made using the **Options** dialog box (**Edit** > **Options**) are recorded. However, after playing back the session file, the preference changes generally affect the current session only and are not reflected in the **Options** dialog box.

---

**Note:**

A session file cannot be played if it contains the `Undo` command. To run a session file that contains the `Undo` command, edit the session file first to remove the command.

---

**Note:**

Because the `Topology Set` object is now suspended by default, session and state files from version 11 or earlier might not play or load correctly. To support older session and state files, the **Play Session File** and **Load State File** dialog boxes have an option named **Unsuspend TOPOLOGY SET before loading**. Selecting this option causes the `Topology Set` object to be unsuspended before playing/loading a session/state file. When starting TurboGrid

from the command line, adding the command line parameter `-u` causes the `Topology Set` object to be initially unsuspended.

---



---

### Note:

A session file from a recent release of TurboGrid is generally playable in the current release. However, the results might differ from those of the older release that recorded the session file.

---



---

### Note:

Workbench journal files for TurboGrid and TurboGrid session files from 12.0/12.1 may not work with 14.0. If the recorded file does not explicitly set the topology type, the following CCL block must be added:

```
TOPOLOGY SET:
ATM Topology Optimizer = off
END
```

For Workbench journal files, this CCL block should be added immediately before the following line occurs:

```
> um mode=normal, object=/TOPOLOGY SET
```

The line above appears only in journal files that were recorded while the `Topology Set` object was processed (unsuspended).

For session files, this CCL block should be added to the beginning of the session file.

---

For information about using Workbench journaling and scripting with TurboGrid, see [Using Ansys Workbench Journaling and Scripting with TurboSystem in the TurboSystem User's Guide](#).

## 4.3. New Session Command

---

To create a new Session file, select **Session > New Session** from the main menu or click *New Session*

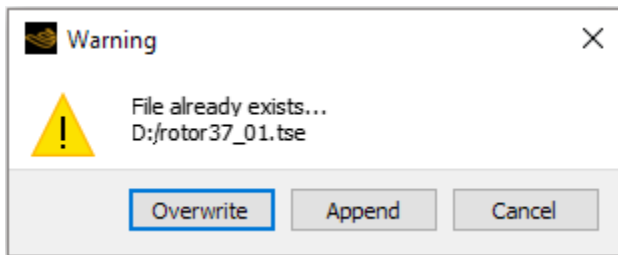


on the toolbar. The **Set Session File** dialog box is displayed.

Upon clicking **Save** in the dialog window, that file becomes the current session file. Commands are not written to the file until recording begins. For details, see [Start Recording Command \(p. 58\)](#).

Session files should be saved with a `.tse` file extension. The extension is added to a filename if `TG Session Files (*.tse)` is selected as the file type.


If you create more than one session file during a TurboGrid session, the most recently created file is by default the current session file. To set a different file to be the current session file, select an existing file from the **Set Session File** dialog box and then click **Save**. The following message then appears:



Click **Overwrite** to delete the existing session file and create a new file in its place. Click **Append** to add selected commands to the end of the existing session file when recording begins.

## 4.4. Start Recording Command


---

To start recording a session file, select **Session > Start Recording** from the main menu or click *Start Recording*  on the toolbar. This activates recording of CCL commands issued to the current session file. A session file must be set before recording can begin. For details, see [New Session Command \(p. 57\)](#).

While recording a session file, the *Undo*  and *Redo*  icons are disabled.

## 4.5. Stop Recording Command

---

To stop recording a session file, select **Session > Stop Recording** from the main menu or click *Stop Recording*  on the toolbar. This terminates recording of CCL commands issued to the current session file. Start and stop recording to a session file as many times as necessary.

---

# Chapter 5: Insert Menu

---

The following sections describe the commands available in the **Insert** menu.

---

## Note:

Some of the commands in the **Insert** menu are not available until you create a mesh.

---

[5.1. Introduction](#)

[5.2. Mesh Command](#)

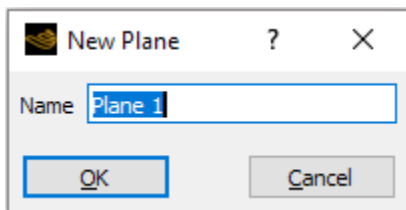
[5.3. User Defined Submenu](#)

---

## 5.1. Introduction

The **Insert** menu contains a list of objects that can be used as tools to analyze the mesh and geometry (although they are not necessary for mesh generation).

When you select any of the objects from the **Insert** menu, a dialog box appears in which you can either accept the default name or type a new one for the object (in this case a plane). The name should be different from any current object of the same type to avoid overwriting the existing object. Ansys TurboGrid does not enable you to create different types of objects with the same name.




Click **OK** or press **Enter** to open the relevant object editor. The object does not exist in the database until you click **Apply** in the object editor. For information on how to create objects from the command line see [Object Creation and Deletion in the TurboGrid Reference Guide](#)

---

## 5.2. Mesh Command

The flow path through the rotating machine is divided into small but discrete volumes. These volumes are primarily hexahedral elements, which have six sides and eight corners (wedge elements may be included in the blade tip region). There are nodes placed on each corner and the assembly of these nodes forms the mesh. The mesh fills the entire flow path and is used by Ansys CFX solvers. The quality of the solution depends partly on the quality of the Mesh. To create a mesh, select **Insert > Mesh** from

the main menu, click *Insert Mesh*  on the toolbar, or right-click anywhere in the viewer or object selector and select **Insert Mesh** from the shortcut menu.

Ansys TurboGrid creates the mesh using the current state of the `Topology Set` and `Mesh Data` objects. The time it takes to create the mesh depends on its size and the number of smoothing iterations chosen. Displayed in the status bar at the bottom left corner of the application window is an estimate of the total number of nodes and the total number of elements in the mesh. After the mesh is created, the color and rendering settings in the viewer window can be controlled for each individual mesh surface.

If changes are made to any of the `Geometry`, `Topology Set`, `Mesh Data`, or `Layers` objects, the mesh must be recreated in order for these changes to be included in the mesh.

## 5.3. User Defined Submenu

---

The **User Defined** submenu lists commands that create new objects.

Each command invokes a dialog box to prompt you for the name of the object to create. The name of a new object must be different from any existing object. When you click **OK** or press **Enter** to accept the name of the object, the relevant object editor opens, but the object is not created until you click **Apply** on the object editor.

### 5.3.1. Point Command

A point can exist anywhere within or outside the domain. To create a new point, select **Insert > User Defined > Point** from the main menu.

#### 5.3.1.1. Point: Geometry Tab

##### 5.3.1.1.1. Point Definition

The available options for defining a point are described next:

- `XYZ`

Set the point coordinates. To set the coordinates, first click one of the point coordinate boxes. The point coordinates will be displayed with a yellow background, and the viewer will switch to picking mode. You can then pick a point directly by clicking a visible object in the viewer. The point can lie outside of the domain.

You can alternatively set the coordinates one at a time by typing in the coordinates and/or using the embedded slider that appears beneath the coordinate boxes.

- `Node Number`

Create a point at a nodal location. Use the **Domains** drop-down list to select the domain(s) in which the locator exists. After choosing the domain(s) in which to select the node, enter the node number. Type in the node number or set it using the embedded sliders. When more than one domain is selected, a point is created for the specified node number in each domain (if the node number exists). If the node number does not exist in one domain but exists in another, select only the domain in which the node exists or an error message is displayed.

- `Variable Minimum and Variable Maximum`

Create the point where a variable is at its maximum or minimum value on any named locator. The **Domains** drop-down list is used to select the domain(s) in which the locator exists. After choosing the domain(s), select the locator name and the variable of interest. When more than one domain is selected, a point is created for the maximum/minimum value of the variable within each domain.

#### 5.3.1.1.2. Symbol Definition

The **Symbol Size** must be between 0 and 10 (10 being a similar scale to the geometry). Type in the value or set it using the embedded slider.

Set **Symbol** to one of the available symbols.

Picking Mode can be used to select and/or translate points in the viewer. For details, see [Viewer Toolbar \(p. 81\)](#).

---

**Note:**

You cannot move points that have been defined using `Node Number, Variable Minimum, or Variable Maximum`.

---

#### 5.3.1.2. Point: Color Tab

See [Color Tab \(p. 23\)](#) for details.

#### 5.3.1.3. Point: Render Tab

See [Render Tab \(p. 25\)](#) for details.

### 5.3.2. Line Command

A line locator can exist between two points anywhere within or outside the domain. To create a new line, select **Insert > User Defined > Line** from the main menu.

#### 5.3.2.1. Line: Geometry Tab

##### 5.3.2.1.1. Domains

Select the domain(s) in which the line will exist.

##### 5.3.2.1.2. Line Definition

Lines are created by defining two points. To set the coordinates, first click one of the point coordinate boxes. The point coordinates will be displayed with a yellow background, and the viewer will switch to picking mode. You can then pick a point directly by clicking a visible object in the viewer. The point can lie outside of the domain.

You can alternatively set the coordinates one at a time by typing in the coordinates and/or using the embedded slider that appears beneath the coordinate boxes.

### 5.3.2.1.3. Line Type

Set the **Line Type** to either **Cut** or **Sample**.

- **Cut** extends the line in both directions until it reaches the edge of the domain. Points on the line correspond to points where the line intersects a mesh element face. As a result, the number of points on the line is indirectly proportional to the mesh spacing.
- **Sample** creates the line between the two specified points. The sample line is a set of evenly-spaced sampling points that are independent of the mesh spacing. The number of points along the line corresponds to the value in the **Samples** box.

Picking Mode can be used to select and/or translate lines in the viewer. For details, see [Viewer Toolbar \(p. 81\)](#).

### 5.3.2.2. Line: Color Tab

See [Color Tab \(p. 23\)](#) for details.

### 5.3.2.3. Line: Render Tab

See [Render Tab \(p. 25\)](#) for details.

## 5.3.3. Plane Command

A plane locator is a two-dimensional area that exists only within the boundaries of the geometry. To create a new plane, select **Insert > User Defined > Plane** from the main menu.

### 5.3.3.1. Plane: Geometry Tab


#### 5.3.3.1.1. Domains

Select the domain(s) in which the plane will exist.


#### 5.3.3.1.2. Plane Definition

The available methods for defining a plane are described next:

- YZ Plane


Create a plane normal to the X axis and at a specific X value. Type in a value for **X**, set it using the embedded slider, or click in the box beside **X**, then click *Enter Expression*  to the right of the **X** setting and enter its value as an expression. For details, see [Expressions Command \(p. 92\)](#).

- ZX Plane

Create a plane normal to the Y axis and at a specific Y value. Type in a value for **Y**, set it using the embedded slider, or click in the box beside **Y**, then click *Enter Expression*  to the right

of the **Y** setting and enter its value as an expression. For details, see [Expressions Command \(p. 92\)](#).

- XY Plane

Create a plane normal to the Z axis and at a specific Z value. Type in a value for **Z**, set it using the embedded slider, or click in the box beside **Z**, then click *Enter Expression*  to the right of the **Z** setting and enter its value as an expression. For details, see [Expressions Command \(p. 92\)](#).

- Point and Normal

Create a plane using a single point on the plane and a vector normal to the plane.

To set the point coordinates, first click one of the point coordinate boxes. The point coordinates will be displayed with a yellow background and the viewer will switch to picking mode. You can then pick a point directly by clicking a visible object in the viewer. The point can lie outside of the domain.

You can alternatively set the coordinates one at a time by typing in the coordinates and/or using the embedded slider that appears beneath the coordinate boxes.

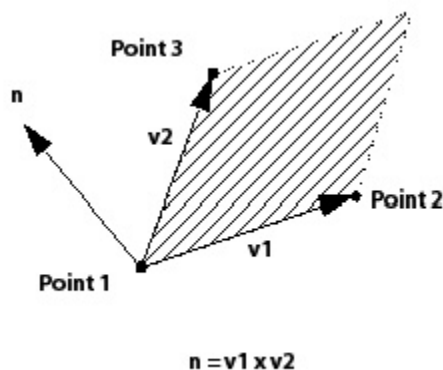
When picking a point from the viewer for the normal vector, the vector is taken from the origin to the point picked.

- Three Points

Create a plane using three points. To set the point coordinates, first click one of the point coordinate boxes. The point coordinates will be displayed with a yellow background, and the viewer will switch to picking mode. You can then pick a point directly by clicking a visible object in the viewer. The point can lie outside of the domain.


You can alternatively set the coordinates one at a time by typing in the coordinates and/or using the embedded slider that appears beneath the coordinate boxes.

The normal vector to the plane is calculated using the right-hand rule. The first vector is from Point 1 to Point 2 and the second is from Point 1 to Point 3 as shown in the diagram below.



### 5.3.3.1.3. Plane Bounds

The available types of bounds are described next:

- When **None** is selected the plane cuts through a complete cross-section of each domain specified in the **Domains** list. The plane is only bounded by the limits of the domains.
- The **Circular** option defines the bounds of a plane as a circle centered at the point used in the **Plane Definition**. Enter the value of the radius of the circle or click in the box beside **Radius**, then click *Enter Expression*  to the right of the **Radius** setting to enter an expression for the radius of the circle. The plane is undefined in areas where the circle extends outside of the domains specified in the **Domains** list.
- For the **Rectangular** option, the plane bounds are defined by a rectangle centered about the point selected in the **Plane Definition** with lengths in the x and y-directions of **X Size** and **Y Size**, respectively. You may have to specify lengths in other directions, depending on the **Plane Definition**. The size is determined with reference to the plane center (that is, the plane is resized around its center). The **X Angle** value rotates the plane counter-clockwise about its normal by the specified number of degrees. The plane is undefined in areas where the rectangle extends outside of the domains specified in the **Domains** list.

Both the circular and rectangular options have an **Invert Plane Bound** check box. If this check box is selected, the area defined by the rectangle or circle is used as a cut-out area from a slice plane that is bounded only by the domain(s). The area inside the bounds of the rectangle or circle does not form part of the plane, but everything on the slice plane outside of these bounds is included.

### 5.3.3.1.4. Plane Type

Set **Plane Type** to either **Slice** or **Sample**.

**Slice** extends the plane in all directions until it reaches the edge of the domain. Points on the plane correspond to points where the plane intersects an edge of the mesh. As a result, the number of points in a slice plane is indirectly proportional to the mesh spacing.

**Sample** creates the plane with either circular or rectangular bounds, depending on the plane bounds selected. For the **Circular** option, the density of points on the plane corresponds to the radius of the plane specified in the **Plane Bounds** frame, and the values for the **Radial** and **Circumferential** settings in the **Plane Type** frame. For rectangular bounds, the density of points on the plane corresponds to the size of the bounds for the plane in each of the plane directions, and the sample values provided in the **Plane Type** frame. A sample plane is a set of evenly-spaced points that are independent of the mesh spacing.

Picking Mode can be used to select and/or translate planes in the viewer. For details, see [Viewer Toolbar \(p. 81\)](#).

### 5.3.3.2. Plane: Color Tab

See [Color Tab \(p. 23\)](#) for details.

### 5.3.3.3. Plane: Render Tab

See [Render Tab \(p. 25\)](#) for details.

## 5.3.4. Turbo Surface Command


A *turbo surface* is a surface that exists only within the boundaries of the geometry and is defined using variables specific to rotating machinery. To create a new turbo surface, select **Insert > User Defined > Turbo Surface** from the main menu.

### 5.3.4.1. Turbo Surface: Geometry Tab

#### 5.3.4.1.1. Domains

Select the domain(s) in which the turbo surface will exist.

#### 5.3.4.1.2. Turbo Surface Definition

Ansys TurboGrid creates a turbo surface using a defined variable and its value. Set the domain to either `All Domains` or `DOMAIN: Passage`. Select a variable and set a value to define the location of the turbo surface. (The variable is "K" by default, which is the mesh plane index that varies from hub to shroud for the H-Grid and J-Grid topologies.) You may enter the value directly, or, for the "K" variable, use the up/down arrows, or, for any other variable, click in the **Value** box, then click *Enter Expression*  (to the right of the **Value** box) to enter an expression for the value, or use the slider.

### 5.3.4.2. Turbo Surface: Color Tab

See [Color Tab \(p. 23\)](#) for details.

### 5.3.4.3. Turbo Surface: Render Tab

See [Render Tab \(p. 25\)](#) for details.

## 5.3.5. Volume Command

A *volume* object is a collection of mesh elements; it can be created anywhere within the domain. To create a new volume, select **Insert > User Defined > Volume** from the main menu.

### 5.3.5.1. Volume: Geometry Tab

#### 5.3.5.1.1. Domains

Select the domain(s) in which the volume will exist.

#### 5.3.5.1.2. Volume Definition

The available methods for defining a volume are described next:

- Sphere

Create a volume in the shape of a sphere by defining the center point and radius for the sphere.

To set the coordinates of the center points, first click one of the point coordinate boxes. The point coordinates will be displayed with a yellow background, and the viewer will switch to picking mode. You can then pick a point directly by clicking a visible object in the viewer. The point can lie outside of the domain.



You can alternatively set the coordinates one at a time by typing in the coordinates and/or using the embedded slider that appears beneath the coordinate boxes.

Type in the radius value or set it using the embedded slider. Ensure that the units are set correctly. When **Mode** is set to *Intersection*, the volume consists of all elements that intersect the sphere surface. When **Mode** is set to *Below Intersection* or *Above Intersection*, the volume consists of all elements that are "below" or "above" the sphere surface, respectively. If the **Inclusive** check box is selected, all elements that intersect the sphere surface are added to the volume.

- From Surface

Create a volume using an existing surface. Set **Location** to one or more of the available surfaces for defining the volume (use **Ctrl** to multi-select). When **Mode** is set to *Intersection*, the volume consists of all elements that intersect the sphere surface. When **Mode** is set to *Below Intersection* or *Above Intersection*, the volume consists of all elements that are "below" or "above" the sphere surface, respectively. If the **Inclusive** check box is selected, all elements that intersect the sphere surface are added to the volume.

- Isovolume

Create a volume based on the value(s) of a variable. Click  next to the variable box to see the available variables for defining the volume. Type in the value(s), set the value(s) using the embedded slider, or click in the box, click *Enter Expression*  to the right of the box, then enter the value(s) as an expression. Set the units for the variable(s) appropriately. When **Mode** is set to *At Value*, the volume consists of all elements having the defined variable value. When **Mode** is set to *Below Value* or *Above Value*, the volume consists of all elements that are "below" or "above" the defined variable value, respectively. When **Mode** is set to *Between Values*, the volume consists of all elements that are "between" the defined variable values. If the **Inclusive** check box is selected, the elements having the defined variable value are added to the volume.

### 5.3.5.1.3. Hybrid/Conservative

For volumes made with **Method** set to *Isovolume*, the option of using hybrid or conservative values is available. Unless you are postprocessing CFD results using Ansys TurboGrid, this option can be ignored. For details, see [Hybrid and Conservative Variable Values \(p. 96\)](#).

---

**Note:**

Volumes are not displayed as perfect shapes (for example, a perfect sphere) because mesh elements are either included or excluded from the volume.

---

### 5.3.5.2. Volume: Color Tab

See [Color Tab \(p. 23\)](#) for details.

### 5.3.5.3. Volume: Render Tab

See [Render Tab \(p. 25\)](#) for details.

## 5.3.6. Isosurface Command


An *isosurface* object is a surface upon which a particular variable has a constant value, called the "level". To create a new Isosurface, select **Insert > User Defined > Isosurface** from the main menu.

### 5.3.6.1. Isosurface: Geometry Tab

#### 5.3.6.1.1. Domains

Select the domain(s) in which the isosurface will exist.

#### 5.3.6.1.2. Isosurface Definition

Set **Variable** to one of the available variables for defining the isosurface. Type in the variable value, set it using the embedded slider, or click in the box, click *Enter Expression*  (to the right of the box), and enter an expression. Set appropriate units for the variable. The isosurface connects all locations with the specified variable value.

#### 5.3.6.1.3. Hybrid/Conservative

The option of using hybrid or conservative values is available. Unless you are postprocessing CFD results using Ansys TurboGrid, this option can be ignored. For details, see [Hybrid and Conservative Variable Values \(p. 96\)](#).

### 5.3.6.2. Isosurface: Color Tab

See [Color Tab \(p. 23\)](#) for details.

---

#### Note:

You may color the isosurface using any variable or choose a constant color. You should not select the `Local` range option when coloring an isosurface with the variable used to define it. In this case the local range is zero by definition, so a plot would be colored according to the effects of round-off error.

---

### 5.3.6.3. Isosurface: Render Tab

See [Render Tab \(p. 25\)](#) for details.

## 5.3.7. Polyline Command

### 5.3.7.1. Polyline: Geometry Tab

A *polyline* object is a set of connected line segments that connect a series of points. To create a polyline, select **Insert > User Defined > Polyline** from the main menu.

Available methods of creating a polyline are:

- From File
- Boundary Intersection

The file format for a polyline is shown below:

```
[Name]
Polyline 1
[Data]
X [ m ], Y [ m ], Z [ m ], Area [ m^2 ], Density [ kg m^-3 ]
-1.04539007e-01, 1.68649014e-02, 5.99999987e-02, 0.00000000e+00, ...
-9.89871025e-02, 3.27597000e-02, 5.99999987e-02, 0.00000000e+00, ...
.
.
.
[Lines]
0, 1
1, 2
.
.
.
[Name]
Polyline 2
.
.
.
```

In this example, the two lines containing data are shown word-wrapped onto the next line. In the actual file, all data for a given point must be on a single line.

The name of each locator is listed under the **Name** section. Point coordinates and the corresponding variable values are stored in the **Data** section. Line connectivity data is listed in the **Lines** section; this data makes reference to the points that are defined in the **Data** section. For this purpose, point numbering in the **Data** section is consecutive, starting at zero.

Comments in the file are preceded by # (or ## for the CFX-5.6 polyline format) and can appear anywhere in the file.

Blank lines are ignored and can appear anywhere in the file (except between the line "[Data]" and first line of data following it).

### 5.3.7.2. Polyline: Color Tab

See [Color Tab \(p. 23\)](#) for details.

### 5.3.7.3. Polyline: Render Tab

See [Render Tab \(p. 25\)](#) for details.

## 5.3.8. Surface Command

To create a user surface, select **Insert** > **User Defined** > **Surface** from the main menu.

### 5.3.8.1. User Surface: Geometry Tab

#### 5.3.8.1.1. Method

Choose one of the available methods for defining a user surface:

- From File

Select the `From File` alternative when you are not able to create the required surface using the `Boundary Intersection` method. This more versatile option reads data describing the surface from a file. The points may have *path variables* (variables that are only defined on the surface) associated with them. For details on the required format for a surface data file, see [Surface Data Format \(p. 69\)](#). Click the browse icon to open an **Import** dialog box and browse to the surface data file. Alternatively, you can type the path and filename into the **Input File** box.

- Boundary Intersection

`Boundary Intersection` forms a user surface where a given locator intersects a specified list of boundaries. The **Domains** setting specifies the domains in which this user surface exists.

To select multiple boundaries for the **Boundary List**, click  and hold **Ctrl** as you select each boundary. Set **Location** to one of the existing graphic objects.

The user surface consists of mesh element faces on the specified boundaries, for the mesh elements that are intersected by the locator. The user surface is usually narrow with a varying width. The fluctuation in width is more noticeable in a coarser mesh.

#### 5.3.8.1.2. Surface Data Format

When the `From File` method is selected, an external file must exist that defines the surface. A set of surfaces can be defined in a simple text file with the following format.

```
## Comment line - optional.
## List of path variables
# <varName1>
# <varName2>
# ...
## List of point locations with path variable values
## Each line in the following list is numbered 0,1,2...
<X> <Y> <Z> <Var1Value> <Var2Value> ...
<X> <Y> <Z> <Var1Value> <Var2Value> ...
...
## Next line is a keyword that starts the definition of faces
# Faces
## List of 3 - 6 point numbers to define faces.
<Point0> <Point1> <Point2>
<Point1> <Point2> <Point3> <Point4> <Point5> <Point6>
...
```

Comments in the file are preceded by ## and can appear anywhere in the file. A single # does *not* indicate a comment; words appearing after a single # are keywords such as `Faces`.

The start of the file should begin with a list of path variables (up to 256 characters, spaces allowed). These are variables that are only defined on the user surface. Ensure that the names of these variables do not conflict with the names of existing variables. There is no need to define any path variables (if you just want to define the location of a user surface), in which case the file begins with the point location values.

The point location list in `X Y Z` format follows the optional path variable list. You must also include a value for each path variable that you have defined at the start of the file (if any). Surfaces are defined by typing # `Faces` followed by lists of 3 (triangle) to 6 (hexagon) points to define each surface. Each surface is automatically closed by connecting the last point to the first point. The list of point locations are numbered 0,1,2....n-1 where n is the number of points in the list. When defining faces, use these numbers to reference the points in the point location list. The faces specification is *not* optional.

Blank lines are ignored and can appear anywhere in the file.

The following example defines one quadrilateral face with two path variables at each point of the face:

```
# Time
# MyVar
1 1 1 1.2 500
1 2 1 2.1 200
2 2 1 3.4 300
2 1 1 4.65 400
# Faces
0 1 2 3
```

### 5.3.8.2. User Surface: Color Tab

See [Color Tab \(p. 23\)](#) for details.

### 5.3.8.3. User Surface: Render Tab

See [Render Tab \(p. 25\)](#) for details.

### 5.3.8.4. Surface Groups

Surface groups are produced automatically when a mesh is generated. They are found in the 3D `Mesh` branch of the object selector. Each surface group shows a surface of the mesh. The available surface groups vary according to the number of blades, the types of leading and trailing edges, and a user preference. A representative list follows:

- HUB, SHROUD
- One of the following two sets of surface groups, depending on user preference **Enable regions based on high and low blade geometry**, which is found under **Edit > Options > TurboGrid**:
  - With the preference selected:

[blade name] LOWBLADE GEO LOW, [blade name] LOWBLADE GEO HIGH, [blade name] HIGHBLADE GEO LOW, [blade name] HIGHBLADE GEO HIGH

- With the preference not selected:

[blade name] HIGHBLADE, [blade name] LOWBLADE

If the blade set contains more than one blade, the blade name is included in each surface group name. LOWBLADE and HIGHBLADE refer to sides of the blade, as divided according to the mesh topology. For example, the superset of all LOWBLADE surface groups for a particular blade forms a surface that is mainly on the low-theta side of the blade. GEO LOW and GEO HIGH refer to sides of the blade, as divided by the leading and trailing edges, considering the centroid of each mesh element face on the blade surface. For example, the superset of all GEO LOW surface groups for a particular blade forms a surface, the face centroids of which are all on the low-theta side of the blade between the leading and trailing edges.

- [blade name] BLADE LE, [blade name] BLADE TE

If there is one blade in the blade set, then the surface groups BLADE TE and BLADE LE will be available, as applicable, after generating a mesh. These surface groups are applicable only for cut-off or square leading/trailing edges. If there are two or more blades in the blade set, the surface group names start with the blade name. For example, if there are two blades named Main and Splitter, and the trailing edge of Main is cut-off, and the leading and trailing edges of Splitter are cut-off, then surface groups Main BLADE TE, Splitter BLADE LE and Splitter BLADE TE will be available after generating a mesh.

- LOWPERIODIC, HIGHPERIODIC
- INLET, OUTLET

"LOW" refers to low Theta value and "HIGH" refers to high Theta value.

A visibility check box next to each surface group enables you to control which are displayed.

Besides the visibility, you may also change the color and render properties for each surface group.

#### 5.3.8.4.1. Surface Group: Definition Tab

You may view the **Domains** and **Locations** information on the **Definition** tab.

#### 5.3.8.4.2. Surface Group: Color Tab

See [Color Tab \(p. 23\)](#) for details.

#### 5.3.8.4.3. Surface Group: Render Tab

See [Render Tab \(p. 25\)](#) for details.

### 5.3.9. Contour Command


A *contour plot* is a series of lines linking points with equal values of a given variable. For example, contours of height exist on geographical maps and give an impression of gradient and land shape. To create a contour, select **Insert > User Defined > Contour** from the main menu.

## 5.3.9.1. Contour Plot: Definition Tab

### 5.3.9.1.1. Domains

Select the domain(s) in which the contour object will exist.

### 5.3.9.1.2. Locations


Select the locator(s) on which to plot the contours. To select multiple locators, click the  icon, hold down **Ctrl**, and select each locator.

### 5.3.9.1.3. Variable

Choose the plot variable for the contour plot.

### 5.3.9.1.4. Range

Set **Range** to one of the available methods for defining the range of the contour plot. This affects the variation of color used when plotting the contours in the viewer. The lowest values of a variable in the selected range are shown in blue in the viewer, the highest values are shown in red.

- **Global** uses the range of the variable over all domains (regardless of the domains selected on the **Geometry** tab) to determine the minimum and maximum values for the contours.
- **Local** uses the range of the variable over the selected locator(s) to determine the minimum and maximum values for the contours.
- When using **User Specified**, enter the minimum and maximum values for the contours. Type in the variable values, set them using the embedded slider or, by clicking *Enter Expression*  to the right of the **Units** box, enter them as an expression.
- Using **Value List**, a list separated by commas, specify the actual values at which contours should be plotted. For example, if plotting minimum face angle, try a value list of 5, 10, 15, 20, 25 degrees. It should be noted that entering a value list overrides the number specified in the **# of Contours** box (see below).

### 5.3.9.1.5. Hybrid/Conservative


The option of using hybrid or conservative values is available. Unless you are postprocessing CFD results using Ansys TurboGrid, this option can be ignored. For details, see [Hybrid and Conservative Variable Values](#) (p. 96).

### 5.3.9.1.6. Number of Contours

Set the **# of Contours** to appear in the plot. This is the number of bands plus one. This number is overridden if the **Range** setting is set to **Value List**.

### 5.3.9.2. Contour Plot: Labels Tab

The **Show Numbers** check box determines whether numbers corresponding to the number of contours are displayed on the plot. To view the values of the plotted variable at each contour, create a legend of the contour plot. See [Legend Command \(p. 74\)](#) for more details. To change the size of the text that appears on the contour plot, type a new value into the **Text Height** box or use the embedded slider (which has a maximum value of 1 and a minimum value of 0). The **Text Height** number is a fraction of the viewer height. **Text Font** controls which font is used for the numbers that appear in the contour plot. To change the text color, set **Color Mode** to *User*

*Specified* then choose a color by clicking in the **Text Color** box or by clicking the  icon, which invokes a color selector.

### 5.3.9.3. Contour Plot: Render Tab

See [Render Tab \(p. 25\)](#) for details.

---

#### Note:


When the **Draw Faces** check box is selected, the area between contour lines is shaded with a color that corresponds to a value midway between the upper and lower contour line value. For example, for a contour line at 1000 Pa and a contour line at 1200 Pa, the shaded area has a color that corresponds to 1100 Pa. If you have created a legend for the contour plot, the legend adopts flat shading between 2 contour levels. By referring to the legend, the variable values can quickly be associated with the shaded regions of the plot.

---






---

#### Note:

To view the contour lines as a single color, select the **Constant Coloring** check box. With this option selected, you can either leave the color set to default or you can set **Color Mode** to *User Specified* then choose a color by clicking in the **Text Color** box or by clicking the  icon, which invokes a color selector.

---

## 5.3.10. Instance Transform Command

Due to rotational symmetry, only one blade passage must be meshed, reducing computing cost and time. *Instance transforms* are used to replicate sections of the computational domain in the viewer for viewing purposes. For example, you may use a rotational transform to copy a blade passage to produce a plot of an entire rotor or stator, or a fraction thereof. However, for most purposes, it is sufficient to make use of the special toolbar icons dedicated to viewing 1, 2, or all instances (, , and  respectively). Instance transforms are more flexible than these special toolbar icons in that they can be applied to single objects rather than to only all (qualified) objects at the same time.

To create an instance transform, select **Insert > User Defined > Instance Transform** from the main menu.

### 5.3.10.1. Instance Transform: Definition Tab

---

**Note:**

In this release of Ansys TurboGrid, instancing is purely visual. This means that quantitative calculations can be carried out only for the original geometry.

---

#### 5.3.10.1.1. Number of Copies

The **# of Copies** is the number of times the domain is replicated in the viewer. Type in the value, increase or decrease the value by 1 by clicking ▲ or ▼ respectively.

To see the whole rotating machine, the number of copies must equal the number of blade sets, as defined in the `Machine Data` object.

#### 5.3.10.1.2. CCL Editing

If you edit an instance transform in the **Command Editor** dialog box, changes to settings, other than **Number of Copies**, will be lost the next time the **Apply** button is clicked in the instance transform editor. This happens because the instance transform editor overwrites all of the ccl parameters, except **# of Copies**, with the values stored in the default instance transform object.

### 5.3.11. Legend Command

A legend can be created for any object that plots a variable. The legend gives an approximate quantitative value to the colors representing the variable on a locator. To create a legend, select **Insert > User Defined > Legend** from the main menu. After a legend object is created, it is listed in the object selector under `User Defined`.

#### 5.3.11.1. Legend: Definition Tab

##### 5.3.11.1.1. Plot

Set **Plot** to the object for which to create a legend.

---

**Note:**

Any existing object can be selected, but if there is not a variable for which to create a legend, one is not created.

---

##### 5.3.11.1.2. Location

The exact position of the legend in the viewer can be controlled using the **Location** settings. Set **X Justification** to `Left`, `Center`, `Right`, or `None`. Similarly, set **Y Justification** to `Top`, `Center`, `Bottom`, or `None`. In both cases, if `None` is selected, type in the position value(s) (or use the embedded sliders, which have a maximum value of 1 and a minimum value of 0) in the appropriate **Position** box (the left box is for **X Justification** and the right box is for **Y Justification**). The position values represent a fraction of the viewer width from the left side for **X Position**, or a fraction of the viewer height from the bottom for **Y Position**. The position entered is the bottom-left corner of the legend.

## 5.3.11.2. Legend: Appearance Tab

### 5.3.11.2.1. Sizing Parameters

The size of the legend can be set as a fraction of the viewer window height. Increasing the value of the **Size** setting increases both the height and width of the legend. Type in the size value or use the embedded slider (which has a maximum value of 1 and a minimum value of 0).

The **Aspect** setting controls the width of the color range bar displayed in the legend. Type in the aspect value or use the embedded slider (which has a maximum value of 0.2 and a minimum value of 0).

### 5.3.11.2.2. Text Parameters

The **Precision** setting controls the number of digits after the decimal point, for numbers displayed in the legend. Set **Precision** to a format of either `Scientific` or `Fixed`.

The **Value Ticks** setting specifies the number of graduations (with labels) displayed in the legend. For example, for a scale ranging from 0 to 10, a setting of 3 ticks produces graduations at 0, 5 and 10. For a contour plot, each number assigned to a contour line is displayed on the legend, along with its associated variable value. The **Value Ticks** setting has no effect when the legend is for a contour plot.


## 5.3.12. Text Command

Text can be added to the viewer, for example for annotation or comments. To create text, select **Insert > User Defined > Text** from the main menu. After a text object is created, it is listed in the object selector under `User Defined`.

### 5.3.12.1. Text: Definition Tab

#### 5.3.12.1.1. Location

Set **Position Mode** to one of the available methods for defining the location of the text.


- Using **Two Coords**, position the text at a fixed location in the viewer. Set **X Justification** to `Left`, `Center`, `Right`, or `None`. Similarly, set **Y Justification** to `Top`, `Center`, `Bottom`, or `None`. In both cases, if `None` is selected, type in the position value(s) (or use the embedded sliders, which have a maximum value of 1 and a minimum value of 0) in the appropriate **Position** box (the left box is for **X Justification** and the right box is for **Y Justification**). The position values represent a fraction of the viewer width from the left side for **X Position**, or a fraction of the viewer height from the bottom for **Y Position**. The position entered is the bottom-left corner of the text.
- Using **Three Coords**, position the text using Cartesian coordinates attached to the geometry. The text moves with the geometry, but its orientation remains fixed. The X, Y, and Z coordinates are required to set the text location. Type in the position values or use the embedded sliders. Type in the rotation value, or set it using the embedded slider (which has a maximum value of 360 degrees and a minimum value of 0 degrees), or click in the **Rotation** box, click *Enter*  (to the right of the **Rotation** setting), and enter an expression. A rotation angle

of 0 degrees positions the text horizontally; a positive angle is measured counter-clockwise from that position.

## 5.3.12.2. Text: Appearance Tab

### 5.3.12.2.1. Text Properties

Type in the text height or use the embedded slider (which has a maximum value of 1 and a minimum value of 0).

To change the text color, change **Color Mode** to `User Specified` then choose a color by clicking in the **Color** box or by clicking the  icon, which invokes a color selector.

---

# Chapter 6: Display Menu

---

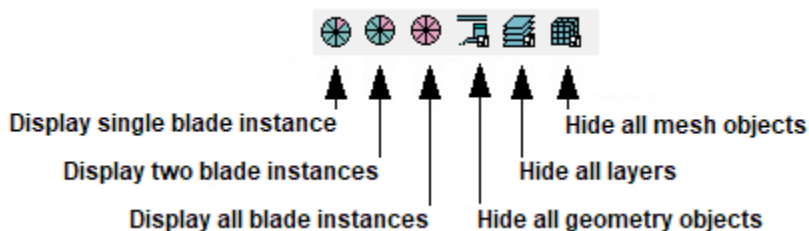
The following sections describe the commands available in the **Display** menu:

- 6.1. Introduction
- 6.2. Display One Instance Command
- 6.3. Display Two Instances Command
- 6.4. Display All Instances Command
- 6.5. Hide/Unhide Geometry Objects Commands
- 6.6. Hide/Unhide Layers Commands
- 6.7. Hide/Unhide Mesh Objects Commands
- 6.8. Use Mesh for Low/High-Fidelity Geometry Commands
- 6.9. Blade-to-Blade View Submenu

## 6.1. Introduction

---

The **Display Menu** options are also available from a toolbar located above the viewer.



## 6.2. Display One Instance Command

---

The default setting, **Display One Instance**, will show one passage of the geometry in the viewer. Because only one instance must be solved, displaying only one instance enables Ansys TurboGrid to work more quickly because less rendering is required.

For further information, see [Instance Transform Command](#) (p. 73).

## 6.3. Display Two Instances Command

---

When **Display Two Instances** is set, two passages of the geometry are shown in the viewer. This can give a better idea of the full machine without having Ansys TurboGrid slowed by rendering the full machine.

For further information, see [Instance Transform Command](#) (p. 73).

## 6.4. Display All Instances Command

---

When **Display All Instances** is set, the full machine is shown in the viewer. This setting shows the full geometry; the mesh can be seen on the entire geometry. It is not recommended to work with this setting because the constant rendering of the full machine will slow down processing speed.

For further information, see [Instance Transform Command](#) (p. 73).

## 6.5. Hide/Unhide Geometry Objects Commands

---

Select **Hide Geometry Objects** to turn off the visibility of all geometry objects (that is, hub, shroud, blade, and so on) and give a clear view of the remaining objects. Select **Unhide Geometry Objects** to do the opposite.

## 6.6. Hide/Unhide Layers Commands

---

Select **Hide Layers** to turn off the visibility of all of the layers. This is much more efficient than turning off the visibility for each individual layer in the object selector. Select **Unhide Layers** to do the opposite.

## 6.7. Hide/Unhide Mesh Objects Commands

---

Select **Hide Mesh Objects** to turn off the visibility of the 3D Mesh objects. This can be useful for viewing the geometry after a mesh has been created. Select **Unhide Mesh Objects** to do the opposite.

## 6.8. Use Mesh for Low/High-Fidelity Geometry Commands

---

Select **Use Mesh for Low-Fidelity Geometry** for generating a mesh based on the low-fidelity geometry.

Select **Use Mesh for High-Fidelity Geometry** for generating a mesh based on the high-fidelity geometry, for example a geometry with a complex blade end.

These commands affect any existing mesh, any mesh subsequently generated, and any mesh subsequently saved.

## 6.9. Blade-to-Blade View Submenu

---

In the blade-to-blade view, some parts of the mesh might appear to be distorted, wavy, or overlapping<sup>[1]</sup>. You might be able to reduce the amount of distortion by selecting an appropriate command in the **Blade-to-Blade View** submenu. Each command affects the portion of the geometry on which the transform is based. The optimal choice depends on the blade geometry.

The **Blade-to-Blade View** submenu commands are:

- **Use Default Transform**

---

[1] By contrast, the Cartesian view does not typically exhibit distortion, except at extremely high zoom levels.

The **Use Default Transform** command chooses the transform method automatically by effectively choosing either the **Use Full Transform** command or the **Use Passage Transform** command.

- **Use Full Transform**

The **Use Full Transform** command causes the blade-to-blade coordinates to be calculated using the complete hub and shroud curves.

- **Use Passage Transform**

The **Use Passage Transform** command causes the blade-to-blade coordinates to be calculated using the portion of the hub and shroud curves that fall within the passage mesh, truncated at the inlet and outlet (that is, excluding the portions of the hub and shroud curves that lie within the inlet and outlet domains).

- **Use Passage Excluding Tip Transform**

The **Use Passage Excluding Tip Transform** command is similar to the **Use Passage Transform** command, except that the tip regions are excluded in the spanwise direction. For example, if a blade has no hub tip and a profile-based shroud tip, the blade-to-blade coordinates are calculated using the portion of the hub curve that falls within the passage and the profile curve at the shroud tip. This transform may be the best choice if the hub/shroud tip is defined by a profile that varies significantly from a constant span when viewed in the other transforms.

The Passage and Passage Excluding Tip transforms:

- usually exhibit less distortion than the Full transform,
- are available only after the topology has been created,
- cause geometry objects to be omitted from the blade-to-blade view (even if these transforms are used indirectly via the default transform).



---

# Chapter 7: Viewer

---

- [Introduction \(p. 81\)](#)
- [Viewer Toolbar \(p. 81\)](#)
- [Viewer Hotkeys \(p. 83\)](#)
- [Multiple Viewports \(p. 84\)](#)
- [Selecting and Dragging Objects while in Viewing Mode \(p. 85\)](#)
- [Stereo Viewer \(p. 86\)](#)

## 7.1. Introduction

---

The viewer in Ansys TurboGrid plays a central role in the mesh creation process. Its interactive interface, including the mouse, toolbars and hotkeys, allows you to inspect and alter your work.

The viewer toolbar and hotkeys are described in this chapter. Mouse controls are described in [Viewer Setup: Mouse Mapping \(p. 52\)](#).

---

### Note:



Depending on the graphics card and driver version, you may experience problems with the accuracy of mouse clicks in the viewer. For example, after dragging an inlet point, the point might move to a location far from where you dragged the mouse pointer. If you experience such problems, try lowering the hardware acceleration setting of your graphics card.

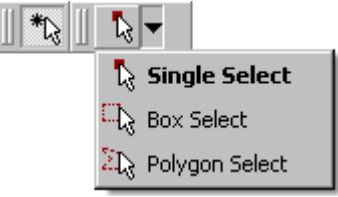












---



## 7.2. Viewer Toolbar

---

The viewer has the following tools:

Tool	Description
	Enters picking mode. To choose the behavior of this mode, use the drop-down arrow on the adjacent toolbar icon (which becomes visible only after you click <i>Select</i>  ):

Tool	Description
	
	<p>This is the <b>Single Select</b> option for the <i>Select</i>  tool. You can use it to select objects or drag certain objects to new locations, using the mouse.</p> <p>When a number of objects overlap, the one closest to the camera is picked. The text at the bottom of the viewer window shows which object would be picked at the mouse's current location. If you cannot pick the object you want because other objects overlap it, turn off the visibility of the overlapping objects, or adjust the camera to make it possible to pick the object.</p>
	<p>This is the <b>Box Select</b> option for the <i>Select</i>  tool. You can use it to select objects using a box. Drag a box around the object(s) you want to select.</p>
	<p>This is the <b>Polygon Select</b> option for the <i>Select</i>  tool. You can use it to select objects using an enclosed polygon. Click to drop points around the object(s). Double-click to complete the selection.</p>
	<p>Rotates the view as you drag with the mouse. Alternatively, hold down the middle mouse button to rotate the view.</p>
	<p>Pans the view as you drag with the mouse. Alternatively, you can pan the view by holding down <b>Ctrl</b> and the middle mouse button.</p>
	<p>Adjusts the zoom level as you drag with the mouse vertically. Alternatively, you can zoom the view by holding down <b>Shift</b> and the middle mouse button.</p>
	<p>Zooms to the area enclosed in a box that you create by dragging with the mouse. Alternatively, you can drag and zoom the view by holding down the right mouse button.</p>
	<p>Centers all visible objects in the viewer.</p>
	<p>Toggles highlighting. Highlighting makes it easier to select the correct object from the viewer. While in picking mode, highlighting by default puts a red box around the object that would be selected at the mouse's current location. The style of highlighting is</p>

Tool	Description
	controlled by <b>Edit &gt; Options &gt; TurboGrid &gt; Viewer &gt; Highlight Type.</b>
	Selects the viewport arrangement; by default, each view displays a different transform. Independent zoom, rotation, and translate options can be carried out in each viewport.
	Displays the <b>Viewer Key Mapping</b> dialog box. See <a href="#">Viewer Hotkeys (p. 83)</a> for details.

## 7.3. Viewer Hotkeys

A number of hotkeys are available to carry out common viewer tasks. Before using a viewer hotkey, place the mouse focus on the viewer window.



Key	Action
<b>Space</b>	Toggle between picking and viewing mode.
<b>Up</b> or <b>Down</b> or <b>Left</b> or <b>Right</b>	Rotate about horizontal and vertical axes.
<b>Ctrl+Up</b> or <b>Ctrl+Down</b>	Rotate about an axis normal to the screen.
<b>Shift+Up</b> or <b>Shift+Down</b> or <b>Shift+Left</b> or <b>Shift+Right</b>	Moves the light source.

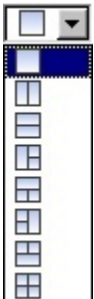
Key	Action
<b>Ctrl+Shift</b>	When in viewing mode, you can hold <b>Ctrl+Shift</b> to select objects in the viewer. Releasing <b>Ctrl+Shift</b> returns you to viewing mode.
<b>1</b>	Switches to one viewport.
<b>2</b>	Switches to two viewports.
<b>3</b>	Switches to three viewports.
<b>4</b>	Switches to four viewports.
<b>c</b>	Centers the graphic object in the viewer window.
<b>n</b>	Toggles the projection between orthographic and perspective.
<b>r</b>	Resets the view to the initial orientation.
<b>u</b>	Undoes transformation.
<b>U</b>	Redoes transformation.
<b>x</b>	Sets view from +X axis.
<b>X</b>	Sets view from -X axis.
<b>y</b>	Sets view from +Y axis.
<b>Y</b>	Sets view from -Y axis.
<b>z</b>	Sets view from +Z axis.
<b>Z</b>	Sets view from -Z axis.

## 7.4. Multiple Viewports

Initially the viewer contains one single viewport. The viewer can be divided into more than one window or multiple viewports, ranging from one to four.

By default, the viewer shows the current objects in a 3D Cartesian view.

The currently active viewport layout is shown in the box to the right of the *Highlighting*  icon on the viewer toolbar. Click  to the right of the current viewport layout picture to select a new viewport layout from the drop-down list on the viewer toolbar.



Change the active viewport by placing the mouse pointer over a viewport and clicking with any of the three mouse buttons. The viewport that contains the mouse pointer is then set as the active viewport.

Mouse-controlled transformations are applied automatically to the viewport that contains the mouse pointer. When the transformation is complete the viewport also becomes the active viewport.

Hard copy plots (postscript or other image file formats) always show all visible viewports in the viewer (a verbatim copy of the viewer window).

If the viewer consists of four viewports, each with object(s), and the viewport layout is changed to have less than four viewports, the graphic in the now "hidden" viewports remains intact, but not visible. When the viewport layout returns to the former layout all graphic objects are present, unchanged.


Each viewport has a default coordinate system.

- Viewport 1: Cartesian
- Viewport 2: 2D Blade-to-blade
- Viewport 3: 2D Meridional
- Viewport 4: 3D Turbo

You can change the coordinate system for each viewport by right-clicking on a blank area in the viewer, and selecting one of the **Transformation** commands.

---

### Note:

If a viewport initially appears to be empty when it is used for the first time, try clicking *Fit View* . This will center the objects and reset the zoom level.

---




## 7.4.1. Selecting, Adding, and Deleting Views

Each viewport can use any of the 4 pre-defined views or user-defined views. To switch between views, use the drop-down menu in the upper-left corner of the viewport. To add a new view based on the current state of the viewer, right-click in the viewer and select **Create New View** from the shortcut menu. To delete an existing user-defined view, right-click in the viewer and select **Delete View** while the view is selected.


## 7.5. Selecting and Dragging Objects while in Viewing Mode

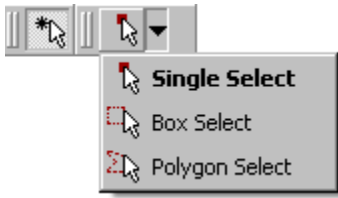
---

*Picking mode* allows you to select and move objects, such as points, curves, and planes.

To enter picking mode, click *Select* . To leave picking mode (that is, switch back to *viewing mode*), click *Rotate*  or *Pan* .

To temporarily enter picking mode when you are in viewing mode, hold **Ctrl+Shift**. When you release the **Ctrl+Shift** keys, you will return to viewing mode.

When you click *Select*  to enter picking mode, a toolbar icon appears to enable you to change the selection method:



For details, see [Viewer Toolbar \(p. 81\)](#).

## 7.6. Stereo Viewer

---

If you:

1. Have a standard stereo display
2. Have a graphics card that supports quad buffering OpenGL output
3. Have set your graphics card to "Stereo"
4. Have set your view to Perspective mode (right-click in the Viewer and select **Projection > Perspective**)

...you can view output in stereo. To enable this functionality:

1. Select **Edit > Options**.
2. In the **Options** dialog box, select **TurboGrid > Viewer**.
3. On the **Viewer** panel:
  - a. Set the Stereo **Mode** to **Stereo**.
  - b. Set the **Stereo Effect**. The value of the "stereo effect" that is required is related to the distance between the observer and the display. If the stereo effect is too strong, either move away from the display, or move the slider towards **Weaker**.
4. Click **OK** to save the settings.

---

# Chapter 8: Tools Menu

---

The following sections describe the commands available in the **Tools** menu:

- 8.1. Calculator Command
- 8.2. Expressions Command
- 8.3. Variables Command
- 8.4. Command Editor Command
- 8.5. Reset Inlet/Outlet Points Command

## 8.1. Calculator Command

---

The built-in calculator provides quantitative information about the geometry and mesh. To view the calculator, select **Tools > Calculator** from the main menu. The **Function Calculator** dialog box is displayed.

---


### Note:

In this release of Ansys TurboGrid, quantitative calculations can only be carried out for the original geometry. Any applied instance transforms are purely visual and do not affect the calculation results.

---

### 8.1.1. Function Calculator Dialog Box

#### 8.1.1.1. Function

Click  to select a function from the drop-down list. The table below outlines the available quantitative functions.

Function Name	Operation
<a href="#">area (p. 88)</a>	Area of location
<a href="#">areaAve (p. 88)</a>	Area-weighted average
<a href="#">areaInt (p. 88)</a>	Area-weighted integral (can be projected to a direction)
<a href="#">ave (p. 89)</a>	Arithmetic average
<a href="#">count (p. 89)</a>	Number of calculation points
<a href="#">length (p. 89)</a>	Length of a curve
<a href="#">lengthAve (p. 89)</a>	Length-weighted average
<a href="#">lengthInt (p. 90)</a>	Length-weighted integration

Function Name	Operation
<a href="#">maxVal (p. 90)</a>	Maximum Value
<a href="#">minVal (p. 90)</a>	Minimum Value
<a href="#">probe (p. 90)</a>	Value at a point
<a href="#">sum (p. 90)</a>	Sum over the calculation points
<a href="#">volume (p. 91)</a>	Volume of a 3D location
<a href="#">volumeAve (p. 91)</a>	Volume-weighted average
<a href="#">volumeInt (p. 91)</a>	Volume-weighted integral

### 8.1.1.1.1. area

The `area` function is used to calculate the area of a 2D location. The following example demonstrates use of the function.

- **Function:** `area`, **Location:** `Plane1`. This example calculates the total area of the locator `Plane1`.

### 8.1.1.1.2. areaAve

The `areaAve` function calculates the area-weighted average of an expression on a 2D location. The area-weighted average of a variable is the average value of the variable on a location when the mesh element sizes are taken into account. Without the area weighting function, the average of all the nodal variable values would be biased towards variable values in regions of high mesh density. The following examples demonstrate use of the Function.

- **Function:** `areaAve`, **Location:** `Outlet`, **Variable:** `Velocity`. This example calculates the average magnitude of the velocity on the `outlet` location. Note that flow direction is not considered since the magnitude of a vector quantity at each node is calculated. Use the scalar components of velocity (for example, `Velocity u`) to include a directional sign, for example:
- **Function:** `areaAve`, **Location:** `Outlet`, **Variable:** `max(Velocity u, 0.0[m s-1])`. This example calculates the area-weighted average value of `Velocity u`, with negative values of the variable replaced by zero. Note that this is not the average positive value since zero values contribute to the average.

### 8.1.1.1.3. areaInt

The `areaInt` function integrates a variable over the specified 2D location. To perform the integration over the total face area, the **None** option should be selected from the Direction drop-down list. If a direction is selected, the result is an integration over the projected area of each face onto a plane normal to that direction. Each point on a location has an associated area that is stored as a vector and therefore has direction. By selecting a direction in the calculator you are using only a single component of the vector in the area-weighting function. Since these components can be positive or negative, depending on the direction of the normal on the location, it is possible for areas to cancel out. An example of this would be on a closed surface where the projected area is always zero (the results returned are not in general zero since the variable values differ over the closed surface). On a flat surface the normal vectors always point in the same direction and never cancel out. The following examples demonstrate use of the function.

- **Function:** `areaInt`, **Location:** `Plane1`, **Variable:** `Pressure`, **Direction:** `None` This example integrates pressure over `Plane1`. The result returned is the total pressure force acting on `Plane1`. The magnitude of each area vector is used and so the direction of the vectors is not considered.
- **Function:** `areaInt`, **Location:** `Plane1`, **Variable:** `Pressure`, **Direction:** `Global X`. This example integrates pressure over the projected area of `Plane1` onto a plane normal to the X axis. The result is the pressure force acting in the X direction on `Plane1`. This differs slightly from using the force function to calculate the X-directional force on `Plane1` — the force function includes forces due to the advection of momentum when calculating the force on an internal arbitrary plane or a non-wall boundary (such as inlets).

#### 8.1.1.1.4. ave

The `ave` function calculates the arithmetic average (the mean value) of a variable or expression on the specified location. This is the sum of the values at each node on the location divided by the number of nodes. Results are biased towards areas of high nodal density on the location. To obtain a mesh-independent result, use the `lengthAve`, `areaAve`, `volumeAve` or `massFlowAve` functions. The following example demonstrates use of the function.

The average of a vector value is calculated as an average of its magnitudes, not the magnitude of component averages. As an example, for velocity,  $|v|_{\text{ave}} = \frac{|v_1| + |v_2|}{2}$

$$\text{where } |v_i| = \sqrt{(v_{xi}^2 + v_{yi}^2 + v_{zi}^2)}$$

- **Function:** `ave`, **Location:** `MainDomain`, **Variable:** `Temperature`. This example calculates the mean temperature at all nodes in the selected domain.

#### 8.1.1.1.5. count

The `count` function returns the number of nodes on the specified location. The following example demonstrates use of the function.

- **Function:** `count`, **Location:** `MainDomain`. This example returns the number of nodes in the specified domain.

#### 8.1.1.1.6. length

Computes the length of the specified line as the sum of the distances between the points making up the line. The following example demonstrates use of the function.

- **Function:** `length`, **Location:** `Polyline1`. Calculates the length of the Polyline.

#### 8.1.1.1.7. lengthAve

Computes the length-based average of the variable on the specified line. This is the 1D equivalent of the `areaAve` function. The result is independent of the nodal distribution along the line since a weighting function assigns a higher weighting to areas of sparse nodal density. The following example demonstrates use of the function.

- **Function:** `lengthAve`, **Location:** `Polyline1`, **Variable:** `Velocity`. This calculates the average velocity on the location `Polyline1` using a length-based weighting function to account for the distribution of points along the line.

#### 8.1.1.1.8. `lengthInt`

Computes the length-based integral of the variable on the specified line. This is the 1D equivalent of the `arealnt` function. The following example demonstrates use of the function.

#### 8.1.1.1.9. `maxVal`

Returns the maximum value of the specified variable on the specified locator. Create a user variable if you want to find the maximum value of an expression. The following example demonstrates use of the function.

- **Function:** `maxVal`, **Location:** `Default`, **Variable:** `Yplus`. This returns the maximum `Yplus` value on the `Default` wall boundaries.

#### 8.1.1.1.10. `minVal`

Returns the minimum value of the specified variable on the specified locator. Create a user variable if you want to find the minimum value of an expression. The following example demonstrates use of the function.

- **Function:** `minVal`, **Location:** `MainDomain`, **Variable:** `Temperature`. These settings return the minimum temperature in the domain.

#### 8.1.1.1.11. `probe`

Returns the value of the specified variable on the specified point object. The following example demonstrates use of the function.

- **Function:** `probe`, **Location:** `Point1`, **Variable:** `Density`. Returns the density value at `Point1`.

---

**Important:**

This calculation should only be performed for point locators described by single points. Incorrect solutions are produced for multiple point locators.

---

#### 8.1.1.1.12. `sum`

Computes the sum of the specified variable values at each point on the specified location. The following example demonstrates use of the function.

- **Function:** `sum`, **Location:** `SubDomain1`, **Variable:** `Volume of Finite Volume` Returns the sum of the finite volumes assigned to each node in the location `SubDomain1`. In this case this sums to the volume of the subdomain.

### 8.1.1.1.13. volume

The `volume` function is used to calculate the volume of a 3D location. The following example demonstrates use of the function.

- **Function:** `volume`, **Location:** `Volume1`. Returns the sum of the volumes of each mesh element included in the location `Volume1`.

### 8.1.1.1.14. volumeAve

The `volumeAve` function calculates the volume-weighted average of an expression on a 3D location. This is the 3D equivalent of the `areaAve` function. The volume-weighted average of a variable is the average value of the variable on a location weighted by the volume assigned to each point on a location. Without the volume weighting function, the average of all the nodal variable values would be biased towards values in regions of high mesh density. The following example demonstrates use of the function.


- **Function:** `volumeAve`, **Location:** `Volume1`, **Variable:** `Density`. This example calculates the volume-weighted average value of density in the region enclosed by the location `Volume1`.

### 8.1.1.1.15. volumeInt


The `volumeInt` function integrates the specified variable over the volume location. This is the 3D equivalent of the `areaInt` function. The following example demonstrates use of the function.

- **Function:** `volumeInt`, **Location:** `Volume1`, **Variable:** `Density`. This calculates the integral of density (the total mass) in `Volume1`.

## 8.1.1.2. Location

Click  to select a location from the drop-down list. Only locations valid for the selected function are available.

## 8.1.1.3. Variable

Click  to select a variable from the drop-down list. Only variables valid for the selected function are available.

For most functions, click in the **Variable** box and enter an expression to use as the variable. The expression can include other variables and any valid CEL (Ansys CFX Expression Language) function (see [CEL Functions, Constants and System Variables in the TurboGrid Reference Guide](#)). For example, `abs(Velocity u)` could be entered so that the calculation is performed using the absolute values of the variable `Velocity u`.

## 8.1.1.4. Direction

The `areaInt` function requires a direction to be specified before the calculation can be performed. The `areaInt` function projects the location onto a plane normal to the specified direction (if the direction is not set to `None`), and then performs the calculation on the projected location (direction specification can also be `None`). The direction of the normal vectors for the location is important and cancels out for surfaces such as closed surfaces.

### 8.1.1.5. Hybrid and Conservative Variables

In **Ansys TurboGrid** there is no difference between hybrid and conservative variables. Leave all controls for selecting between them at their default values.

## 8.2. Expressions Command

The expression editor is used to create new expressions and modify existing expressions in Ansys TurboGrid. Ansys TurboGrid can use expressions in place of any numeric value (as long as the correct units are returned by the expression). For example, you can use expressions to define object properties or new variables.

### Note:

- When a setting is defined by an expression, and the latter evaluates to a quantity that has no units, the software internally applies the default units for that setting.
- In an expression, a term that has no units can be added to a term that has angular units, in which case the software internally applies radians to the term that has no units.

To create or edit an expression, select **Tools > Expressions** from the main menu. The **Expression Editor** dialog box is displayed.

### 8.2.1. Expression Editor Dialog Box

The **Expression Editor** dialog box contains:

- A tree view for viewing and managing expressions (The tree view has a shortcut menu with commands for creating/deleting/modifying expressions, as described below.)
- A details view for creating/editing/displaying the definition of the new/selected expression

(You can find a list of valid CEL expressions and constants in [Ansys CFX Expression Language in the TurboGrid Reference Guide](#).)

- A **Value** box to show the value of the selected expression when the expression does not contain variables and evaluates to a single value.

You can right-click an existing expression or a blank area in the tree view portion of the **Expression Editor** dialog box in order to access a shortcut menu with the following commands:

**Table 8.1: Expression Editor Shortcut Menu Commands**

Command	Description
Edit	Displays the definition of the selected <b>Expression</b> in the details view portion of the <b>Expression Editor</b> dialog box, and gives it the keyboard focus. An alternative way to do the same is to double-click an expression in the tree view.

Command	Description
Insert > Expression	<p>Opens the <b>New Expression</b> dialog box where you can enter a name for a new expression. After you enter a name, you must provide and apply a definition in order to finish creating the new expression.</p> <p>When selecting an expression name:</p> <ul style="list-style-type: none"> <li>You cannot create an expression with the same name as an object.</li> <li>You cannot create an expression with the same name as a variable.</li> </ul>
Edit in Command Editor	Invokes the <b>Command Editor</b> dialog box populated with the selected expression.
Delete	Deletes the selected expression.
Duplicate	Creates a duplicate of the selected expression after obtaining the name of the new expression from the <b>New Expression</b> dialog box.
Use as Workbench Input Parameter <sup>[a]</sup>	<p>Makes the selected expression a Workbench input parameter so that it is available in, and controllable by, Ansys Workbench.</p> <p>The Workbench input parameter is initialized by the expression definition specified in Ansys TurboGrid. The definition must be a function of only constants; units are permitted.</p> <p>It is not possible to edit the value of an expression in Ansys TurboGrid while the expression is a Workbench input parameter.</p>
Use as Workbench Output Parameter <sup>[a]</sup> <sub>[b]</sub>	Makes the selected expression a Workbench output parameter, causing it to be output from Ansys TurboGrid for use in Ansys Workbench.
Deselect as Workbench Parameter <sup>[a][b]</sup>	Removes the Ansys Workbench parameter status from the selected expression.

<sup>[a]</sup> You can enable/disable the use of a Workbench input parameter directly for many settings in the object editor. For details, see [Object Editor \(p. 22\)](#).

<sup>[b]</sup> For details on Workbench parameters, see [Working with Parameters and Design Points in the Workbench User's Guide](#)

When defining an expression:

- You cannot use a user variable.
- Constants require units inside square brackets.

- You can right-click in the details view to access a shortcut menu for inserting variables, locators, expressions, and functions into your equation.

### 8.2.1.1. Expression Editor Example

To define an expression for distance from the X axis:

- Select **Tools > Expressions** from the main menu to open the **Expression Editor** dialog box.
- When the **New Expression** dialog box appears, enter the name `radial` and click **OK**.
- In the details view portion of the **Expression Editor** dialog box, enter the following expression definition:

```
sqrt ( Y^2+Z^2 )
```

- Click **Apply** to create the expression.

Note that the **Value** box does not show a value for the expression because the expression does not evaluate to a single value.

The expression created in this example (`radial`) is used in the [Variable Editor Example \(p. 96\)](#).







## 8.3. Variables Command


The variable editor is used to create new user variables and modify existing variables in Ansys TurboGrid.

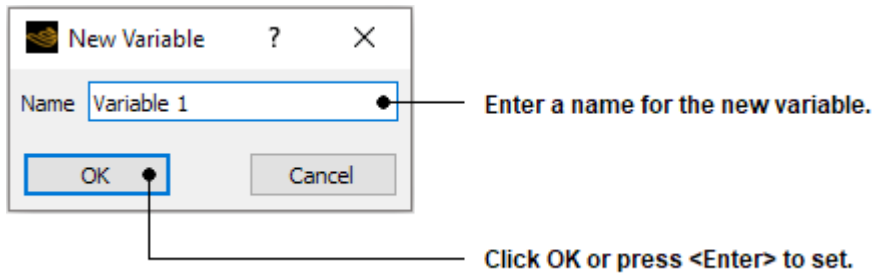
To create or edit a variable, select **Tools > Variables** from the main menu. The **Variable Editor** dialog box is displayed.

### 8.3.1. Variable Editor Dialog Box

All icons become active when you select a variable from the **Variable** list. Alternatively, you can right-click a variable in the list to see the same options. Each icon and its function is described in the following table.

Icon	Description
	<i>New</i> opens the <b>New Variable</b> dialog box where you can enter a name.
	<i>Edit</i> displays the selected <b>Variable</b> in the <b>Expression</b> and/or <b>Units</b> box where you can edit it.
	<i>Copy</i> makes a duplicate of the selected variable and opens the <b>New Variable</b> dialog box where you can enter a name for the copy.
	<i>Delete</i> removes the selected variable from the list.
	<i>Use Hybrid Values</i> sets all variables to hybrid values.
	<i>Use Conservative Values</i> sets all variables to conservative values.

When you click *New*  to create a new variable, the **New Variable** dialog box is displayed.






### 8.3.1.1. Name

There are a few guidelines to follow when selecting a variable name.

- You cannot create a variable with the same name as an object.
- You cannot create a variable with the same name as an expression. For example, if you have an expression named `Radius`, you must choose a different variable name for that expression.
- Within the Ansys CFX Expression Language some variables are known by short names to save typing in the full variable name. For example, `p` refers to `Pressure`. Although it is possible to create a variable with the same name as an abbreviated variable, it is ignored. For example, if you use a variable with the name `p` in an expression, it returns the value `Pressure` in all cases, no matter what the definition of the variable is.

### 8.3.1.2. Type

Click *Edit*  to edit both the fundamental and user variables. Select the expression used to define the variable from a list of existing expressions for user variables. The expressions available from this list are those which you have created in the expression editor. For details, see [Expressions Command \(p. 92\)](#).

For fundamental variables, the units are changeable. Click  next to the Units box to see the available units for the selected variable. This means, for example, that you could create a legend which uses alternative angle units (such as degrees or radians) by clicking on the  icon and selecting new units.

---

#### Note:

These settings override the global units setting, defined in the Edit Menu. For details, see [Options Command \(p. 47\)](#).

---


The variable type used affects all quantitative calculations and plots in Ansys TurboGrid.

### 8.3.1.3. Hybrid and Conservative Variable Values

This is useful only for advanced postprocessing and is not relevant for Ansys TurboGrid. It is included to maintain consistency with other Ansys CFX products. Refer to the CFD-Post documentation for more information on the differences between hybrid and conservative variable values.

### 8.3.1.4. Variable Editor Example

In this example, an isosurface that has a fixed radial distance from the X axis is created using the expression defined in the [Expression Editor Example \(p. 94\)](#).

1. Select **Tools** > **Variables** from the main menu to open the variable editor.
2. Click **New**  to create a new variable. When the **New Variable** dialog box appears enter the name `Radial Distance` and click **OK**.
3. In the variable editor, use the **Expression** drop-down list to select the expression `radial` which you created earlier. Click **Apply** to create the new variable.

This variable now appears in the list of available variables and can be used like any other variable. Notice that the variable type is listed as **User**.

You can now create an isosurface using this variable. For details on creating isosurfaces, see [Isosurface Command \(p. 67\)](#).

1. Select **Insert** > **User Defined** > **Isosurface** from the main menu, enter a name, then click **OK** on the **New Isosurface** dialog box.
2. In the **Geometry** tab for the isosurface, set **Variable** to `Radial Distance`.
3. Set **Value** to `20 m`. This is a suitable value for the Rotor 37 geometry used in Tutorial 1. You may need to alter this value to something more sensible depending on the geometry you are viewing.
4. Click the **Color** tab and set **Mode** to `Variable`.
5. Select a sensible variable (for example, `Maximum Face Angle` or `Axial Distance`) with which to color the isosurface.
6. Set **Range** to `Local` so that the full color range is used on the isosurface.
7. Click **Apply** to create the isosurface object.

You should now see the isosurface in the viewer. All points on the isosurface are a distance of 20 m (or whatever value you used in the **Value** box) from the X-axis.

## 8.4. Command Editor Command

The **Command Editor** dialog box can be used to create or modify any of the objects in Ansys TurboGrid using the CFX Command Language. For further details, see [CFX Command Language in the TurboGrid Reference Guide](#).

Power Syntax can be entered and processed in the **Command Editor** dialog box. For details, see [Power Syntax in the TurboGrid Reference Guide](#). Power Syntax commands should be preceded by the ! symbol.

In addition, any valid CCL command can be typed and processed in the **Command Editor** dialog box. For details, see [Command Actions in the TurboGrid Reference Guide](#). Action Commands should be preceded by the > symbol.

To create an object using CCL, select **Tools > Command Editor** from the main menu. The **Command Editor** dialog box is displayed.

To edit an existing object using CCL, right-click the object in the object selector and select **Edit in Command Editor** from the shortcut menu. The CCL definition of that object is automatically displayed and can be edited to alter the object properties.

You can access the following basic editing tools by right-clicking in the **Command Editor** dialog box:

- **Undo**

Undoes the last edit action.

- **Redo**

Redoes the most recently undone edit action.

- **Cut**

Cuts the selected text and places it on a clipboard.

- **Copy**

Places the selected text on a clipboard.

- **Paste**

Pastes the clipboard text at the insertion point, or replaces the selection.

- **Clear**

Clears all of the contents of the **Command Editor** dialog box.

- **Select All**

Selects all of the contents of the **Command Editor** dialog box.

- **Find**

Makes a search tool appear at the bottom of the **Command Editor** dialog box. Enter a search term and click either **Next** or **Previous** to search upwards or downwards from the insertion point or text selection. To hide the search tool, press <Esc>.

## 8.5. Reset Inlet/Outlet Points Command

---

This command moves the inlet and outlet points to their default locations.



---

# Chapter 9: Help Menu

---

The **Help** menu has the following commands:

## **TurboGrid**

Opens [TurboGrid User's Guide \(p. 1\)](#).

## **Contents**

Opens a page that lists various help resources associated with this product.

## **[various help resources]**

Each of these commands goes directly to a particular help resource.

## **Ansys Product Improvement Program**

Provides a brief description of, and enables you to control participation in, the Ansys Product Improvement Program.

## **About TurboGrid**

This gives the point releases and software patches that are installed.

## **Help on Help**

Opens documentation about the help system: [Ansys TurboGrid Help and Conventions in the TurboGrid Introduction](#).



---

# Chapter 10: Ansys TurboGrid Workflow

---

This chapter describes:

- 10.1. Introduction
- 10.2. Steps to Create a Mesh
- 10.3. Geometry
- 10.4. Topology
- 10.5. Mesh Data
- 10.6. Layers
- 10.7. 3D Mesh
- 10.8. Mesh Analysis
- 10.9. User Defined Objects
- 10.10. Default Instance Transform
- 10.11. Shortcut Menu Commands

## 10.1. Introduction

---

This chapter explains the steps required to create a mesh. A brief overview is given in the next section. The remaining sections of this chapter cover each major step in detail.

## 10.2. Steps to Create a Mesh

---

To create a mesh:

1. Define the geometry.  
For details, see [Geometry \(p. 103\)](#).
2. Define the topology object(s) by selecting a method and optionally changing other settings.  
For details, see [Topology \(p. 149\)](#).
3. Optionally modify the `Mesh Data` object to adjust the number of nodes.

If you plan to make a fine (high-resolution) mesh, you can optionally set the mesh density at a later time in order to minimize processing time while establishing the topology. Keep in mind that changing the mesh density can affect the mesh quality.

For details, see [Mesh Data \(p. 166\)](#).

4. Optionally modify the layer objects to improve mesh quality.

For details, see [Layers](#) (p. 183).

5. Generate the 3D mesh.

By default, the 3D Mesh object is initially unsuspended, so mesh generation happens automatically whenever the Layers object is (re)processed. For details, see [3D Mesh](#) (p. 188).

6. Optionally investigate the mesh and refine any of the above objects as necessary.

To help identify problem areas of the mesh, mesh analysis tools are available. For details, see [Mesh Analysis](#) (p. 189).

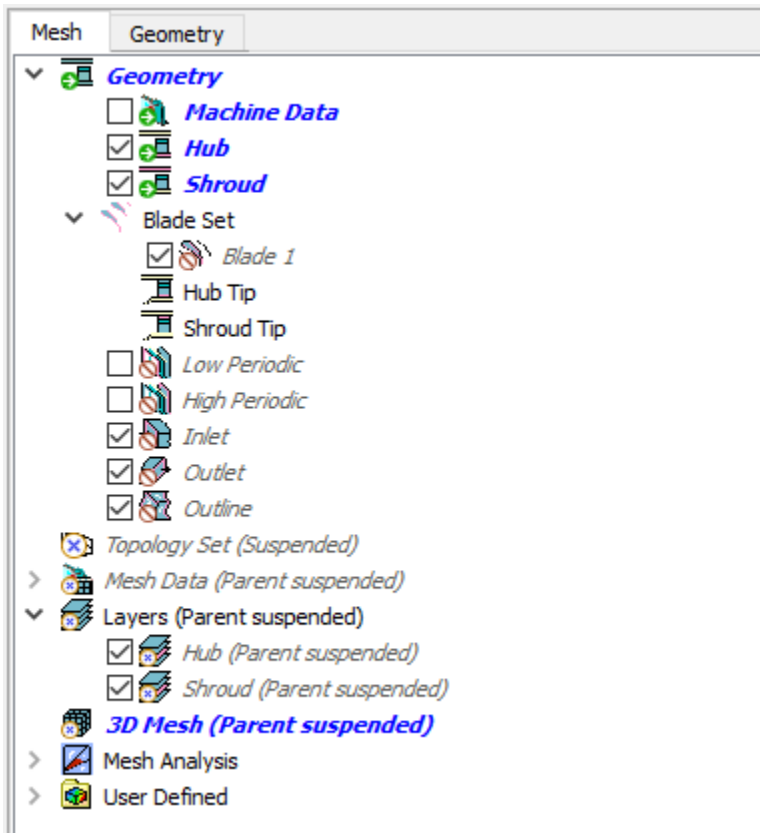
If you change any objects that affect the mesh, you must generate the mesh again.

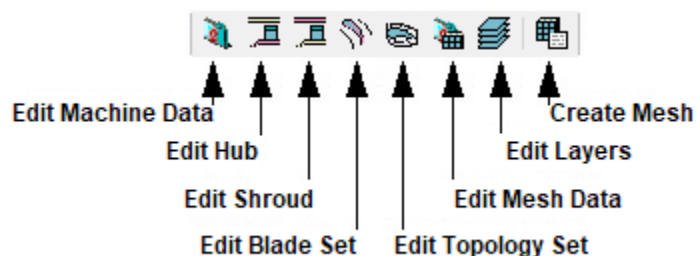
7. Save the mesh to a file.

For details, see [Save Mesh Command](#) (p. 38).

In general, you should proceed from top to bottom in the object selector, or from left to right on the toolbar.

**Figure 10.1: Object Selector**



**Figure 10.2: Part of Toolbar**

The other objects in the object selector are used for visualization and inspection purposes.



Not every object in the object selector requires a definition before a mesh can be created. However, every object represented by an icon on the toolbar must be defined.

## 10.3. Geometry

Geometry can be defined using two types of source data:

- CAD data (for example, from BladeEditor or a Parasolid file)
- Profile point data (for example, from BladeEditor or BladeGen)

The geometry of the blades and main passage can be made using either type of source data. The geometry of any secondary flow paths must be made using CAD data.

After you have finished creating the geometry, you may want to hide some or all of it in the **Mesh** workspace to allow you to see the topology clearly. You can control the visibility of a geometry object by toggling the check box next to the object in the object selector. You can also right-click an object and use shortcut menu commands. The toolbar allows you to hide and unhide all geometry objects via the *Hide all geometry objects*  and *Unhide\_geometry\_objects*  icons.

The following topics are discussed.

[10.3.1. Defining Geometry from a CAD Source](#)

[10.3.2. Defining Geometry from Profile Points](#)

[10.3.3. Defining Geometry from Profile Points and a CAD Input File](#)

[10.3.4. Defining Geometry from Generated CAD: CAD From NDF](#)

[10.3.5. Defining Geometry from Generated CAD: CAD From Profile Points](#)

[10.3.6. The Geometry Objects in the Mesh Workspace](#)

### 10.3.1. Defining Geometry from a CAD Source

CAD data can be used for the geometry of the blades and main passage, and for secondary flow paths.

You can load CAD data from:

- A BladeEditor Geometry cell in Workbench, or

- A Parasolid or other CAD file in stand-alone mode.

---

**Note:**

If TurboGrid crashes while loading CAD geometry, there might be a failure in creating internal representations of the CAD blade surfaces for meshing. A possible workaround is to perform one or more of the following steps, using the **Command Editor** dialog box to make CCL changes:

- Before loading the geometry set `Stop Before Blade Approximate Surface Generation` in `BLADE SET` to `true`.
- Load the geometry.
- Increase `Surface Parameterization Tolerance Multiplier` in `BLADE SET` to `10.0`.
- Increase `Surface Parameterization Number Of Splits` in `BLADE SET` to `200`.
- Change `Parametric Surface Generation Smoother` in the Blade object(s) to **Angle-based**.
- Change `Display Trimmed Surfaces` in the Blade object(s) to `false`.
- Reset `Stop Before Blade Approximate Surface Generation` in `BLADE SET` to `false`.

---

The following topics are discussed.

- 10.3.1.1. Loading CAD Data From BladeEditor
- 10.3.1.2. Loading CAD Data From a CAD Input File
- 10.3.1.3. Objects in the Geometry Workspace (CAD Mode)
- 10.3.1.4. Geometry Browser Settings (CAD Mode)
- 10.3.1.5. Limitations of CAD Geometry Usage

### 10.3.1.1. Loading CAD Data From BladeEditor

To load CAD data from a BladeEditor Geometry cell:

1. In Ansys Workbench, attach a Geometry cell (containing a BladeEditor geometry) upstream of the Turbo Mesh cell.
2. Specify which exported blade(s) to load into TurboGrid by editing the Turbo Mesh cell properties. See [TurboGrid Turbo Mesh Cell Properties in the TurboSystem User's Guide](#) for more information.

---

**Note:**

- This method is applicable only for native BladeEditor blades (not general CAD geometry imported into DesignModeler). If a geometry that is to be imported from BladeEditor

fits within a cube of edge length 10 cm, the DesignModeler session unit (see [Units Menu in the Ansys DesignModeler User's Guide](#)) should be set to **Micrometer** for better geometry precision. For details, see [Export to Ansys TurboGrid in the TurboSystem User's Guide](#).

- When a blade is passed from BladeEditor to TurboGrid (so that a CFD mesh can be made by TurboGrid), two versions of the geometry are passed:
  - The "display geometry", which is the true geometry designed in BladeEditor, and
  - The "meshing geometry", which is used by TurboGrid in constructing the mesh.

The meshing geometry is slightly different from the display geometry because it must meet TurboGrid's mesh generation requirements. For example, the meshing geometry has extended surfaces that overlap so that TurboGrid can reliably locate surface intersections.

In the **Mesh** workspace, the display geometry can be visually compared with the mesh. In the **Geometry** workspace, you can view both the display geometry and the meshing geometry by turning on visibility of the corresponding CAD objects listed in the tree.

In particular, for blades with blends (fillets) from BladeEditor, there will typically be a noticeable difference between the mesh and the display geometry shown in the **Mesh** workspace, and a noticeable difference between the meshing geometry and display geometry shown in the **Geometry** workspace. The overall difference in fillet geometry (for example, the amount of flow blockage) can be affected by changing fillet-related parameters in BladeEditor.

---

### 10.3.1.1.1. Troubleshooting Blends (Fillets) from BladeEditor

If you have a geometry defined by profile points and you want to use BladeEditor to add blends using the rolling ball process with a constant radius, consider using the CAD From Profile Points input mode and add a blade blend object in the **Mesh** workspace. For details, see [Defining Geometry from Generated CAD: CAD From Profile Points \(p. 112\)](#) and [Blade CAD Features \(p. 132\)](#).

For other types of blade blends created in BladeEditor, the following troubleshooting notes apply:

- If you experience a problem with mesh quality near a cut-off edge (LE or TE) and near a blade fillet, then adjusting the following CCL parameters might help:
  - Number Of Hub To Shroud Constant Span Splits
  - Mesh Curves Tesselation Number Of Power Samples = 8
  - Mesh Curves Tesselation Number Of Power Samples Override = true
- If a blade fillet looks wavy where it meets the hub/shroud (more likely for cases with high hub/shroud curvature), increasing the following CCL parameters might help:
  - Turbo Transform Background Mesh Size

- Turbo Transform Background Mesh Size For Topology

### 10.3.1.2. Loading CAD Data From a CAD Input File

To load CAD data from a CAD input file (for example, a Parasolid file):

- In TurboGrid in stand-alone mode, in the **Geometry** workspace, open the geometry browser, set **Input Mode** to CAD (or Profile Points With Secondary CAD) and configure the applicable settings in the geometry browser.

---

**Note:**

When a CAD blade is loaded from a file into TurboGrid, two versions of the geometry might be displayed:

- When available, the "display geometry", which is the best representation of the true geometry, is displayed in the **Geometry** workspace.
- The "meshing geometry", which is used by TurboGrid in constructing the mesh, is displayed in the **Mesh** workspace.

The meshing geometry is slightly different from the display geometry because it must meet TurboGrid's mesh generation requirements. For example, the meshing geometry has extended surfaces that overlap so that TurboGrid can reliably locate surface intersections.

In the **Mesh** workspace, the display geometry can be visually compared with the mesh. In the **Geometry** workspace, you can view both the display geometry (when available) and the meshing geometry by turning on visibility of the corresponding CAD objects listed in the tree.

---

### 10.3.1.3. Objects in the Geometry Workspace (CAD Mode)


The **Geometry** workspace tree holds CAD objects and can be used to visually check that the correct CAD geometry is imported.

The main objects in the **Geometry** workspace are:

The Topological Entity Instances Object	This object holds topological entities, each of which defines a key geometric feature needed by TurboGrid. The topological entities refer to CAD objects, either directly or through child objects.
The CAD Families Object	This object holds CAD objects that each define a geometric feature that was imported from a CAD file. You can select different CAD objects to view them in the <b>3D Viewer</b> . Specific objects from the CAD file can be used by

TurboGrid after they are assigned to topological entities.

Some topological entities can be edited in the object editor. To open the object editor, double-click an object in the tree or select an object and then press **Enter**. In particular:

- Each topological entity that can refer to CAD objects has, in its object editor, a **Location** setting for assigning CAD objects. Beside this setting is a *CAD Entity Selector*  icon for opening the **CAD Region Entities** dialog box, which you can use to select CAD objects.

Note that an alternative method for assigning CAD objects to a topological entity is to select CAD objects from the *CAD Families* branch of the tree and use the shortcut menu.

- The blade topological entity has an object editor, with self-explanatory settings **Blade Name**, **Leading Edge Type**, and **Trailing Edge Type**.
- The *ComplexBladeEnd* topological entity has an object editor. For details, see [Complex Blade End \(p. 223\)](#).

### 10.3.1.4. Geometry Browser Settings (CAD Mode)

To access the geometry browser, switch to the **Geometry** workspace and then, if necessary, double-click a high-level object (either the *Topological Entity Instances* object or the *CAD Families* object) in the object selector.

The geometry browser CAD-related settings, which are available for the *CAD* and *Profile Points With Secondary CAD* input modes, are described next:

#### 10.3.1.4.1. CAD Input Definition

#### 10.3.1.4.2. Geometry Setup (CAD Mode)

##### 10.3.1.4.1. CAD Input Definition

Set **File Name** to the name of a CAD file (for example, \*.x\_b exported from BladeEditor).

Optionally select **Named Selection Processing** to have TurboGrid create CAD families for named selections in the CAD file.

Optionally select **Specify CAD Geometry Units** and specify units for **CAD Length Units**. If you do not specify units, TurboGrid will attempt to determine the units from the file contents; in such cases you should confirm that the units are correct by observing the resulting geometry in the **3D Viewer**.

##### 10.3.1.4.2. Geometry Setup (CAD Mode)

###### 10.3.1.4.2.1. Rotation (CAD Mode)

These settings are similar to the **Rotation** settings of the Machine Data object. For details, see [Rotation \(p. 119\)](#).

#### 10.3.1.4.2.2. # of Bladesets (CAD Mode)

A rotating machine component is made up of adjacent blades that are equally spaced around the circumference of the machine. The Theta extent of one blade set is calculated as 360 degrees divided by the number of main blades. Many rotating machine components have secondary and tertiary blades that are placed between the main blades. These are often called splitter blades. A blade set contains one main blade and optional splitter blades that repeat cyclically around the axis of the rotating machine component.

For rotating machine components without any splitter blades, the number of blade sets equals the total number of blades.

TurboGrid creates a mesh for one blade set only. The mesh can be copied and rotated using an Ansys CFX Pre-processor, if necessary, before it is solved in an Ansys CFX Solver.

#### 10.3.1.5. Limitations of CAD Geometry Usage

TurboGrid uses a specific set of geometric inputs. Because CAD geometry is more general, there are some CAD features that cannot be used by TurboGrid. A list of some such features follows:

- You cannot transform imported CAD geometry.
- Inlet and outlet trimming is governed by selected CAD curves.

#### 10.3.2. Defining Geometry from Profile Points

Profile point data can be used for the geometry of the blades and main passage.

---

**Note:**

- If you plan to add blade blends (fillets) that use the rolling ball process with a constant radius, consider using the `CAD From Profile Points` input mode. For details, see [Defining Geometry from Generated CAD: CAD From Profile Points \(p. 112\)](#).
- Profile point data cannot be used to define secondary flow paths. However, you can add secondary flow paths from a CAD source by using the `Profile Points With Secondary CAD` input mode.

---

To define the geometry in TurboGrid using profile point data, you can do any of the following:

- Load profile point data from a BladeEditor Geometry cell (in Workbench):
  1. In BladeEditor, create an `ExportPoints` feature. For details, see [Export to Ansys TurboGrid in the TurboSystem User's Guide](#).
  2. In Workbench, attach a Geometry cell upstream of the Turbo Mesh cell.
  3. Set the Turbo Mesh cell properties. For details, see [TurboGrid Turbo Mesh Cell Properties in the TurboSystem User's Guide](#).

4. In TurboGrid, in the **Geometry** workspace, open the geometry browser, set **Input Mode** to `Profile Points` (or `Profile Points With Secondary CAD`) and configure the applicable settings in the geometry browser.
5. Review the settings in the `Geometry` objects in the **Mesh** workspace, making and applying changes as required.

For details, see [The Geometry Objects in the Mesh Workspace \(p. 118\)](#).

- Load profile point data from a BladeGen Blade Design cell (in Workbench):
  1. In Ansys Workbench, attach a Blade Design cell upstream of the Turbo Mesh cell.
  2. In TurboGrid, review the settings in the geometry browser in the **Geometry** workspace, making and applying changes as required.


Note that **Input Mode** should be set to `Profile Points`.

3. Review the settings in the `Geometry` objects in the **Mesh** workspace, making and applying changes as required.

For details, see [The Geometry Objects in the Mesh Workspace \(p. 118\)](#).

Note that, in this case, some of the geometry objects in the **Mesh** workspace (such as Machine Data, Hub, Shroud, Blade Set, and Blade) have their settings taken from the upstream cell, and changes to these settings cannot be made in Ansys TurboGrid. Other settings in geometry objects are taken from the upstream cell by default, but can be overridden. To override a particular setting, select the appropriate check box in the object editor or the geometry browser. For example, although the machine rotation axis is, by default, taken from the upstream cell, you can reverse the rotation axis direction by first selecting the **Override Theta Direction for Topology** check box in the **Machine Data** object editor or the geometry browser, and then selecting **Right Handed** or **Left Handed**.

- Load profile point data from a BladeGen \*.inf file:

1. Select **File > Load TurboGrid Init File** or click *Load TurboGrid Init File*  then select and open an \*.inf file.

The \*.inf file contains machine data, the names of geometry definition files, the choice of coordinate system, and leading/trailing edge settings. For details on \*.inf files, see [BladeGen.inf File \(p. 30\)](#).

2. Review the settings in the geometry browser in the **Geometry** workspace, making and applying changes as required. Note that **Input Mode** should be set to `Profile Points`.
3. Review the settings in the `Geometry` objects in the **Mesh** workspace, making and applying changes as required. For details, see [The Geometry Objects in the Mesh Workspace \(p. 118\)](#).

- Load profile point data from separate curve (.curve or .crv) files:

1. Select **File > Load Profile Points Or CAD** or click *Load Profile Points Or CAD* .

2. In the geometry browser in the **Geometry** workspace, specify curve files and make and apply other changes as required. Note that **Input Mode** should be set to `Profile Points`.
3. Review the settings in the `Geometry` objects in the **Mesh** workspace, making and applying changes as required. For details, see [The Geometry Objects in the Mesh Workspace \(p. 118\)](#).

For details on curve files, see [Curve File \(p. 31\)](#).

- In the **Mesh** workspace, define the `Machine Data` object, then each of the remaining geometry objects separately.

Note that, in the geometry browser in the **Geometry** workspace, **Input Mode** must be set to `Profile Points`.

Using the geometry browser with the `Profile Points` input mode is a convenient way to define multiple geometry objects, compared to editing each geometry object separately in the **Mesh** workspace. It is useful when you first open TurboGrid and when you want to use the same settings on a slightly modified geometry.

The information specified in the geometry browser is used to overwrite the corresponding information in the appropriate `Geometry` objects in the **Mesh** workspace. No other previously-defined settings are affected. For example, if a control angle is defined for the `Hub` object, it will not be changed by using the geometry browser.

The curve file for the blades can specify one or more blades for the blade row; one blade is generated for each blade defined in the curve file. If blade names are not included in the curve file, then names will be generated in the form `Blade n`, where  $n$  is an integer.

---

**Note:**

If a curve file that specifies multiple blades (defining a blade set) is used to define a particular blade object (stored in an object under the `Blade Set` object in the **Mesh** workspace), only the first blade in the file is used.

---

The following topics are discussed:

[10.3.2.1. Objects in the Geometry Workspace \(Profile Points Mode\)](#)

[10.3.2.2. Geometry Browser Settings \(Profile Points Mode\)](#)

### 10.3.2.1. Objects in the Geometry Workspace (Profile Points Mode)

The objects in the **Geometry** workspace are not used when loading geometry from profile points. They are used only to verify that the geometry is imported correctly from a CAD source. For details, see [Objects in the Geometry Workspace \(CAD Mode\) \(p. 106\)](#).

### 10.3.2.2. Geometry Browser Settings (Profile Points Mode)

To access the geometry browser, switch to the **Geometry** workspace and then, if necessary, double-click a high-level object (either the `Topological Entity Instances` object or the `CAD Families` object) in the object selector.


The geometry browser settings for `Profile Points` mode are described next:

### 10.3.2.2.1. Point Data Definition


#### 10.3.2.2.2. Geometry Setup (Profile Points Mode)

##### 10.3.2.2.1. Point Data Definition

###### 10.3.2.2.1.1. TurboGrid Curve Files

To specify a curve file, set a filename using a path relative to the working directory. You can click the corresponding *Browse*  icon to select a curve file using a browser.

###### 10.3.2.2.1.2. Coordinates and Units

The **Coordinates**, **Angle Units**, and **Length Units** settings each apply to the hub, shroud, and blade files. Click  next to a setting to select the appropriate option. For a given setting, if there is no single option that applies to all three geometry files, select *Curve Dependent* and use the object selector in the **Mesh** workspace to define that setting individually for each geometry file.

##### 10.3.2.2.2. Geometry Setup (Profile Points Mode)

###### 10.3.2.2.2.1. Rotation (Profile Points Mode)

These settings are similar to the **Rotation** settings of the Machine Data object. For details, see [Rotation \(p. 119\)](#).

###### 10.3.2.2.2.2. # of Bladesets (Profile Points Mode)

A rotating machine component is made up of adjacent blades that are equally spaced around the circumference of the machine. The Theta extent of one blade set is calculated as 360 degrees divided by the number of main blades. Many rotating machine components have secondary and tertiary blades that are placed between the main blades. These are often called splitter blades. A blade set contains one main blade and optional splitter blades that repeat cyclically around the axis of the rotating machine component.

For rotating machine components without any splitter blades, the number of blade sets equals the total number of blades.

TurboGrid creates a mesh for one blade set only. The mesh can be copied and rotated using an Ansys CFX Pre-processor, if necessary, before it is solved in an Ansys CFX Solver.

###### 10.3.2.2.2.3. Leading/Trailing Edge Definition on the Blade (Profile Points Mode Only)

Optionally select *Cut-off* or *square*. For details, see [Cut-off or square \(p. 130\)](#).

### 10.3.3. Defining Geometry from Profile Points and a CAD Input File

In stand-alone mode, you can use a combination of geometry sources:

- Profile point data for the blades and main passage
- A CAD input file, such as a Parasolid file, for secondary flow passages.

To do this:

1. Optionally open a TurboGrid Initialization file (**File > Load TurboGrid Init File**).

This step can save time by configuring file names in the settings of the geometry browser.

2. In the **Geometry** workspace, open the geometry browser and set **Input Mode** to `Profile Points With Secondary CAD`.

3. Configure the settings in the geometry browser.

For details, see [Geometry Browser Settings \(Profile Points With Secondary CAD Mode\)](#) (p. 112).

4. Click **Apply**.

The following topics are discussed:

[10.3.3.1. Geometry Browser Settings \(Profile Points With Secondary CAD Mode\)](#)

### 10.3.3.1. Geometry Browser Settings (Profile Points With Secondary CAD Mode)

These settings are a superset of those for the `CAD` and `Profile Points` input modes, which are detailed in the following sections:

- [Geometry Browser Settings \(CAD Mode\)](#) (p. 107)
- [Geometry Browser Settings \(Profile Points Mode\)](#) (p. 110)

### 10.3.4. Defining Geometry from Generated CAD: CAD From NDF

If your geometry is defined by an NDF file, you can use the **Import NDF File** command to have TurboGrid automatically generate a CAD definition from the NDF file and then load a particular blade row from that CAD definition. The CAD definition consists of a CAD file and associated TurboGrid Initialization file. The flow path and any secondary flow paths defined in the NDF file are automatically included in the generated CAD and the resulting loaded geometry. In subsequent sessions of TurboGrid, you can load any other blade row from the same generated CAD without having to regenerate the latter. For details, see [Import NDF File Command](#) (p. 32).

After importing the NDF file, the geometry browser has **Input Mode** set to `CAD`. For details, see [Loading CAD Data From a CAD Input File](#) (p. 106).

### 10.3.5. Defining Geometry from Generated CAD: CAD From Profile Points

If your geometry is defined by profile point data, you can use the `CAD From Profile Points` input mode to have TurboGrid automatically generate and import a CAD model created from the profile points. The CAD model is then used as the geometry for all TurboGrid meshing operations. Note that the profile point data can be re-read at any time to update the CAD model.

When this input mode is selected, you can also define additional geometry features to include in the CAD model, such as blade blends (fillets) or partial tips. For details, see [Blade CAD Features \(p. 132\)](#).

---

**Note:**

- When you use the `CAD From Profile Points` input mode, a BladeModeler or BladeBuilder license is required, depending on the selected **CAD Generation Tool** (geometry browser, **CAD Properties** tab).
- The `CAD From Profile Points` input mode is available only in stand-alone mode; it is not supported in Ansys Workbench.
- If you plan to include additional geometry features, you might want to also generate the geometry without those features in order to evaluate the effect on the geometry of generating CAD. Specifically, you might want to compare the geometry obtained from profile points alone (via **Input Mode** set to `Profile Points`) with the geometry obtained by generating CAD from the same profile points (via **Input Mode** set to `CAD From Profile Points`).
- If used with Release 2023 R2 or newer of TurboGrid, state and session files recorded from Releases 2022 R2 or 2023 R1 might include tip clearance in the CAD file, rather than apply tip clearance in TurboGrid after loading the CAD file. If you want to retain the previous behavior, for example because applications other than TurboGrid use, and expect no tip clearance in the CAD file, you can modify such state or session files to effectively turn off **Include Tip Clearance in CAD File**. As a guide, examine a state or session file that was recorded in Release 2023 R2 or newer, with **Include Tip Clearance in CAD File** being turned off in the geometry browser. The **Include Tip Clearance in CAD File** setting is described in [Geometry Browser Settings \(CAD From Profile Points Mode\) \(p. 114\)](#).

---

To generate CAD from profile point data:

1. Load the profile point data.

For details, see [Defining Geometry from Profile Points \(p. 108\)](#) bearing in mind the requirement to run in stand-alone mode.

2. In the **Geometry** workspace, open the geometry browser and set **Input Mode** to `CAD From Profile Points`.
3. Configure the settings in the geometry browser.

For details, see [Geometry Browser Settings \(CAD From Profile Points Mode\) \(p. 114\)](#).

4. Click **Apply And Generate CAD**.

---

**Note:**

- It takes a significant amount of time for the CAD file to be generated (or regenerated, if generated previously).

- The CAD geometry is only ever regenerated on demand, that is, it will not be updated automatically if the profile point data changes or if the blade blend settings are modified. If you make changes that will affect the CAD geometry, you must then manually regenerate the CAD geometry, either using the geometry browser or by using **File > Update Geometry**.

After loading a state file that refers to an existing CAD file created from the `CAD From Profile Points` input mode, TurboGrid assumes that the existing CAD geometry is up-to-date with any relevant settings in the state file, for example, the blade blend settings (see [Blade CAD Features \(p. 132\)](#)). Therefore, such objects will appear to be up-to-date in the object selector even if they are not. To avoid inadvertently using an out-of-date CAD geometry, you can make it a practice to manually regenerate the CAD geometry after loading the state file, if you are not sure whether it is up-to-date or not.

---


The following topics are discussed:

[10.3.5.1. Geometry Browser Settings \(CAD From Profile Points Mode\)](#)

[10.3.5.2. Limitations of CAD From Profile Points Input Mode](#)

### 10.3.5.1. Geometry Browser Settings (CAD From Profile Points Mode)

- On the **Input Selection** tab:

The **Generated CAD Definition > File Name** setting holds the name of a CAD file. Specify a file name (relative or full path optional) that has extension ".x\_b" or ".pmdb", for example "geometry.x\_b". You can click the *Browse*  icon to select an existing file (to be overwritten) using a browser. The appropriate extension depends on the **CAD Generation Tool** selection on the **CAD Properties** tab: ".x\_b" for BladeEditor; ".pmdb" for BladeBuilder.

The remaining settings on the **Input Selection** tab are the same as the corresponding settings available for the `Profile Points` input mode. For details, see [Geometry Browser Settings \(Profile Points Mode\) \(p. 110\)](#).

- On the **CAD Properties** tab:

The **CAD Scaling** setting can be set to `Actual Size` or `Small Model Scaling`. You should consider selecting the `Small Model Scaling` option if the length scale of the blade (or smallest blade of a blade set) is less than a few centimeters, especially if the CAD geometry fails to be generated with the `Actual Size` option or if the leading/trailing edge of the blade is not properly resolved. As a consequence of using the `Small Model Scaling` option, the geometry is scaled in the resulting CAD file. Although TurboGrid takes account of this scaling, loading the generated CAD geometry into other products results in scaled geometry.

---

#### Note:

The `Small Model Scaling` option is not supported when **CAD Generation Tool > Tool** is set to `BladeBuilder`.

---

The **Blend Style** setting has the following options:

- Approximate Blend (Low Fidelity Geometry Only)

Use this option to save time if you do not intend to produce a mesh using high-fidelity geometry for the true blend.

With this option:

- The generated CAD geometry does not include high-fidelity geometry for the true blend.
- The generated CAD geometry includes low-fidelity geometry for an approximate blend. This low-fidelity representation of the blade blend facilitates structured meshing where the blend reaches the hub/shroud.

- True Blend (Low and High Fidelity Geometry)

With this option:

- The generated CAD geometry includes high-fidelity geometry for the true blend.
- The generated CAD geometry includes low-fidelity geometry for an approximate blend. This low-fidelity representation of the blade blend facilitates structured meshing where the blend reaches the hub/shroud.

In the context of TurboGrid, a blade blend represented with high-fidelity geometry is considered to be a type of *complex blade end*. Use this option if you intend to produce a mesh for high-fidelity geometry. Note that high-fidelity geometry requires meshing with ATM3D topology. For related information, see [Meshing High-Fidelity Geometry \(p. 219\)](#).

**Figure 10.6: Approximate Blend versus True Blend (p. 133)** shows that an approximate blend meets the hub/shroud at a non-zero angle whereas a true blend meets asymptotically.

**Include Tip Clearance in CAD File**, when selected, causes any specified tip clearance from the `Hub Tip` and/or `Shroud Tip` objects to be included in the resulting CAD geometry. **Include Tip Clearance in CAD File** is selected by default for new cases.

Details on the **Include Tip Clearance in CAD File** setting:

- To provide support for all tip clearance options, ensure that **Include Tip Clearance in CAD File** is selected. In particular, the `Profile Number` tip clearance option requires that **Include Tip Clearance in CAD File** is selected; for details, see [Clearance Type \(p. 141\)](#).
- If you want to use the CAD geometry in other software with the tip clearance included, ensure that **Include Tip Clearance in CAD File** is selected.
- If **Include Tip Clearance in CAD File** is not selected, then the CAD geometry does not include tip clearance from the `Hub Tip` and `Shroud Tip` objects, and, for eligible tip clearance types only, TurboGrid applies such tip clearance after loading the CAD geometry.

Note that the mesh can differ slightly depending on whether **Include Tip Clearance in CAD File** is selected. You might want to clear the **Include Tip Clearance in CAD File** check box to, for example, revert to the behavior prior to Release 2023 R2.

- Most state and session files recorded from Releases 2022 R2 or 2023 R1 will continue to operate as before; that is, the tip clearance will not be included in the CAD file and will instead be applied by TurboGrid after loading the CAD file. However, some session files from Releases 2022 R2 and 2023 R1 (those recorded without applying settings from the geometry browser after setting `CAD From Profile Points` input mode) will effectively have **Include Tip Clearance in CAD File** selected, and so will add the specified tip clearance to the CAD file.

If you want to retain the previous behavior, for example because applications other than TurboGrid use, and expect no tip clearance in the CAD file, you can modify such state or session files to effectively turn off **Include Tip Clearance in CAD File**. As a guide, examine a state or session file that was recorded in Release 2023 R2 or newer, with **Include Tip Clearance in CAD File** being turned off in the geometry browser.

The **Profile Points For CAD Generation > Method** setting is used to determine whether the CAD geometry is generated directly from the original user-supplied profile points (in the specified curve files) or whether the CAD geometry is generated based on the surfaces already created by TurboGrid from the profile points.

If the original profile points are to be re-used, then each profile must have the same number of points. Tagging the leading and trailing edge points in the file is highly recommended to help ensure that the CAD geometry can be created successfully.

The **Profile Points For CAD Generation > Method** setting can be set to `Extract Profile Points From Geometry` or `Use Original Profile Points`. If you select `Use Original Profile Points`, then the original profile points will be re-used (preserved) if possible. If the original profile points cannot be re-used, or if you select `Extract Profile Points From Geometry`, then new profile points are generated based on the geometry; the number and distribution of new profile points are generally different than the number and distribution of the original profile points. After you click **Apply And Generate CAD** with `Use Original Profile Points` selected, the object editor indicates whether original points were successfully preserved. If you select `Extract Profile Points From Geometry`, there is no need to tag the leading and trailing edge points in the data file, and no need to ensure the same number of points per profile.

The **CAD Generation Tool > Tool** setting specifies which tool is used to generate CAD geometry for TurboGrid. The default selection is controlled by preference **Default CAD Generation Tool** (see [TurboGrid Options \(p. 48\)](#)).

The options are:

- `BladeEditor` `BladeEditor` generates CAD geometry and saves it to the file with extension `.x_b` mentioned on the **Input Selection** tab. If a feature such as a blend or partial tip exists, all aspects of the geometry, except for tip gaps and plain ends, are saved as high-fidelity geometry; otherwise, only low-fidelity geometry is saved.
- `BladeBuilder` `BladeBuilder` generates CAD geometry and saves it to the file with extension `.pmdb` mentioned on the **Input Selection** tab. The **Included High Fidelity Features** setting controls such CAD generation in `BladeBuilder` according to the selected **Option**:
  - `Automatic` All aspects of the geometry, except for tip gaps and plain ends, are saved as high-fidelity geometry.
  - `All` All aspects of the geometry are saved as high-fidelity geometry.

→ `None` Only low-fidelity geometry is saved.

The following limitations apply to the use of BladeBuilder as a CAD generation tool for TurboGrid:

→ BladeBuilder is available only on Windows.

→ Partial tips are not supported.

→ The **CAD Scaling** setting option `Small Model Scaling` is not supported.

The following requirements apply to the use of a CAD generation tool for TurboGrid:

– Leading and trailing edges must be round.

– Blends must use a constant radius.

If the selected tool cannot generate the geometry because of known limitations, you will be given notice upon attempting to generate the geometry and CAD generation will fail.

#### • **Apply And Generate CAD**

Clicking the **Apply And Generate CAD** button applies the geometry browser settings and then generates (or regenerates) the CAD geometry.

Generation (or regeneration) of the CAD geometry involves running BladeEditor or BladeBuilder temporarily in the background to generate the CAD file and associated TurboGrid Initialization file, then loading those two files into TurboGrid, resulting in new/updated CAD objects and (in accordance with the contents of the TurboGrid Initialization file) new/updated topological entities in the **Geometry** workspace tree.

### 10.3.5.2. Limitations of CAD From Profile Points Input Mode

- The `CAD From Profile Points` input mode is available only in stand-alone mode; it is not supported in Ansys Workbench.
- Blade blends and partial tips cannot be enabled together.
- Blade blends and partial tips are not available if any Hub Tip or Shroud Tip has been enabled.
- Blade stagger cannot be enabled if blade blends are enabled.
- Secondary flow paths are not supported.
- There is no support for blades with **Cut-off or square** leading or trailing edges.
- There is no support for splitter blades.
- The `CAD From Profile Points` input mode is supported only if the rotation axis is Z.
- All curves are generated using the `Bspline` option rather than the `Piece-wise linear` option, regardless of the settings. For example, the Hub object > **Data Hub** tab > **Curve Type** setting is ignored.

### 10.3.6. The Geometry Objects in the Mesh Workspace

The **Mesh** workspace can be used to set up geometry from a source of profile points. It can also control the visibility of the geometric objects that TurboGrid generates, regardless of the source of geometric data (profile points or CAD data).

The `Geometry` objects in the **Mesh** workspace are outlined below with a summary of the function performed by each.

The Machine Data Object (p. 118)	This object defines the geometry and rotation properties of the rotating machine.
The Hub and Shroud Objects (p. 120)	These objects define the hub of the rotating machine.
The Blade Set Object and Blade Objects (p. 124)	These objects define the blade(s) of the rotating machine.
The Hub Tip and Shroud Tip Objects (p. 141)	These objects define the hub tip and shroud tip of the blade(s).
The Low Periodic and High Periodic Objects (p. 142)	These objects control the methods for creating the low and high periodic surfaces.
The Inlet and Outlet Objects (p. 143)	These objects define the locations and profiles of the inlet and outlet domains of the mesh.
The Outline Object (p. 149)	This object controls the display of the outline of the geometry.

#### 10.3.6.1. The Machine Data Object

The `Machine Data` object contains geometric data that applies to the entire turbo machine (for example, the location of the rotation axis). Defining the machine data is an essential step in creating a mesh in Ansys TurboGrid, and can be accomplished by one of the following methods:

- Editing the `Machine Data` object.
- In Ansys Workbench, attaching a `Geometry` or `Blade Design` cell upstream of the `Turbo Mesh` cell.

Note that, in this case:


- The **Pitch Angle** setting is taken from the upstream cell, and cannot be changed in Ansys TurboGrid.
- The **Rotation** settings are taken from the upstream cell by default, but can be overridden by selecting the **Rotation** check box.
- The **Base Units** setting can be changed and is not affected by changes to the upstream cell.

- Loading a BladeGen `.inf` file by selecting **File > Load TurboGrid Init File** or clicking *Load TurboGrid Init File* .

You can define machine data, hub, shroud, and blade objects by loading a BladeGen `.inf` file. For details, see [Load TurboGrid Init File Command \(p. 31\)](#).

- Loading curve files by selecting **File > Load Profile Points Or CAD** or clicking *Load Profile Points Or CAD* .

You can define the machine data, hub, shroud, and blade objects by loading curve files. For details, see [Load Profile Points Or CAD Command \(p. 32\)](#).

To edit the machine data, open the `Machine Data` object from the object selector or click *Edit Machine Data* .

### 10.3.6.1.1. Data Tab

#### 10.3.6.1.1.1. Pitch Angle

A rotating machine component is made up of blade sets that are identical, adjacent to each other, and equally spaced around the entire circumference. In the context of Ansys TurboGrid, the *pitch angle* is the Theta extent of one blade set. To calculate the pitch angle based on the number of blade sets per 360°, set **Method** to `Bladeset Count`; the pitch angle is then calculated as 360° divided by the number of blade sets. If you need to specify the pitch angle directly (as in the Deformed Turbine tutorial), set **Method** to `Specified Angle`.


Ansys TurboGrid creates a mesh for one blade set only. The mesh can be copied and rotated using an Ansys CFX Pre-processor, if necessary, before it is solved in an Ansys CFX Solver.

#### 10.3.6.1.1.2. Rotation

The **Method** for defining the rotation axis can be set to one of the following:

- **Principal Axis** - select the X, Y, or Z axis. Select **Right Handed** or **Left Handed** to indicate the direction of Theta.

If you are running TurboGrid in Workbench and need to change the direction of Theta (for example, if the available topologies do not match the geometry in terms of blade ordering within a blade set), select **Override Theta Direction for Topology** and then select the appropriate setting: **Right Handed** or **Left Handed**.

- **Rotation Axis** - define a custom axis by entering the Cartesian coordinates of two points on the axis. The coordinates of the points can be typed in or selected using the  icon in the object editor.

#### 10.3.6.1.1.3. Units

Base units are applicable only when using profile point data as a geometry source.

It is necessary to specify base units because the geometry kernel has an optimal range for storing numbers between 1 and 500. If, for example, the geometry data is given in mm, but

the machine is on the order of 10 m in size, the numbers stored in mm would be outside of the optimal range. In this case, the base units should be cm (default), to reflect the scale of the machine.

The base units specified in the `Machine Data` object are used only to adjust the range of numbers to suit the geometry kernel; they are not used to interpret geometric data contained in files (for example, the hub curve file), nor are they used to specify the units of an exported mesh.

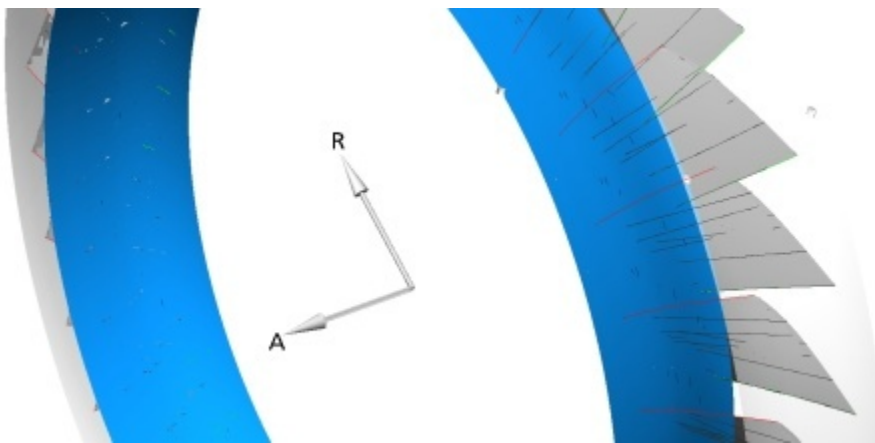
#### 10.3.6.1.1.4. Machine Type

Set **Machine Type** to an appropriate value or leave it set to `Other`. The value will be used by TurboGrid to help choose appropriate topology templates.

#### 10.3.6.1.2. Rotation Axis Visibility


By turning on the visibility of the `Machine Data` object, the rotation axis becomes visible in the viewer, along with a sample radial direction vector. An example is shown in [Figure 10.3: Rotation Axis](#) (p. 120).


**Figure 10.3: Rotation Axis**



The visibility of the `Machine Data` object is off by default. For details on setting visibility, see [Visibility Check Box](#) (p. 22).

#### 10.3.6.2. The Hub and Shroud Objects

The hub is the surface of the machine closest to the axis of rotation. It defines the inner fluid flow surface. To edit or define the machine component's hub, edit the `Hub` object from the selector or click *Edit Hub* .

The shroud is the surface of the machine farthest from the axis of rotation. It defines the outer fluid flow surface. To edit or define the machine component's shroud, edit the `Shroud` object from the selector or click *Edit Shroud* .

The hub and shroud can be defined only after the machine data has been defined, unless all of these objects are defined together using the `Profile Points` settings in the geometry browser (in the **Geometry** workspace). For details, see [Defining Geometry from Profile Points](#) (p. 108).

## 10.3.6.2.1. Data Hub and Data Shroud Tabs

### 10.3.6.2.1.1. Coordinate System and Hub/Shroud File Definition

Set **Coordinates**, **Angle Units**, and **Length Units** according to the data in the hub/shroud file. The hub/shroud file must contain data points that define the hub/shroud curve. To specify a hub/shroud file, set **File Name** using a path relative to the working directory. You can click the

*Browse*  icon to select a curve file using a browser.

The hub/shroud curve runs upstream to downstream and must extend upstream of the blade leading edge and downstream of the blade trailing edge. The points must be listed in the file line by line, in free-format ASCII style, in order from upstream to downstream.

For example:

```
-1.0  0.0  1.0
 2.0  0.0  1.0
```

No names or labels may be present in the file.

---

#### Note:

If the hub/shroud file was used in earlier versions of CFX-TurboGrid, you can use the file again in this release without any modifications. However, any data in the fourth field formerly used for parametric values is now ignored.

---

### 10.3.6.2.1.2. Geometric Representation

#### 10.3.6.2.1.2.1. Curve Type

The curve used to loft the hub/shroud surface may be constructed in one of two ways, as determined by the **Curve Type** setting. The available **Curve Type** options are:

- Piece-wise linear

The *Piece-wise linear* option causes the points listed for the hub/shroud curve to be connected to one another with straight lines.

- B spline (default)

The *B spline* option results in the interpolation of a smooth curve for the hub/shroud using points listed in the curve file. This option may be necessary if the hub/shroud curve is defined with a small number of points. This option is discouraged when you have a large number of points in the curve file, due to undesirable effects on the shape of the spline, and due to limitations of the software.

#### 10.3.6.2.1.3. Curve or Surface Visibility

By default, the hub/shroud surface is visible while the hub/shroud curve is not visible. To see the curve, the following steps could be taken in the **Data Hub/Data Shroud** tab:

1. Select the **Curve or Surface Visibility > Show Curve** check box.
2. Clear the **Curve or Surface Visibility > Show Surface** check box.
3. Click **Apply**.
4. Right-click the `Hub/Shroud` object and select **Render (Properties) > Edit Options**.
5. On the **Render** tab, select the **Draw Lines** check box.
6. Click **OK**.

#### 10.3.6.2.1.4. Draw Trimmed Portion Of Curve

Select **Draw Trimmed Portion Of Curve** to show the entire curve including the portion(s) beyond the inlet/outlet.

#### 10.3.6.2.1.5. Reread Button

If the file that defines the hub/shroud has been modified since it was last loaded, clicking the **Reread** button will cause the hub/shroud data to be reloaded. The `Blade Set` object and individual blade objects also have this feature.

### 10.3.6.2.2. Transform Tab

This tab contains specifications for rotational and translational transforms that adjust the geometry. The settings on this tab adjust the hub/shroud geometry only. The `Blade Set` object and individual blade objects can also have their coordinates transformed using the corresponding tabs of those objects.

#### 10.3.6.2.2.1. General Rotation

Specify an axis via two points and a rotation angle about the axis.

#### 10.3.6.2.2.2. Translation

Specify the direction and distance to translate the geometry. The direction may be any one of: `Machine Axis`, `X Axis`, `Y Axis`, `Z Axis`, or `Vector` (any specified direction).

#### 10.3.6.2.2.3. Axial Rotation

Specify the angle to revolve the geometry about the machine axis.

### 10.3.6.2.3. Hub Regions and Shroud Regions Tab

You can create regions on the hub and/or shroud within the inlet and/or outlet domains. Each region is defined either by two points on the hub curve or two points on the shroud curve.

Hub/shroud regions can be used to precisely position (in the streamwise direction) mesh elements at the hub/shroud so that mesh nodes are placed at key locations, facilitating the subsequent addition of geometric features at those locations (using other software). The hub/shroud regions affect the distribution of mesh elements along the inlet/outlet domain (in the streamwise direction), generally causing elements to be distributed finely (in the streamwise direction) near the region

borders. This fine distribution could potentially help to resolve flow details near subsequently added geometric features.

To add a new region, click **New**  or right-click in the list and select **New**. (To delete an existing region, select the region in the list and click **Delete**  or right-click a region in the list and select **Delete**.)


Each region is specified according to pairs of region boundary points on the hub or shroud curve. Although the points within a pair are named "Start" and "End", there is no directionality; the points can be swapped without effect. When specifying the boundary points for a region, choose the **Method** most appropriate for the geometry:

- Set A (Estimate R)

Choose this method if the hub or shroud curve extends mainly in the axial direction (or at least not very closely in the radial direction).

- Set R (Estimate A)

Choose this method if the hub or shroud curve extends mainly in the radial direction (or at least not very closely in the axial direction).

Enter point coordinates using the keyboard or, after clicking on a coordinate field, click (with **Select**  active) a point on the hub/shroud curve in the **3D Viewer**.

When you click **Apply**, the points are defined. Each point is represented visually in the **3D Viewer** by an octahedron on the low-Theta edge of the hub/shroud.

When a point is visible in the **3D Viewer**, you can adjust its location with the mouse by holding **Ctrl+Shift** while dragging.

The **Hub Region Proximity Tolerance/Shroud Region Proximity Tolerance** setting controls the minimum element size in the streamwise direction. This setting can normally be left at its default value, which is controlled by a preference. For details, see [TurboGrid Options \(p. 48\)](#).

After you click **Apply**, the hub/shroud region is defined.

When hub/shroud regions are defined on the inlet and/or outlet domain, the **Mesh Data** object has additional settings on its **Inlet** and/or **Outlet** tabs, respectively. Those **Mesh Data** settings control:

- Whether the defined hub and shroud regions are enabled. By default, they are enabled, in which case they are used to divide their respective inlet/outlet domains into segments.
- The mesh element count and distribution (in the streamwise direction) within each segment of the inlet/outlet domain.

For details, see [Inlet and Outlet Tabs \(p. 178\)](#).

As just mentioned, when the defined hub and shroud regions are enabled, segments are formed within the inlet/outlet domain. Some segments directly correspond to named hub/shroud regions. Other segments are formed automatically in order to cover any remaining portions of the

hub/shroud within the inlet/outlet domain. The resulting 3D mesh includes 2D regions that correspond with the segments.

Figure 10.16: Inlet Domain Segments (p. 181) shows an example of a 3D Mesh object with 2D regions that correspond with the segments inside the inlet domain.

The source of a segment determines how the corresponding 2D region is named:


- For a segment that originates from a defined hub/shroud region, the name of the hub/shroud region (as specified in the **Hub Regions** or **Shroud Regions** tab) is used for the corresponding 2D region.
- For any other segment, the corresponding 2D region is automatically given a name like "HUB1", "HUB2", "SHROUD1", "SHROUD2", with numbers increasing for segments further from the passage domain.

---

**Note:**

- A region cannot have one end in the inlet domain and the other end in the outlet domain. Regions on the hub must not overlap with other regions on the hub. Regions on the shroud must not overlap with other regions on the shroud.
  - A region defined on the hub affects the mesh distribution through the inlet/outlet domain to the shroud and vice versa. A topological line segment is created between each specified region boundary point and its corresponding point on the opposite side (hub or shroud). The corresponding point is created by projecting across the domain, unless such a projected point would be too close (according to the **Hub Region Proximity Tolerance/Shroud Region Proximity Tolerance** specification) to an existing region boundary point, in which case the latter serves as the corresponding point.
  - If you have trimmed the inlet/outlet domain (see [Trim Inlet Tab](#) or [Trim Outlet Tab - Changing the Inlet or Outlet Boundary \(p. 148\)](#)), a region that crosses the trim location is adjusted so that the defining point that lies within the trimmed region is effectively applied at the trim location.
- 

### 10.3.6.3. The Blade Set Object and Blade Objects

To edit or define the machine component's blades as a set, edit the `Blade Set` object from the object selector or click *Edit Blade Set* . To define or modify a single blade, edit the corresponding object stored under the `Blade Set` object. You can only define the blade set or individual blades after the hub and shroud have been defined, although all of these can be defined in one step. For details, see [Steps to Create a Mesh \(p. 101\)](#).

The count and naming of the individual geometry (and topology, and mesh data) objects for each blade in the blade set are managed automatically by Ansys TurboGrid. If a profile file containing multiple blades is specified for the `Blade Set` object (whether directly or indirectly, for example, by loading a `BladeGen .inf` file), then a blade object will be created for each set of data in the file and named according to the name specified in the file (unless a name is not specified, in which case a generic name of the form "Blade *n*" will be given, where *n* is an integer starting at 1).

The individual blade objects contain a profile curve file specification and a setting to indicate which single blade in the file applies; numbering starts at zero.

### 10.3.6.3.1. Blade Tab

#### 10.3.6.3.1.1. Apply Settings To All Blades Check Box

When **Apply Settings To All Blades** is selected, `Blade Set` settings overwrite the corresponding blade object settings. If **Apply Settings To All Blades** is cleared, the settings that apply to all blades are disabled.

If you change a setting for a blade object, and the `Blade Set` object has a corresponding setting, then the **Apply Settings To All Blades** check box will be cleared to help prevent accidentally losing the blade-specific settings in case you reapply the `Blade Set` object.

#### 10.3.6.3.1.2. Coordinate System and Blade File Definition

##### 10.3.6.3.1.2.1. File Name

Set **File Name** to the name of the blade set file. To open a file selector, click *Browse* .

The blade file must contain data points that define the blade profile (or rib) curves and should have the file extension `.curve` or `.crv`.

The profile points must be listed line by line, in free-format ASCII style in a closed-loop surrounding the blade.

A minimum of two profiles are required in the blade file: one which lies close to the hub surface and one which lies close to the shroud surface. The profile is not required to lie exactly on the surface. If it lies between the hub and shroud surfaces, it must be within 8% of the span from the surface. If a profile lies outside of the passage, its distance from the surface has no maximum limit.

Profiles can be used to define tips. For details, see [The Hub Tip and Shroud Tip Objects \(p. 141\)](#). Profiles are allowed, and sometimes required, beyond a tip.

The profiles must be listed in the file in order from hub to shroud. Individual profile datasets are separated by a line beginning with the "#" character. Any text following the "#" character is used as the name associated with the subsequent profile.

For example:

```
# Hub Profile
0.0 0.0 1.0 le
1.0 0.0 1.0
1.0 1.0 1.0 te
0.0 1.0 1.0
0.0 0.0 1.0
# Intermediate Blade Profile
0.0 0.0 2.0 le
1.0 0.0 2.0
1.0 1.0 2.0 te
0.0 1.0 2.0
0.0 0.0 2.0
# Shroud Tip Profile
0.0 0.0 2.75 le
```

```

1.0 0.0 2.75
1.0 1.0 2.75 te
0.0 1.0 2.75
0.0 0.0 2.75
# Shroud Profile
0.0 0.0 3.0 le
1.0 0.0 3.0
1.0 1.0 3.0 te
0.0 1.0 3.0
0.0 0.0 3.0

```

There is no restriction on the number of profiles or the number of data points in a profile dataset. However, as noted above, a minimum of two profiles is required.

### 10.3.6.3.1.2.2. Coordinates, Angle Units, and Length Units

Set **Coordinates**, **Angle Units**, and **Length Units** according to the data in the blade file.

#### 10.3.6.3.1.3. Geometric Representation

Ansys TurboGrid generates blade surfaces using a two-step process:

1. Curves are generated.
2. A surface is created by lofting across the set of curves (that is, sweeping from one curve to the next).

The **Geometric Representation** settings control how these steps are performed.

#### 10.3.6.3.1.3.1. Method

The **Method** setting allows two preset methods, and one general method, for controlling the settings that govern the geometric representation of blade surfaces (that is, the **Lofting**, **Curve Type**, and **Surface Type** settings, described shortly).

The available **Method** options are:

- BladeModeler

The `BladeModeler` option sets **Lofting** to `Spanwise`, **Curve Type** to `Bspline`, and **Surface Type** to `Ruled`.

- Flank Milled

The `Flank Milled` option sets **Lofting** to `Streamwise`, **Curve Type** to `Piece-wise Linear`, **Surface Type** to `Ruled`.

- Specify

The `Specify` option makes the **Lofting**, **Curve Type**, and **Surface Type** settings available for direct specification.

The following rules are followed by Ansys TurboGrid for selecting the geometric representation method:

- If you load a blade from a BladeModeler .inf file, the BladeModeler option will be selected automatically.
- If the BladeModeler option is selected and there are only 2 blade profile curves (for the applicable blade), the selected option will change to Flank Milled (which is equivalent, in this case).
- If the Flank Milled option is selected and there are more than 2 blade profile curves (for the applicable blade), the selected option will change to BladeModeler and a warning message will be issued.

### 10.3.6.3.1.3.2. Lofting

The direction in which lofting occurs is set by the **Lofting** setting. The available **Lofting** options are:

- Streamwise

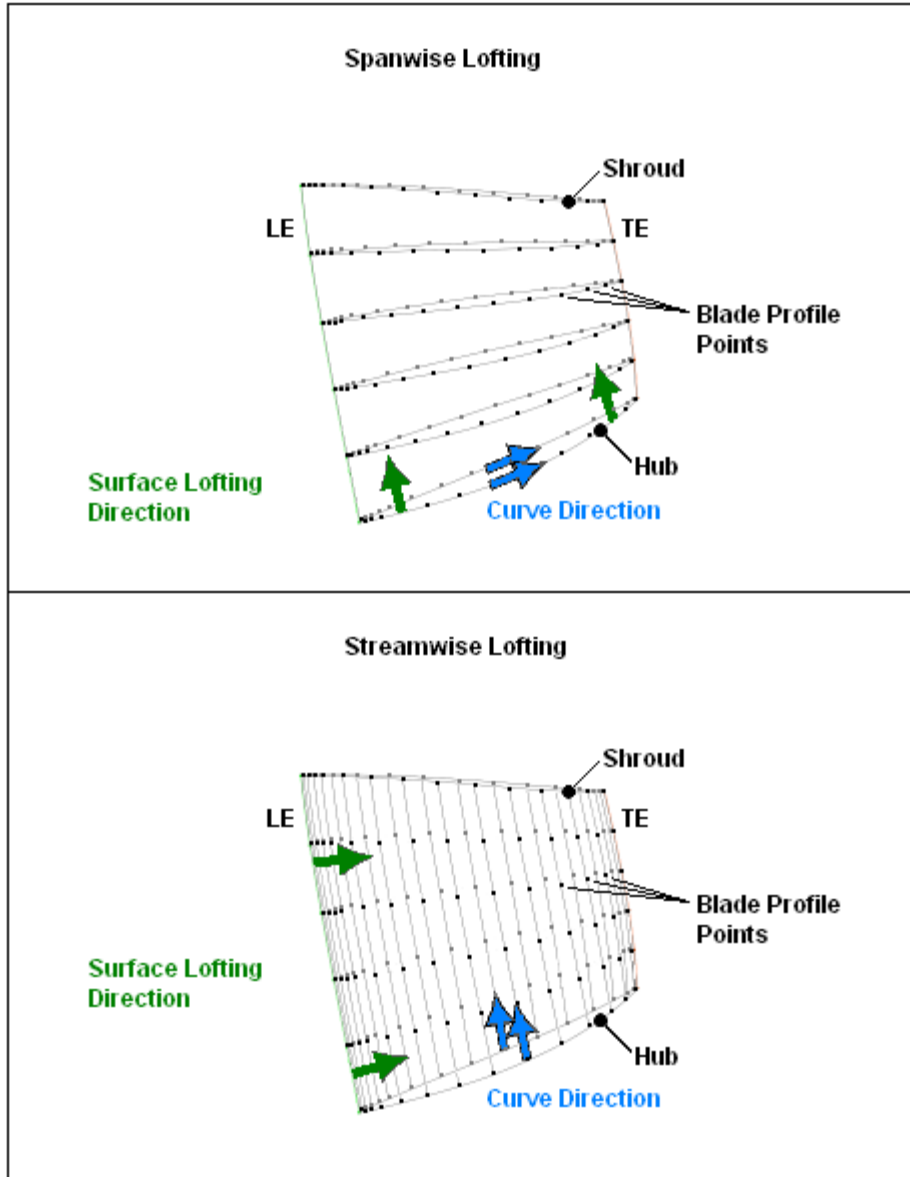
The curves that are swept in the process of streamwise lofting are formed by connecting corresponding points between adjacent blade profiles. For this method of lofting, you must ensure that:

- each blade profile has the same number of points
- the points are ordered in the same direction
- the first,  $n$ th, and last point of each blade profile are at similar locations around the blade

- Spanwise

The curves that are swept in the process of spanwise lofting are the blade profile curves.

Figure 10.4: Spanwise Lofting versus Streamwise Lofting (p. 128) shows how spanwise lofting and streamwise lofting differ.

**Figure 10.4: Spanwise Lofting versus Streamwise Lofting**

The appropriate lofting setting depends on how the geometry was produced. For example, BladeModeler can produce blades via streamwise or spanwise lofting. The appropriate setting is stored in the `BladeGen.inf` file. As another example, a flank-milled blade should be generated and interpreted using two blade profiles (one at (or near) the hub and one at (or near) the shroud) with streamwise lofting.

---

**Note:**

The method of "sweeping" through the curves in the lofting process is controllable via the **Surface Type** setting, which can be set to `Bspline` (default) or `Ruled`. For details, see [Surface Type](#) (p. 129). For streamwise lofting, the `Bspline` method may cause unexpected results. In this case, the workaround is to use the `Ruled` method.

---

### 10.3.6.3.1.3.3. Curve Type

The set of curves that are used to loft the blade surface may be constructed in one of two ways, as determined by the **Curve Type** setting. The available **Curve Type** options are:

- Piece-wise linear

The `Piece-wise linear` option causes the points listed for each profile in the blade file to be connected to one another with straight lines. You should use this option when you have a large number of points (1000 or more) in a profile.

- B-spline

The `B-spline` option results in the interpolation of a smooth curve for each profile using points listed in the blade file. This option may be necessary if the profile curves are defined with a small number of points. This option is discouraged when you have a large number of points (1000 or more) in a profile, due to undesirable effects on the shape of the spline, and due to limitations of the software. The `B-spline` option is the default when **Method** is set to `BladeModeler` or `Specify`.

---

**Note:**

- The `Piece-wise linear` option is always used when **Method** is set to `Flank Milled`.
- 

### 10.3.6.3.1.3.4. Surface Type

The manner in which the set of curves is lofted is controlled by the **Surface Type** setting. The available **Surface Type** options are:

- Ruled

A ruled surface is created by sweeping along linear paths that connect one curve to a corresponding place on the next curve. The `Ruled` method for surfaces is similar to the `Piece-wise linear` method for curves, but is done in 2 dimensions instead.

- B-spline

A B-spline surface is created by sweeping along curvilinear paths that connect one curve to a corresponding place on the next curve. The `B-spline` method may be necessary if the blade is defined with a small number of profile curves.

### 10.3.6.3.1.4. Leading and Trailing Edge Definitions

Ansys TurboGrid uses an algorithm to determine the location of the leading edge curve(s) and trailing edge curve(s) and in most cases gets the correct locations (double edges are possible in the case of cut-off or square edges). In some instances, the location of one or more of the points may need to be adjusted. See [Leading Edge \(p. 138\)](#) and [Trailing Edge \(p. 139\)](#) for information about adjusting the leading and trailing edges.

If you know which points on each profile should lay on the leading and trailing edge curves, you can optionally designate them in the blade file using a fourth field. This ensures the correct

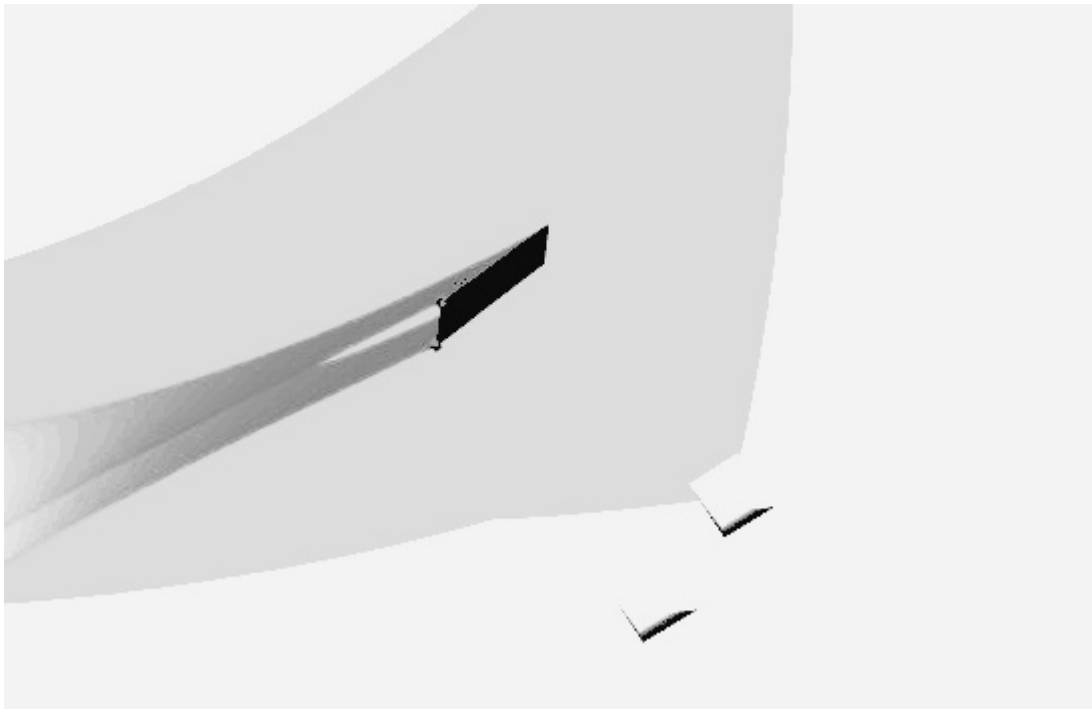
positioning of the leading and trailing edge curves in the beginning. The leading edge point on a profile should have **le1** (or **le2** for the second leading edge in the case of a double leading edge) in the fourth field and the trailing edge point on a profile should have **te1** (or **te2**) in the fourth field. These points are then guaranteed to be listed in the leading edge or trailing edge objects. For an example of the format of a blade file containing leading and trailing edge designations, try the following:

1. Load a blade file into Ansys TurboGrid.
2. Optionally change the type of leading or trailing edge between single and double (using the **LE Type** and **TE Type** settings, which are described below).
3. Save the blade file and examine it in a text editor, noting the **le1**, **te1** (and, for double edges **le2** and **te2**) entries in the fourth field.

#### 10.3.6.3.1.4.1. Cut-off or square

The **Cut-off or square** option is an option that, when *not* using ATM3D, may be used to create a flat leading/trailing edge between two edge curves. See [Figure 10.5: "Cut-off or square" Trailing Edge \(p. 130\)](#) for an example of a trailing edge that uses the **Cut-off or square** option; note that the trailing edge is defined by a pair of trailing-edge curves.

**Figure 10.5: "Cut-off or square" Trailing Edge**



The **Cut-off or square** option is intended for use on blade profiles that have discontinuities in slope at the edges. Use of this option on rounded or blunt leading/trailing edges may result

in edge curves that are not placed in optimal locations (that is, further adjustments may be necessary).

---

**Note:**

Using the **Cut-off or square** option prevents using the ATM3D meshing approach.

---

A "cut-off or square" edge having the same axial and radial coordinates at both cut-off points will have lines of rotation that coincide. In this case, TurboGrid considers the edge to be "cut-off". If the cut-off points do not have the same axial and radial coordinates, TurboGrid considers the edge to be "square", whether or not the edge appears, in a blade-to-blade view, to be orthogonal to the camberline.

### 10.3.6.3.1.5. Curve or Surface Visibility

The visibility of curves and surfaces for a blade object works in the same way as for the hub. For details, see [Curve or Surface Visibility](#) (p. 121).

### 10.3.6.3.1.6. Reread Button

If the file that defines the blade profile has been modified since it was last loaded, clicking the **Reread** button will cause the blade profile data to be reloaded. The hub and shroud objects also have this feature.

### 10.3.6.3.1.7. Save & Load Button

If a leading or trailing edge point has been moved, the **Save & Load** button appears in place of the **Reread** button. Selecting this button causes the modified blade profile data to be saved to a profile file (you will be prompted for the name), then reloaded (from the same file) in order for the new geometry to take effect. The leading edge and trailing edge objects also have this feature.

## 10.3.6.3.2. Transform Tab

The **Transform** tab contains specifications for rotational and translational transforms that adjust the geometry. The settings on this tab adjust the blade geometry only. The hub and shroud can also have their coordinates transformed using the corresponding tab of those objects.

### 10.3.6.3.2.1. General Rotation

Specify an axis via two points and a rotation angle about the axis.

### 10.3.6.3.2.2. Translation

Specify the direction and distance to translate the geometry. The direction may be any one of: `Machine Axis`, `X Axis`, `Y Axis`, `Z Axis`, or `Vector` (any specified direction).

### 10.3.6.3.2.3. Axial Rotation

Specify the angle to revolve the geometry about the machine axis.

### 10.3.6.3.3. Blade CAD Features

Blade CAD features include:

- Hub and/or shroud blends (fillets) made by the rolling ball process with a constant radius,
- Partial tips,
- Blade stagger.

---

**Note:**

Blade CAD features require:

- Use of the `CAD From Profile Points` input mode.

For details, see [Defining Geometry from Generated CAD: CAD From Profile Points](#) (p. 112) and [Limitations of CAD From Profile Points Input Mode](#) (p. 117).

- A BladeModeler or BladeBuilder license, depending on the selected **CAD Generation Tool** (geometry browser, **CAD Properties** tab).

---

To add a blade CAD feature, right-click the blade object and select **Insert > Blade CAD Feature > [feature object name]**. The **Blade CAD Features** object editor opens with the applicable tab displayed. You can set up other blade CAD features by switching to different tabs in the object editor.

---

**Note:**

Some blade CAD features are mutually exclusive; for example, a blend and a partial tip are mutually exclusive at the same end of a blade.

---

The **Blade CAD Features** object editor settings are described in the following sections:

[10.3.6.3.3.1. Blade Blends Tab](#)

[10.3.6.3.3.2. Hub Partial Tip and Shroud Partial Tip tabs](#)

[10.3.6.3.3.3. Blade Stagger tab](#)

[10.3.6.3.3.4. Common Settings](#)

#### 10.3.6.3.3.1. Blade Blends Tab

The **Blade Blends** tab can specify hub and/or shroud blends (fillets) made by the rolling ball process with a constant radius, provided that there is no partial tip specified on either the **Hub Partial Tip** tab or the **Shroud Partial Tip** tab.

After applying the settings of the **Blade Blends** tab, the `Blade Blend` object appears, if applicable, under the blade object. The `Blade Blend` object, when edited, opens the **Blade Blends** tab of the **Blade CAD Features** object editor.

The settings of the **Blade Blends** tab are discussed in the following sections:

## 10.3.6.3.3.1.1. Hub Blend and Shroud Blend

## 10.3.6.3.3.1.2. Blade CAD Rotation Axis

## 10.3.6.3.3.1.3. Generated CAD Definition

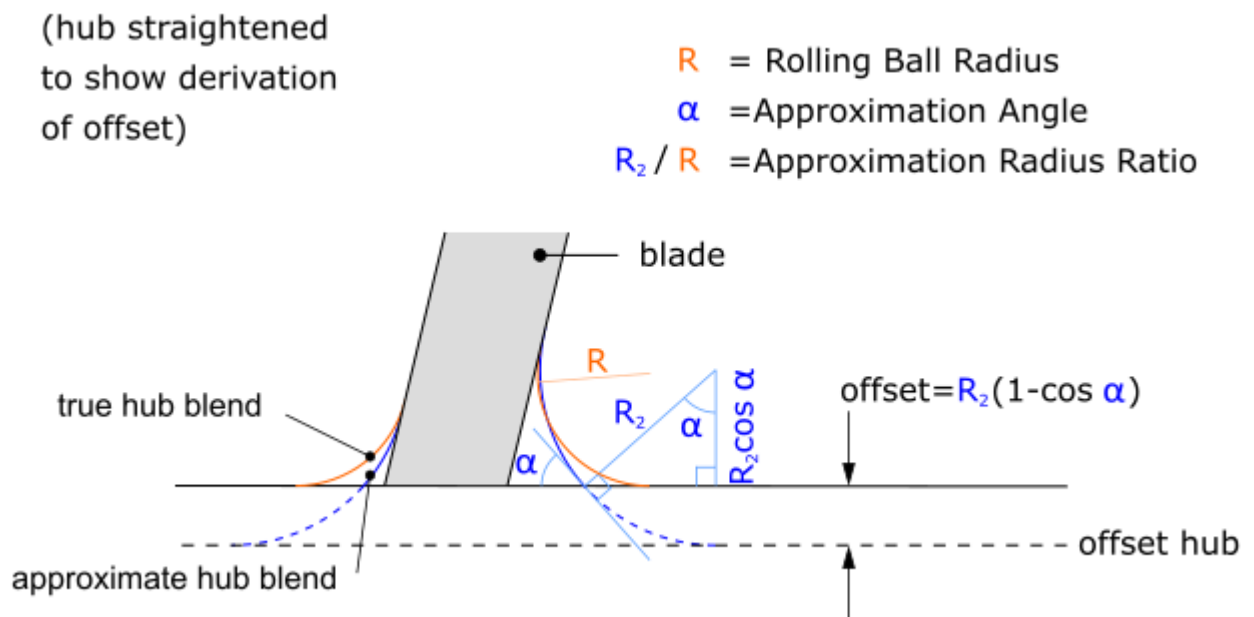
**10.3.6.3.3.1.1. Hub Blend and Shroud Blend**

Select **Hub Blend** and/or **Shroud Blend** as appropriate to specify which blends to generate using the rolling ball process. Specify the applicable **Rolling Ball Radius** values to be used in generating the blade blends.

A specified blade blend can optionally be rendered into a high-fidelity geometric representation, or *true blend*, for meshing with high-fidelity geometry. If you intend to mesh with high-fidelity geometry, see the description for the **Blend Style** setting in [Defining Geometry from Generated CAD: CAD From Profile Points in the TurboGrid User's Guide](#) (p. 112).

Figure 10.6: [Approximate Blend versus True Blend](#) (p. 133) shows how an approximate blend differs from a true blend; an approximate blend meets the hub/shroud at a small angle whereas a true blend meets the hub/shroud asymptotically. The small angle is required to support ATM topology; no such requirement exists for ATM3D topology.

**Figure 10.6: Approximate Blend versus True Blend**



A true blend is considered to be a type of *complex blade end*. As a result, when you specify a true blend:

- The **Complex Blade End** object appears in the object selector, in the **Geometry** branch, under the blade object. For details, see [Complex Blade End Object](#) (p. 139).

- Meshing high-fidelity geometry is required. Note that meshing high-fidelity geometry requires meshing with ATM3D or ATM v2 topology. For details, see [Meshing High-Fidelity Geometry \(p. 219\)](#).

---

**Note:**

If geometry generation fails to make a blend of constant radius, try changing the radius slightly.

---

### 10.3.6.3.3.1.2. Blade CAD Rotation Axis

For details, see [Blade CAD Rotation Axis \(p. 137\)](#).

### 10.3.6.3.3.1.3. Generated CAD Definition

For details, see [Generated CAD Definition \(p. 137\)](#).

### 10.3.6.3.3.2. Hub Partial Tip and Shroud Partial Tip tabs

The **Hub Partial Tip** tab and **Shroud Partial Tip** tab can specify partial tips at the hub and shroud, respectively, provided that there is no blade blend specified on the **Blade Blends** tab.

After applying the settings of the **Hub Partial Tip** tab and **Shroud Partial Tip** tab, the `Hub Tip Geometry` object and `Shroud Tip Geometry` object appear, as applicable, under the blade object. The `Hub Tip Geometry` object and `Shroud Tip Geometry` object, when edited, open the respective tab, **Hub Partial Tip** and **Shroud Partial Tip**, of the **Blade CAD Features** object editor. Also after applying the settings, the `Complex Blade End` object appears under the blade object. For details on the `Complex Blade End` object, see [Complex Blade End Object \(p. 139\)](#).

The settings of the **Hub Partial Tip** tab and **Shroud Partial Tip** tab are discussed in the following sections:

- [10.3.6.3.3.2.1. Hub Tip and Shroud Tip](#)
- [10.3.6.3.3.2.2. Blade CAD Rotation Axis](#)
- [10.3.6.3.3.2.3. Generated CAD Definition](#)

### 10.3.6.3.3.2.1. Hub Tip and Shroud Tip

Set **Tip Option** to one of the following:

- None

No partial tip is specified.

- `Blade Button`

A partial tip is specified using the following settings:

- **Button Body Radius** — specifies the radius of the circular cylinder used to form the button.

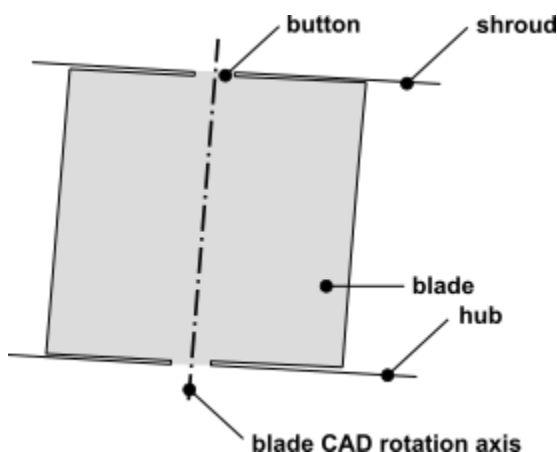
- **Tip Clearance Option** — specifies the tip gap that exists for the blade while the blade is in its data-defined position. Note that the data-defined position is considered to have a blade stagger angle of 0 degrees.

The available options are `Normal Distance` and `Variable Normal Distance`:

- With option `Normal Distance` selected, set **Distance** to indicate the distance from the hub/shroud to the blade end, measured normal to the hub/shroud, with the blade in its data-defined position.
- With option `Variable Normal Distance` selected, set **Leading Edge Distance** and **Trailing Edge Distance** to indicate the distance from the hub/shroud to the corresponding edge of the blade end, measured normal to the hub/shroud, with the blade in its data-defined position. The tip gap varies linearly with meridional coordinate along the hub/shroud, from leading edge to trailing edge.

Figure 10.7: Blade with Partial Tips (p. 135) shows a blade with partial tips. Note that (although not shown in the figure) the blade buttons are shaped to allow rotation about the specified blade CAD rotation axis. To apply rotation from the data-defined orientation, even for a blade without partial tips, specify a non-zero stagger angle on the **Blade Stagger** tab of the **Blade CAD Features** object editor.

**Figure 10.7: Blade with Partial Tips**



A partial tip is considered to be a type of *complex blade end*. As a result, when you specify a partial tip:

- The `Complex Blade End` object appears in the object selector, in the `Geometry` branch, under the blade object. For details, see [Complex Blade End Object \(p. 139\)](#).
- Meshing high-fidelity geometry is required. Note that meshing high-fidelity geometry requires meshing with `ATM3D` or `ATM v2` topology. For details, see [Meshing High-Fidelity Geometry \(p. 219\)](#).

#### 10.3.6.3.2.2. Blade CAD Rotation Axis

For details, see [Blade CAD Rotation Axis \(p. 137\)](#).

### 10.3.6.3.3.2.3. Generated CAD Definition

For details, see [Generated CAD Definition \(p. 137\)](#).

#### 10.3.6.3.3.3. Blade Stagger tab

The **Blade Stagger** tab specifies whether to rotate the blade from its data-defined orientation about the defined blade CAD rotation axis, and, if rotating the blade, the angle by which to rotate it.

After applying the settings of the **Blade Stagger** tab, the `Blade Stagger` object appears, if applicable, under the blade object. The `Blade Stagger` object, when edited, opens the **Blade Stagger** tab of the **Blade CAD Features** object editor.

The settings of the **Blade Stagger** tab are discussed in the following sections:

[10.3.6.3.3.3.1. Blade Stagger](#)

[10.3.6.3.3.3.2. Blade CAD Rotation Axis](#)

[10.3.6.3.3.3.3. Generated CAD Definition](#)

#### 10.3.6.3.3.3.1. Blade Stagger

For a blade without blends, you can rotate the blade from its data-defined orientation by a specified blade stagger angle about the defined blade CAD rotation axis.

Set **Option** to one of the following:

- None

The blade is not rotated about the blade CAD rotation axis, so remains in its data-defined orientation (blade stagger angle of 0 degrees).

- Specified Angle

Set **Stagger Angle** to the angle by which to rotate the blade, about the blade CAD rotation axis, from its data-defined orientation. Note that the sense of positive rotation is established by the axis definition.

---

**Note:**

To avoid a degenerate mesh, the stagger angle must be within a reasonable range so that the blade (excluding the button) maintains an adequate amount of tip clearance.

---

---

**Note:**

If geometry generation fails after applying a non-zero stagger angle to a blade that has no partial tips, verify that the blade sufficiently intersects the hub and shroud boundaries. You might find it useful to use **View 3** (meridional view) for this purpose. For details, see [Selecting, Adding, and Deleting Views \(p. 85\)](#).

---

### 10.3.6.3.3.2. Blade CAD Rotation Axis

For details, see [Blade CAD Rotation Axis](#) (p. 137).

### 10.3.6.3.3.3. Generated CAD Definition

For details, see [Generated CAD Definition](#) (p. 137).

### 10.3.6.3.3.4. Common Settings

#### 10.3.6.3.3.4.1. Blade CAD Rotation Axis


Specify the X, Y, and Z coordinates of two points, **From** and **To**, that define a vector along the rotation axis of the *first* (as given by the profile point data) blade instance. The right-hand rule, as applied to that vector, determines the sense of positive rotation.

The **Blade CAD Rotation Axis** specification appears on each tab of the object editor as a convenience because it is needed by multiple CAD features (for example, to help define the shape and position of the button part of a partial tip, and to define the rotation axis when applying a blade stagger angle).

#### 10.3.6.3.3.4.2. Generated CAD Definition

The **Generated CAD Definition** settings are only available if, in the geometry browser (in the **Geometry** workspace), **Input Mode** is still set to `Profile Points`. These settings are provided as a convenience to allow you to specify the blade CAD feature settings, change the input mode to `CAD From Profile Points` and generate the geometry, all from the **Blade CAD Features** object editor. Once the input mode is set to `CAD From Profile Points`, the **Generated CAD Definition** settings are not displayed in the **Blade CAD Features** object editor and you must go to the geometry browser to make changes to the name of the CAD file.

The blade CAD feature settings are independent of the geometry except during generation of the CAD definition, at which time the CAD definition is updated according to the current blade CAD feature settings.

To change the specified CAD file to be used for the blade CAD features (and rest of the geometry), you can click the *Browse*  icon to select a CAD file using a browser: the **Set CAD File** dialog box.

#### 10.3.6.3.3.4.3. Apply And Generate CAD Button

Clicking the **Apply And Generate CAD** button applies the blade CAD feature settings and then generates (or regenerates) the CAD geometry.

After the blade CAD feature settings have been applied, one or more objects appear in the **Mesh** workspace tree, in the *Geometry* branch, under the applicable blade.

Generation (or regeneration) of the CAD geometry involves running *BladeEditor* or *BladeBuilder* temporarily in the background to generate the CAD file and associated *TurboGrid Initialization* file, then loading those two files into *TurboGrid*, resulting in new/updated CAD objects and (in accordance with the contents of the *TurboGrid Initialization* file) new/updated topological entities in the **Geometry** workspace tree.

The resulting CAD geometry not only includes the blade CAD features but the entire geometry, including the blade and passage.

#### 10.3.6.3.4.4. Apply Button

While working, you might want to reduce the number of times that the CAD geometry is generated because of the significant processing time required.

Clicking the **Apply** button applies the blade CAD feature settings without generating (or regenerating) the CAD geometry.

After the blade CAD feature settings have been applied, one or more objects appear in the **Mesh** workspace tree, in the *Geometry* branch, under the applicable blade.

To generate (or regenerate) the CAD geometry at any later time, even after saving a state file and then loading that state file into a new session of TurboGrid, you can select **File > Update Geometry** from the main menu or you can click the **Apply And Generate CAD** button.

#### 10.3.6.3.4. Leading Edge

The leading edge curve is the most upstream part of the blade. To modify the blade's leading edge, select the leading edge object from the object selector. The leading edge can only be defined after the blade has been defined.

##### 10.3.6.3.4.1. Leading Edge Tab


###### 10.3.6.3.4.1.1. Curve

The leading edge curve appears green in the viewer window and each point on the curve (one for each blade profile, or rib, curve) is marked with an octahedron symbol. The points are listed from hub to shroud. Edit the individual leading edge point locations to create the correct leading edge curve for the blade.

The quality of the mesh is dependent on the accuracy of the leading edge curve. In most cases, the initial placement of leading edge points is adequate. However there are some cases when some points may need to be moved.

Any change to the leading edge changes the blade surfaces, which changes the periodic surfaces as well as the hub and shroud surfaces. In order to avoid the delays associated with recreating the entire geometry after any modification to the leading edge, another step is required. The blade must be saved after the modifications to the leading edge are complete. For details, see [Save Blade As Command \(p. 36\)](#). The saved blade file contains data in the fourth field to designate the leading edge points. To complete the leading edge modification, the blade object must be updated with this new blade file. For details, see [The Blade Set Object and Blade Objects \(p. 124\)](#).

There are several steps required to edit the location of a point on the leading edge curve.

1. Select the point to edit by clicking on it in the list of points. Alternatively, you can pick the point to edit from the viewer after clicking **Select**  (on the viewer toolbar).
2. Click in any coordinate widget and then click the new location for the point in the viewer. The coordinates of the picked point are displayed in the object editor. Alternatively, type

the coordinates of the new location for the point or use the embedded sliders. The units of the coordinates are the base units or solution units. For details, see [Units \(p. 119\)](#).

3. Click **Apply** to save the new location of the current point.

*Picking Mode* can also be used to select and translate leading edge points in the viewer. For details, see [Selecting and Dragging Objects while in Viewing Mode \(p. 85\)](#). After moving the leading edge point, it will snap to the nearest point on the current profile curve.

If the leading edge points were designated in the blade file, it may not be necessary to adjust them since, in that case, the leading edge curve may already be acceptable. For details, see [Coordinate System and Blade File Definition \(p. 125\)](#).

#### 10.3.6.3.4.1.2. Save & Load Button

If a leading or trailing edge point has been moved, the **Save & Load** button appears. Clicking this button causes the modified blade profile data to be saved to a profile file (you will be prompted for the name), then reloaded (from the same file) in order for the new geometry to take effect. The blade and trailing edge objects also have this feature.

#### 10.3.6.3.5. Trailing Edge

The trailing edge curve is the most downstream part of the blade. The trailing edge curve appears red in the viewer window. Because of the similarity of the trailing edge with the leading edge, see [Leading Edge \(p. 138\)](#) for details.

#### 10.3.6.3.6. Complex Blade End Object

If you have defined a complex blade end (see [Meshing High-Fidelity Geometry \(p. 219\)](#)), the `Complex Blade End` object appears.

In the object selector, you can toggle the visibility check box of the `Complex Blade End` object, or any of its child objects, to toggle the visibility of the corresponding high-fidelity geometry in the **3D Viewer**.

For more information, see [Geometry Branch \(p. 225\)](#).

##### 10.3.6.3.6.1. Complex Blade End Tab

The **Topology Connection Tolerance** settings indicate the distances (one applicable near the hub and one applicable near the shroud) within which separate parts of the geometry should be topologically connected.

**Tolerance Mode** can be set to:

- `Relative`

The connection tolerance distances near the hub and shroud are both specified by a single **Relative Tolerance** value, which is a fraction of a characteristic length of the bounding box surrounding the whole geometry.

- `Absolute` (default)

The connection tolerance distances near the hub and shroud are specified by **Hub Absolute Tolerance** and **Shroud Absolute Tolerance**, respectively.

If the specified distance is too small, parts of the geometry that are intended to be connected might remain separate, causing the meshing process to fail while attempting to add extremely small elements between parts that are meant to be joined.

If the specified distance is too large, parts of the geometry that are intended to remain separate might be considered joined, causing a lack of mesh elements between parts that are meant to be separate.

---

**Note:**

The CEL parameter `Triangulation Tolerance` controls the CAD generation process, affecting the resolution of the resulting geometry. If you need to resolve features that are very small compared to a bounding box surrounding the whole geometry, or if some CAD surfaces are imported into TurboGrid improperly or not at all, consider reducing `Triangulation Tolerance` below its default of  $1e-4$ . Note that reducing the value increases memory and computational requirements. In order to reduce the value, open the **Command Editor** dialog box for the Geometry object in the **Mesh** workspace, set `Override Triangulation Tolerance = true`, and reduce `Triangulation Tolerance`. Note that, after adjusting `Triangulation Tolerance`, you might need to adjust **Topology Connection Tolerance**.

---

The **High-fidelity Hub Features** settings indicate the general extent of the high-fidelity features near the hub.

Regarding these settings:

- If any high-fidelity hub/blade geometry does not match the low-fidelity hub where it crosses the periodic boundaries of the passage, you must select **Cross Periodic Boundaries**.
- If you have selected **Cross Periodic Boundaries** and any part of the high-fidelity hub/blade geometry extends upstream of the inlet end of the passage domain, select **Extend Upstream of Inlet**. Similarly, if any part extends downstream of the outlet end of the passage domain, select **Extend Downstream of Outlet**.
- These settings affect the `Mesh Data` settings for the complex blade end, specifically the settings related to the tip shell that (when properly specified) surrounds the high-fidelity geometric features of the complex blade end. For details, see [Setting Descriptions for Complex Blade End Mesh Data Objects](#) (p. 225).
- To avoid producing an excessive number of unstructured mesh elements surrounding the high-fidelity geometric features, select these settings only as required.

The **High-fidelity Shroud Features** settings are analogous to the **High-fidelity Hub Features** settings.

### 10.3.6.4. The Hub Tip and Shroud Tip Objects

The shroud tip is the portion of the blade surface that exists as a result of the blade not extending all the way to the shroud. By default, the blade extends all the way to the shroud (that is, no shroud tip exists). To create a shroud tip, edit the `Shroud Tip` object after the blade has been defined.

The hub tip is similar to the shroud tip, except that it is at the hub end of the blade. The settings for the hub tip are stored in the `Hub Tip` object.

---

**Note:**

It is possible to specify a hub tip and a shroud tip for the same blade.

---

---

**Note:**

Using a tip clearance on one blade but not another is possible by changing the CCL for individual blades, but could produce an invalid mesh near the tip of any blade that does not have tip clearance.

---

#### 10.3.6.4.1. Hub Tip and Shroud Tip Tabs

The following sections describe the object editor settings for the `Hub Tip` and `Shroud Tip` objects:

##### 10.3.6.4.1.1. Override Upstream Geometry Options

##### 10.3.6.4.1.2. Clearance Type

#### 10.3.6.4.1.1. Override Upstream Geometry Options

Selecting the **Override Upstream Geometry Options** check box enables you to change the hub/shroud tip settings for the case of TurboGrid running in Workbench with an upstream geometry provided by BladeEditor. If you do not select the **Override Upstream Geometry Options** check box, the hub/shroud tip settings are displayed but are not editable, being controlled by the upstream BladeEditor geometry.

---

**Note:**

The shroud tip geometry, but not the hub tip geometry, can be defined in BladeEditor.

---

#### 10.3.6.4.1.2. Clearance Type

The **Tip Option** setting has the following options:

- None

This option causes the blade to extend from hub to shroud.

- Constant Span

This option creates a tip at the span location that you specify. A span of 0.0 represents the hub and a span of 1.0 represents the shroud. The span value for the hub tip must be between 0 and 0.5; the span value for the shroud tip must be between 0.5 and 1.0.

- Normal Distance

This option creates a tip so that the gap distance is whatever you specify.

- Variable Normal Distance

This option creates a tip so that the gap distance varies from leading edge to trailing edge using values that you specify. The gap distance varies linearly with m-coordinate as calculated on the hub (for a hub tip) or shroud (for a shroud tip).

- Profile Number

This option creates a tip at the blade profile that you specify. The available profiles are defined in the blade file. For details, see [The Blade Set Object and Blade Objects \(p. 124\)](#). The profile closest to the hub is profile 1.

---

**Note:**

- The blade surface and resulting mesh can be adversely affected when a blade shroud tip is defined by the second-last profile, and the last profile has a shape that is inconsistent with where the blade would exist if the blade were extended in the spanwise direction without the last profile. To avoid this problem, you can either ensure that the last profile is consistent with the second last profile, or manually remove the last profile from the curve file. A similar statement applies for the hub end of a blade having a hub tip.
  - If you are using the `CAD From Profile Points` input mode, the `Profile Number` option requires that **Include Tip Clearance in CAD File** is selected. For details, see [Defining Geometry from Generated CAD: CAD From Profile Points \(p. 112\)](#).
- 

### 10.3.6.5. The Low Periodic and High Periodic Objects

The low periodic surface extends from hub to shroud and inlet to outlet along the side of the mesh that has the lowest Theta values. (The direction of Theta is determined by the machine rotation axis and the right-hand rule.) The high periodic surface is on the opposite side of the passage from the low periodic surface (that is, on the high-Theta side).

The periodic surfaces can be modified only after the blade has been defined.

#### 10.3.6.5.1. Data Tab

Set **Method** to one of the following:

- Automatic

This option creates the periodic surface automatically. The shape of the periodic surface is based on the geometry of the adjacent blades.

- From File

This option allows you to load a periodic surface that was saved to a file. The **Length Units** setting controls the units that will be used to interpret the coordinate data in the file. The **Rotation Angle** setting transforms the loaded periodic surface by rotating by the specified angle about the machine rotation axis. Note that the **# of Bladesets** setting of the `Machine Data` object will not change automatically due to changes to the **Rotation Angle** setting.

### 10.3.6.5.2. Rendering Properties

To modify the color or rendering of the low or high periodic surface, right-click the `Low Periodic` or `High Periodic` object (respectively), then select **Render (Properties) > Edit Options**.

### 10.3.6.6. The Inlet and Outlet Objects

The `Inlet` object defines a surface that governs the meridional shape of the inlet end of the passage mesh. This end of the passage mesh can represent a stage interface. Note that if you subsequently add an inlet domain (via the `Mesh Data` object), this end of the passage mesh serves as an interface between the inlet domain and the passage mesh.

---

#### Note:

In this documentation, the upstream and downstream ends of the passage may be referred to as interfaces, even though they might represent the ends of the mesh, because they have the potential to become interfaces if you subsequently add inlet or outlet domains (via the `Mesh Data` object).

---

The `Outlet` object is similar to the `Inlet` object, except that it applies to the opposite end of the passage, and has an extra option for how it can be defined.

You can modify the `Inlet` and `Outlet` objects only after the blade has been defined.

The following topics are discussed:

[10.3.6.6.1. Inlet Tab or Outlet Tab](#)

[10.3.6.6.2. Trim Inlet Tab or Trim Outlet Tab - Changing the Inlet or Outlet Boundary](#)

[10.3.6.6.3. Using Stage Interfaces](#)

#### 10.3.6.6.1. Inlet Tab or Outlet Tab

The **Inlet** tab enables you to move and shape the interface between the inlet domain and the passage domain. The **Outlet** tab acts similarly for the interface between the outlet domain and the passage domain.

The following **Interface Specification Method** options are available, as applicable:

- **Parametric**

This is the default option. You specify hub and shroud locations parametrically. You have the option of making the interface conic or allowing it to bend according to the geometries of the blade, hub, and shroud. For details, see [Interface Specification Method: Parametric \(p. 144\)](#).

- **Points**

You specify a set of points that controls the shape and position of the interface. For details, see [Interface Specification Method: Points \(p. 145\)](#).

- **Adjacent blade**

You load a curve file that describes the shape of the next blade upstream or downstream of the passage, depending on whether you are defining an inlet or outlet object, respectively. TurboGrid then positions the interface automatically, taking into account the geometry of the blade being meshed and the geometry of the adjacent blade. For details, see [Interface Specification Method: Adjacent blade \(p. 147\)](#).

- **Fully extend**

This option causes all inlet points to move to the very beginning of the passage (as far away from the blade as possible), and all outlet points to move to the very end of the passage. This option can be used to prevent the blade from crossing the passage inlet or passage outlet.

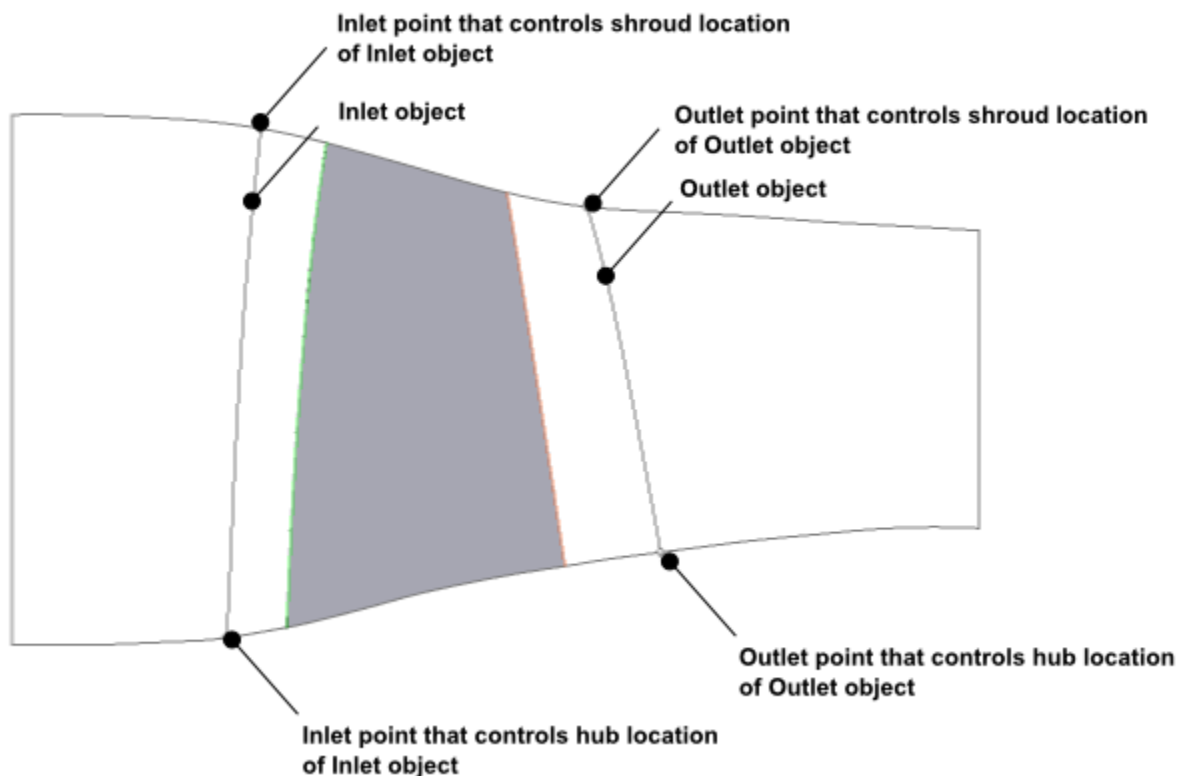
- **Meridional splitter** (available only on the **Outlet** tab)

You load a curve file that describes the shape of a meridional splitter that is immediately downstream of the passage. TurboGrid then positions the interface automatically, taking into account the geometry of the blade being meshed and the geometry of the meridional splitter. For details, see [Interface Specification Method: Meridional splitter \(p. 147\)](#).

#### 10.3.6.6.1.1. Interface Specification Method: Parametric

You can specify the hub and shroud locations of the interface in parametric space in one of two ways:

- By providing a number between 0 (all the way towards the blade) and 1 (all the way away from the blade) for each of the following settings: **Parametric Location Parameters > Hub**, **Parametric Location Parameters > Shroud**.
- By using the mouse to drag the inlet or outlet points in the viewer.



By default, the interface follows a straight path in parametric space between the specified locations on the hub and shroud. Viewed in A-R space, the interface appears to bend in accordance with the geometries of the blade, hub, and shroud. By selecting the **Use simple conic surface** option, you can override this default behavior and instead have the interface follow a straight line in A-R space between the specified locations on the hub and shroud. In this case, the interface forms a conic section in Cartesian space.

### 10.3.6.6.1.2. Interface Specification Method: Points

#### 10.3.6.6.1.2.1. Automatically generate required intermediate points





The **Automatically generate required intermediate points** option is available only before the hub, shroud, and blade are loaded. It is on by default. This option causes a set of intermediate points to be generated between the low hub point and low shroud point upon loading the geometry. TurboGrid aims to add an appropriate number of points, placed at locations that follow the leading or trailing edge of the blade.

#### 10.3.6.6.1.2.2. Curve

The inlet/outlet curve consists of a set of points that guide the meridional shape of the inlet end of the passage mesh as a function of the local spanwise direction. By default, a set of points is automatically generated (see [Automatically generate required intermediate points \(p. 145\)](#)).

The inlet/outlet curve can be defined by creating, modifying, or deleting points in the list of points. There must be a point at both the hub and shroud while additional points are optional. The points are connected linearly and must be listed in order from hub to shroud, or they will be ignored.

A point can be selected by clicking on it in the list of points or by picking it in the viewer window. For details on picking points in the viewer, see [Selecting and Dragging Objects while in Viewing Mode \(p. 85\)](#). The available icons become active when a point is selected. Alternatively, you can right-click an existing point in the list of points to see the same options. Each icon and its function is described in the following table:

Icon	Description
	<i>New</i> adds a point in the list of points below the selected point. The initial location of the new point on the Inlet is between the locations of the selected point and the following point in the list of points. You cannot create a new point after the low shroud point.
	<i>Generate intermediate points</i> removes all points except for the low hub point and the low shroud point, then automatically generates a set of zero or more points between them.
	<i>Delete</i> removes the selected point from the list of points. You cannot delete the low hub point or the low shroud point.
	<i>Read from File</i> opens a load file window. Select the file with the inlet points and click <b>Open</b> .

You can modify the location of the selected point by:

1. Typing in the coordinates (axial, radial, or both, depending on the selected **Method** option), or
2. Selecting and dragging the point in the viewer using the left mouse button. For details, see [How to Select and Move a Point \(p. 147\)](#).

Set **Method** to one of the available options:

- From A and R
- Set A
- Set R

Regardless of the method used to modify the location of the selected point, the location is restricted. The low hub point snaps to the line of intersection between the hub and low periodic surfaces. The low shroud point snaps to the line of intersection between the shroud and low periodic surfaces. Use the OUTLINE object as a guide when dragging the points. Any other points between the hub and shroud snap to the low periodic surface and are not constrained in the spanwise or streamwise directions. Inlet points cannot be moved further downstream on the low periodic surface than a position level with the leading edge of the blade. After you have clicked **Apply**, or dragged a point, the **Selected Point** setting will show the actual position to which the point has moved.

---

### Note:


The points are defined using the A-R coordinate system as opposed to the X-Y coordinate system.

---

The visibility of the inlet or outlet points is controlled by the **Point Visibility** check box.

### 10.3.6.6.1.2.3. How to Select and Move a Point

You can select and move points in the following ways:

- Select a point by holding **Ctrl+Shift**, and clicking the mouse while pointing at a point.
- Select and move a point by holding **Ctrl+Shift** and dragging using the left mouse button.
- To select and drag points without holding down **Ctrl+Shift**, you can click the *Select*  icon, then select and drag points with the left mouse button.

### 10.3.6.6.1.3. Interface Specification Method: Adjacent blade

Specify a curve file that defines the shape of the blade in the adjacent stage.

---

#### Note:

The hub and shroud curves are required to extend past this blade.

---

### 10.3.6.6.1.4. Interface Specification Method: Fully extend

The **Fully extend** option causes all inlet points to move to the very beginning of the passage (as far away from the blade as possible), and all outlet points to move to the very end of the passage. This option can be used to prevent the blade from crossing the passage inlet or passage outlet.

### 10.3.6.6.1.5. Interface Specification Method: Meridional splitter

Specify a curve file that defines the shape of a meridional splitter. Also specify **Length Units** and **Curve Type** to help interpret the curve file.

---

#### Note:

A hub tip is not allowed when using the **Meridional splitter** interface specification method.

---

### 10.3.6.6.1.5.1. Interface parametric locations

You can move the passage interface upstream or downstream by decreasing or increasing the **Hub** and **Shroud** values, which control where the interface meets the hub and shroud, respectively.

### 10.3.6.6.1.6. Point Visibility

The **Point Visibility** option controls the visibility of the control points that govern the shape of the inlet or outlet end of the passage mesh.

### 10.3.6.6.1.7. Control Angle

By default, TurboGrid automatically adjusts the geometry upstream and downstream of the blade. You can override the angle of the periodic surface relative to the rotation axis. This can

be useful for improving the quality of the mesh and/or guiding the mesh in the general flow direction.

### 10.3.6.6.2. Trim Inlet Tab or Trim Outlet Tab - Changing the Inlet or Outlet Boundary

The **Trim Inlet** tab enables you to trim the hub and shroud curves to effectively move the upstream end of the inlet domain. The **Trim Outlet** tab acts similarly for the downstream end of the outlet domain.

---

#### Note:

The upstream end of the inlet domain and downstream end of the outlet domain remain surfaces of revolution, formed by revolving the line segments that connect corresponding ends of the hub and shroud curves, whether or not those ends were moved due to trimming.

---

The following methods of choosing the hub and shroud curve trim locations are available:

- **Set A**

Specify two axial coordinates: one to trim the hub curve, the other to trim the shroud curve. This method offers precise control of trim locations for cases where the hub and shroud curves are aligned primarily in the axial direction near their respective trim locations. Conversely, this method is not suitable (and may fail) for cases where the hub or shroud curve is aligned closely with the radial direction near its trim location.

Ideally, each specified axial coordinate exists in exactly one place on the corresponding (hub or shroud) curve.

If a specified axial coordinate exists in multiple locations, TurboGrid then also considers the radial coordinate (which is displayed beside the specified axial coordinate) and chooses the nearest location. Note that the radial coordinate is changeable by selecting a point via the mouse (for example, by clicking near the intended trim location); it can also be changed using the **Command Editor** dialog box.

If a specified axial coordinate does not exist on the corresponding (hub or shroud) curve, a warning message is given and TurboGrid resets the specified axial coordinate (and associated radial coordinate) to the location of the curve end.

- **Set R**

The description for this method is the same as for *Set A*, except with the words "axial" and "radial" swapped.

- **AR Intersection**

Specify two points in the axial-radial plane. The line that passes through these points is used to trim the hub and shroud curves where it intersects those curves.

If there are multiple intersection points on a given curve, TurboGrid chooses one based on proximity to the line segment that connects the specified points.

If there are no intersections on a given curve, a warning message is given and TurboGrid resets one or both specified points to the location of a curve end.

### 10.3.6.6.3. Using Stage Interfaces

In order to produce a proper stage interface, you should follow these guidelines:

- Use the same hub curve for both stages, and the same **Curve Type** setting: `Bspline` or `Piece-Wise Linear`. The hub curve must extend through both stages to do this. Any trimming of the hub curve must be the same for both stages.

The same applies to the shroud curve.

As a less preferable alternative, you can use hub curves for each stage. The curves should meet at a point, and not overlap. When using this method, there is a risk of a discontinuity in slope where the curves meet.

- Use the same interface points for stage interfaces. Do this by saving the interface (inlet or outlet geometry object, as applicable), then loading the interface for the adjacent stage.

### 10.3.6.7. The Outline Object

The outline is a group of curves on the outer extents of the geometry. The outline includes the full extent of the hub and shroud read from the files, independent of the inlet and outlet locations. To modify the color or rendering of the geometry outline, right-click the `OUTLINE` object, then select **Render (Properties) > Edit Options**. You can only modify the outline after the blade has been defined.

## 10.4. Topology

---

The topology is a structure of blocks that acts as a framework for positioning mesh elements. The topology affects how the mesh is made, and can influence the mesh quality.

TurboGrid uses ATM and ATM3D optimized topology.

The object selector contains two types of topology-related objects:

- Topology Set

The `Topology Set` object handles general topology settings, and often handles all the topology settings.

- Topology

Each topology object handles details of the topology surrounding a particular blade. These details can be inherited from the `Topology Set` object, or (if applicable) can serve to fine tune the topology around a particular blade. Topology objects are given different names, to help distinguish the blade to which they apply; for example, the topology object for the main blade might be named "Main Blade".

This section discusses:

### [10.4.1. About Topology](#)

#### 10.4.2. Basic Usage

#### 10.4.3. The Topology Set Object

#### 10.4.4. Using Splitter Blades with ATM

#### 10.4.5. Using Tandem Vanes with ATM

#### 10.4.6. Advanced Local Refinement Control

#### 10.4.7. Span Location for Controlling Topology

#### 10.4.8. ATM3D

#### 10.4.9. ATM v2

### 10.4.1. About Topology

Topology blocks represent sections of the mesh that contain a regular pattern of hexahedral (hex) elements or an unstructured group of various element types. They are laid out adjacent to each other without overlap or gaps, with shared edges and corners between adjacent blocks, such that the entire domain is filled. By using topology blocks to control the placement of elements, a valid mesh can be generated to fill a domain of arbitrary shape. The topology is invariant from hub to shroud and is viewed on 2D layers that are located at various spanwise stations (see [Layers \(p. 183\)](#)).

The topology for blocks of hexahedral elements should be investigated at various layers (especially the hub and shroud layers) to check its quality since the mesh quality is directly dependent.

---

**Note:**

To visualize the topology on a layer, turn on the visibility using the visibility check box for that layer in the object selector and ensure that at least one topology visibility setting for the layer is turned on. The **Topology Visibility** setting controls the visibility of the yellow line segments that outline the topology block edges. For details, see [Topology Visibility \(p. 187\)](#). The **Master Topology Visibility** setting controls the visibility of the violet line segments that outline the master topology edges. For details, see [Master Topology Visibility \(p. 187\)](#).

---

A key feature of Ansys TurboGrid is the visibility of the surface mesh on the topology. As you adjust the topology, Ansys TurboGrid adjusts the surface mesh in real time so that the true effect of topology changes is visible. To help identify problem areas in the surface mesh before you generate the full 3D mesh, you can visualize mesh statistics on the layers. For details, see [Mesh Statistics \(p. 190\)](#)

Topology blocks contain a number of mesh elements along each side. The mesh elements vary in size across topology blocks in a way that produces a smooth transition within and between blocks. This is accomplished by shifting the nodes ("node biasing"<sup>[1]</sup>) toward, or away from, certain block edges. The topology blocks are positioned by default so that the mesh element sizes vary as smoothly as possible, given the constraints.

### 10.4.2. Basic Usage

The basic steps to using ATM Optimized topology are as follows:

---

[1] Node biasing is an important aspect of mesh generation because it helps to reduce the number of mesh elements.

1. Load the blade geometry and set the geometry parameters (for example, inlet/output points) as necessary.
2. Open **Topology Set**, set **Method** to an appropriate value, and click **Apply**.
3. Right-click **Topology Set** and clear **Suspend Object Updates**.

---

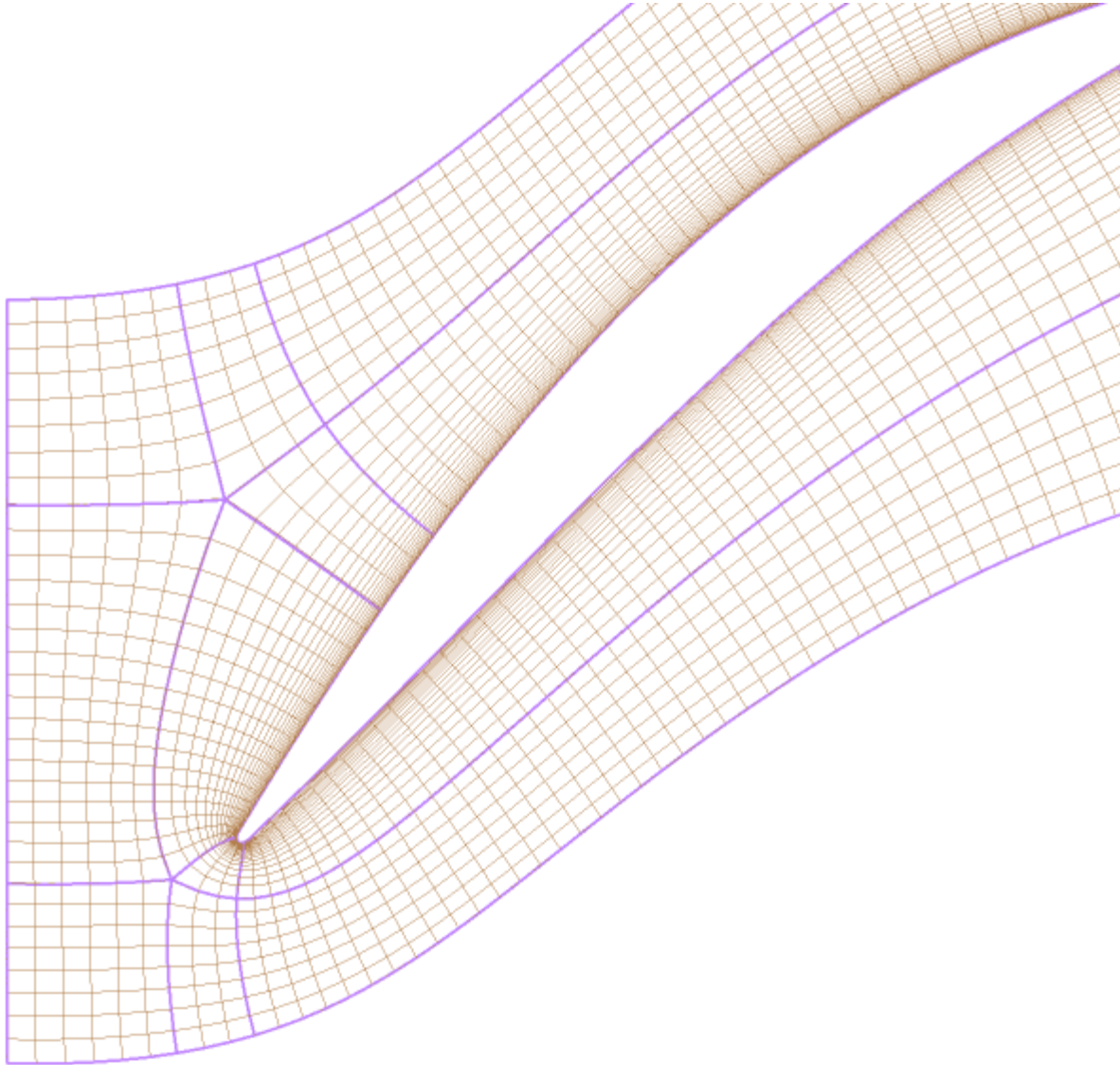
**Note:**

The **Undo** command cannot properly undo a suspension change (for example, to the **Topology Set** object); attempting to undo a suspension change can lead to an inconsistent state.

---

Ansys TurboGrid computes the master topology, topology, and refined mesh (the final mesh for the layer) for the hub and shroud layers based on the current Mesh Data specification.

After the topology has been generated, you can view the 2D mesh on a layer. If you open a layer object (for example, for the hub) in the object editor, you can control the visibility of the master topology, the topology, and the refined mesh; you can also see the mesh statistics for the layer. By default, the master topology and refined mesh are visible, as shown in the following figure.

**Figure 10.8: Master Topology and Refined Mesh on the Hub Layer for an Axial Compressor Blade**

The master topology is shown with heavy lines. The topology (not shown) serves as a background mesh that guides creation of the refined mesh. The refined mesh is shown with fine lines.

### 10.4.3. The Topology Set Object

This section includes:

[10.4.3.1. Definition Tab for the Topology Set Object](#)

[10.4.3.2. Details Tab for the Topology Set Object](#)

#### 10.4.3.1. Definition Tab for the Topology Set Object

The following topics are discussed:

[10.4.3.1.1. ATM Topology](#)

[10.4.3.1.2. Tip Topology Option](#)

### 10.4.3.1.3. Split Mesh Regions

### 10.4.3.1.4. Mesh Generation Mode

#### 10.4.3.1.1. ATM Topology

The **Method** setting enables you to generate an ATM topology in three basic ways.

- **Automatic**

Automatically selects the appropriate topology based on the blade style (cut-off or rounded edges) and blade angles. The topology family chosen by TurboGrid appears beside **ATM Topology > Used**.

- *[Topology Family]*

Allows you to manually select one of several topology packages, called families. If you have chosen a topology family that does not fit your geometry, you will receive an error message, and the topology will not be generated. For details, see [Manually Selecting a Topology Family](#) (p. 153).

- **Manual (Advanced)**

For details, see [Advanced Topology Control](#) (p. 154).

##### 10.4.3.1.1.1. Manually Selecting a Topology Family

By manually selecting a topology family, you are selecting one of the families that TurboGrid uses to generate a topology.

1. Load the blade geometry and set the geometry parameters (for example, inlet/output points) as necessary.
2. Open the **Topology Set** object.
3. Click **Apply**.
4. Click the **Topology Viewer** tab.
5. In **ATM Topology > Method**, scroll through the options in the drop-down menu to see the different topology families. A description of the family appears in the **Topology Viewer** (not the **3D Viewer**) when the family is selected.
6. Select the topology family you want to use.
7. Click **Apply**.
8. Right-click **Topology Set** and clear **Suspend Object Updates**.

---

**Note:**

The **Undo** command cannot properly undo a suspension change (for example, to the **Topology Set** object); attempting to undo a suspension change can lead to an inconsistent state.

---

If you have chosen a topology that fits your geometry, the topology will be generated.

You can view the details of the topology, regardless of whether it has been generated, by clicking the **Details** tab in the **Topology Set** editor. For more details, see [Details Tab for the Topology Set Object](#) (p. 157).

If you choose a topology that does not fit your geometry, the list of topology families in **ATM Topology > Method** shrinks to include only the topology that will fit your geometry. You may select one of the topology families displayed in the **Method** menu and click **Apply** to generate a topology that fits your geometry.

#### 10.4.3.1.1.2. Advanced Topology Control

To use advanced topology control:

1. Click **Edit > Options**.
2. Select **Enable Advanced Features** and click **OK**.
3. Open the **Topology Set** object and set **ATM Topology > Method** to **Manual (Advanced)**.

---

#### Note:

Advanced features are tested but require an advanced level of understanding to be used.

---

A list of the available topology templates is displayed in the object editor. Any templates found in the local working directory (instead of the TurboGrid template directory) will be listed in bold text.

[Table 10.1: Topology Template Naming Conventions](#) (p. 154) describes the naming conventions used for topologies.

**Table 10.1: Topology Template Naming Conventions**

Naming Convention	Description
Prefixed with "S1"	Applies to geometries consisting of a main blade and a single splitter blade
Prefixed with "LP"	Low passage topology pieces
Prefixed with "MP"	Main passage topology pieces
Prefixed with "HP"	High passage topology pieces
Contains "LE"	Applies to the leading edge of a single bladed geometry
Contains "TE"	Applies to the trailing edge of a single bladed geometry


Naming Convention	Description
Contains "Passage"	Applies to the area between blades

### Note:

- These terms can appear together. For example, the template LECircleLowPassage covers both the leading edge and passage areas.
- The topology templates are "passage centric", that is, they are designed to fit between the blade and a rotated copy of the blade. Before the topology is used, it is transformed to "blade centric" where it is placed around the blade.

You must select a combination of templates that covers the entire passage without overlap.

A preview of each topology indicating vertex and edge numbers is provided to help with the template selection process. Previews appear in the **Topology Viewer** when you double-click

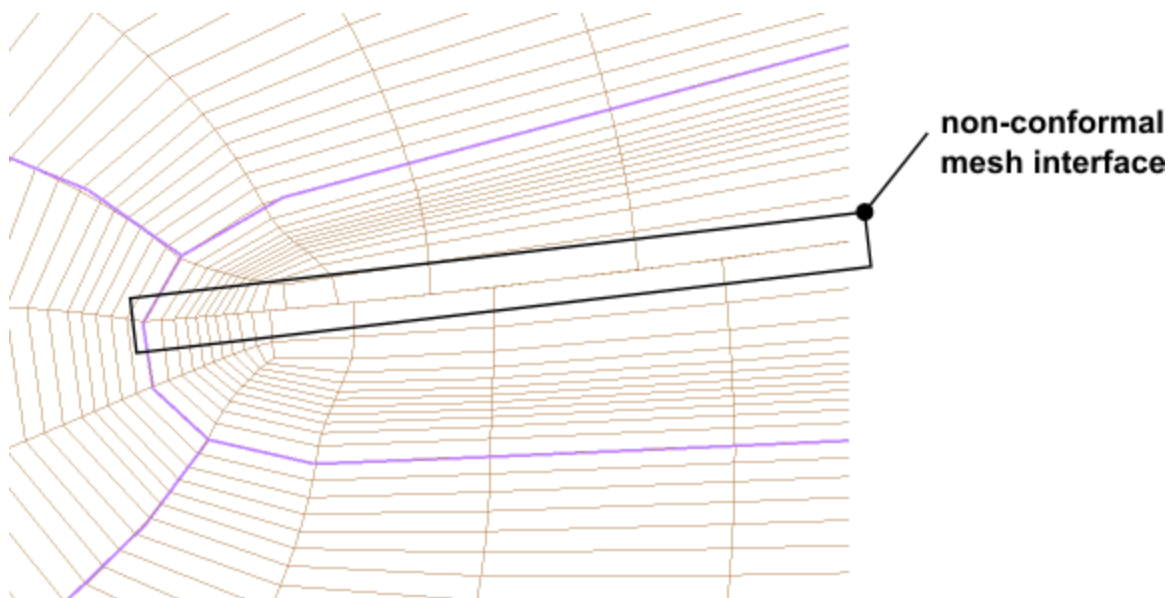
a topology or when you select a topology and then click *Display Topology* . For more details, see [Details Tab for the Topology Set Object \(p. 157\)](#).

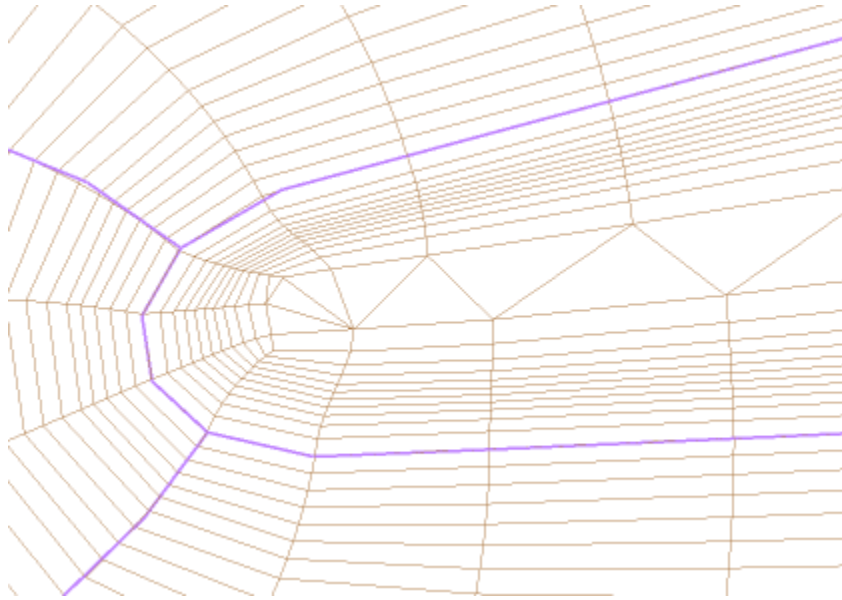
In the **Topology Viewer**, drag the mouse to pan across the image. To zoom in and out, roll the mouse wheel or drag the mouse vertically using the middle mouse button.

#### 10.4.3.1.2. Tip Topology Option

By default, TurboGrid produces a mesh that has a non-conformal mesh interface in the blade tip region. You can select **Tip Topology Option > Conformal Tip Enabled** to produce a conformal mesh in the tip region that is filled with unstructured wedge elements and that involves no mesh interface.

**Figure 10.9: Conformal Tip Not Enabled**



**Figure 10.10: Conformal Tip Enabled****Note:**

CCL parameter `Conformal Tip Minimum Quad Angle` (which can be found by editing the `Topology Set` object using the **Command Editor** dialog box) controls where the unstructured wedge region starts and ends along the blade. Starting from the leading edge and moving towards the trailing edge (and vice versa), quadrilateral elements (hexahedral elements in the 3D mesh) are used to fill in the space until the minimum inside angle of a quadrilateral element would drop below the specified value, at which point triangular elements (wedge elements in the 3D mesh) are used to continue filling in the space. Note that the interior angles of triangular elements may be less than the specified value of `Conformal Tip Minimum Quad Angle`.

**10.4.3.1.3. Split Mesh Regions**

The **Split Mesh Regions At Leading Edge** option generates a line of rotation from each cut-off point of a "cut-off or square" (see [Cut-off or square \(p. 130\)](#)) blade leading edge. The **Split Mesh Regions At Trailing Edge** option generates a line of rotation from each cut-off point of a "cut-off or square" blade trailing edge.

A "cut-off or square" edge having the same axial and radial coordinates at both cut-off points will have lines of rotation that coincide. In this case, the applicable option will divide the passage mesh, hub, and shroud at the cut-off edge by a surface of revolution about the machine axis. Additional 2D and 3D regions will be created as a result of this division of the passage. This allows, for example, the mesh to be used in a simulation where the portion of the passage mesh surrounding the blade rotates, while the rest of the passage mesh is stationary.

These options may still be used if the cut-off points do not have the same axial and radial coordinates, although the interface between the divisions will not be a surface of revolution.

### 10.4.3.1.4. Mesh Generation Mode

The **Mesh Generation Mode** setting has the following options:

- **ATM** — enables traditional ATM topology.
- **ATM3D** — enables the ATM3D meshing approach (see [ATM3D \(p. 165\)](#)), which supports the meshing of high-fidelity geometry (see [Meshing High-Fidelity Geometry \(p. 219\)](#)).
- **ATM v2** — supports most functionality offered by **ATM** plus other functionality (see [ATM v2 \(p. 166\)](#)).

---

#### Note:

You can set the ATM3D approach to be used by default by setting a preference. For details see [TurboGrid Options \(p. 48\)](#).

---

The following table compares the mesh generation modes:


**Table 10.2: Mesh Generation Mode Comparison**

ATM	ATM3D	ATM v2
Meshing with high-fidelity geometry is not supported.	Meshing with high-fidelity geometry is supported.	Meshing with high-fidelity geometry is supported.
Cut-off leading/trailing edges are supported.	Cut-off leading/trailing edges are not supported.	Cut-off leading/trailing edges are supported only if not meshing with high-fidelity geometry.
Square trailing edges can be approximated using high curvature.	Square trailing edges are not supported.	Square trailing edges are supported (for example, for hydro turbines such as Kaplan and Francis).

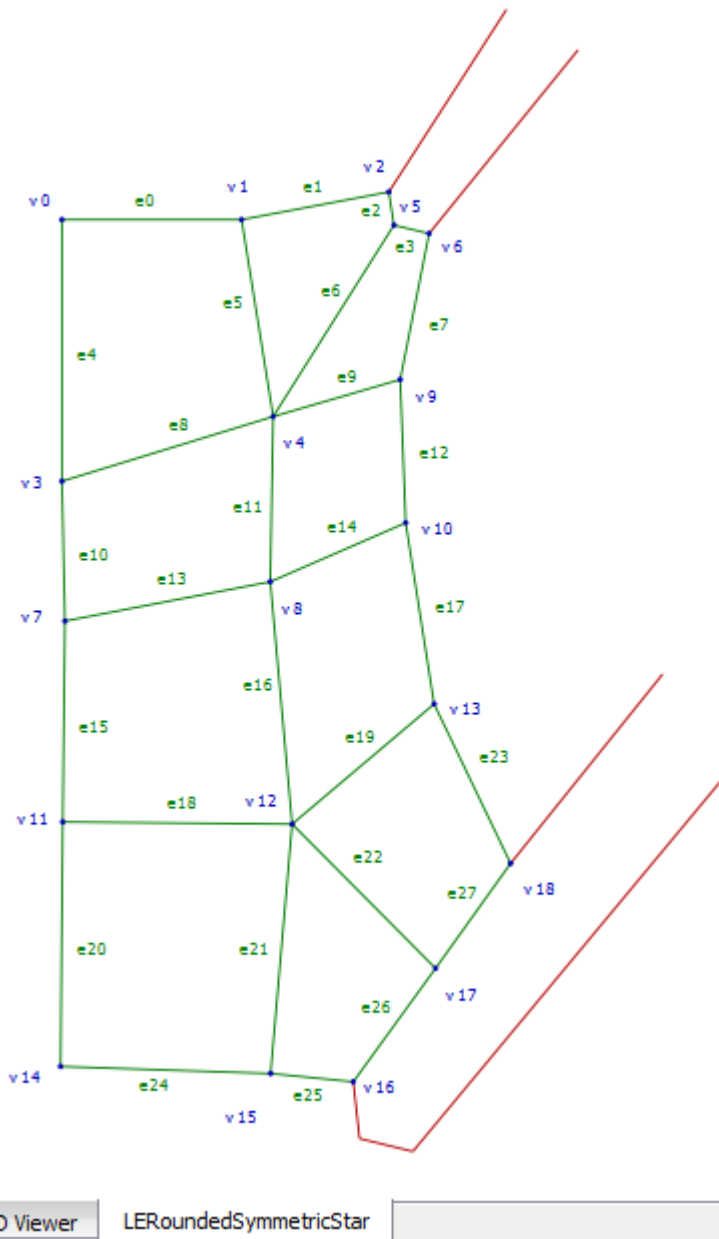
### 10.4.3.2. Details Tab for the Topology Set Object

The **Details** tab lists the topology templates that are being used in the current case.

To display one of the listed topology templates in the **Topology Viewer**, you can do any of the following:

- Right-click a listed topology template and select the **Display Topology** context menu command.
- Select a listed topology template and then click *Display Topology* .
- Double-click a listed topology template.

In the **Topology Viewer**, blue labels are used to indicate topology nodes and green labels are used to indicate topology edges. Blades are outlined in red. As an example, the figure below shows the LERoundedSymmetricStar topology template:



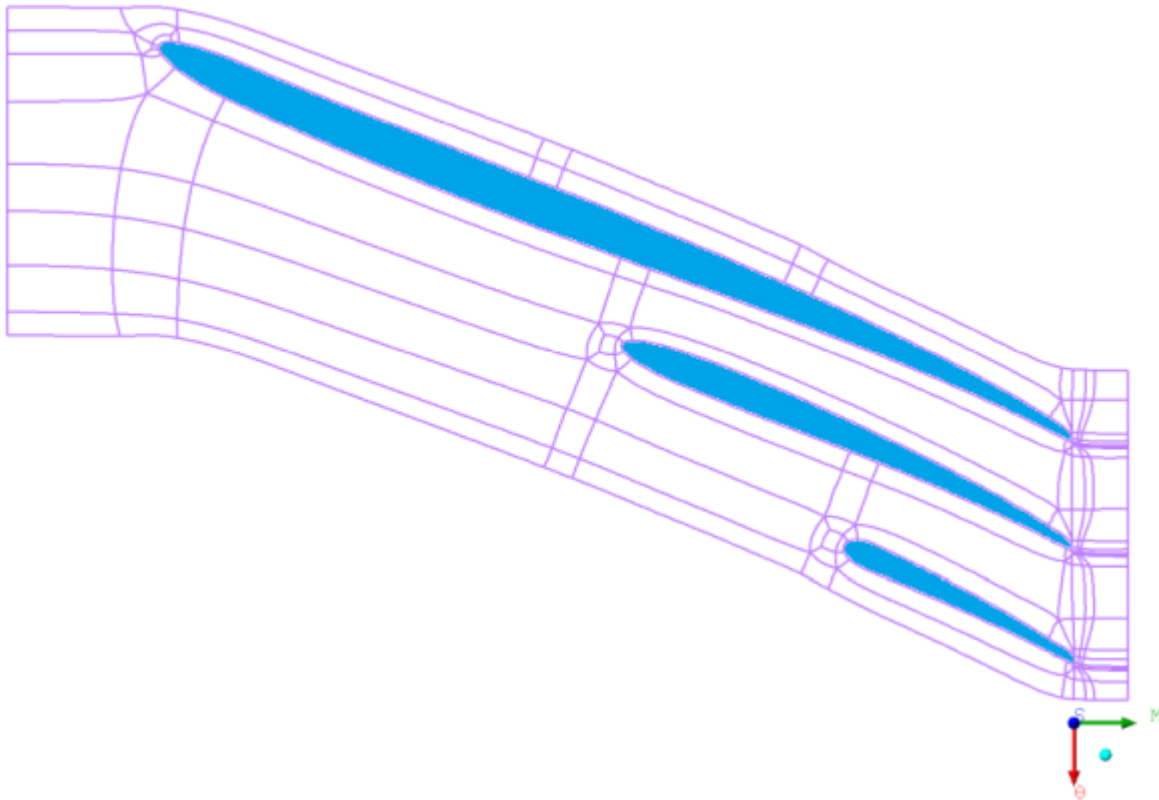
### 10.4.4. Using Splitter Blades with ATM

The ATM method supports up to three splitter blades, provided that the following criteria are met:

- The leading edges of the main and splitter blades are both rounded.
- The trailing edges of the main and splitter blades are both rounded or both cut-off.
- The leading edge of the main blade is positioned, in the meridional direction, significantly ahead of the leading edge(s) of the splitter blade(s).

- The trailing edges of the main and splitter blades, if cut-off, are at the same meridional position.
- Considering the specified direction of theta, the main blade is in the lowest theta position and the shortest-meridional-length splitter blade is in the highest theta position. You can specify the theta direction in the **Machine Data** object. For details, see [Rotation](#) (p. 119).

**Figure 10.11: Example Template for Dual Splitter Blades**



### 10.4.5. Using Tandem Vanes with ATM

The ATM method supports the following tandem vane templates:

**Table 10.3: Tandem Vane Templates**

Template Name	Description
TandemVaneAlignedHigh TandemVaneAlignedLow	<p>The leading edge of the second blade is positioned, in the meridional direction, near or behind the trailing edge of the first blade.</p> <p>The second blade should be offset by approximately half a blade pitch (in the theta direction) compared to the first blade.</p> <p>The leading and trailing edges of the first blade and second blade are rounded.</p>

Template Name	Description
	You must choose one of the two templates. For details, see <a href="#">Choosing the Appropriate TandemVaneAligned Template (p. 160)</a> .
TandemVanesCustomizedNr1	<p>The leading edge of the first blade is positioned, in the meridional direction, near or behind the trailing edge of the second blade.</p> <p>The leading edge of the first blade is positioned at a lower Theta value than the trailing edge of the second blade.</p> <p>The leading edge of the first blade is rounded. The trailing edge of the first blade is cut-off. The leading and trailing edges of the second blade are rounded.</p> <p>For details, see <a href="#">The TandemVanesCustomizedNr1 Template (p. 163)</a>.</p>

To access these templates, enable advanced features. For details, see [Advanced Topology Control \(p. 154\)](#).

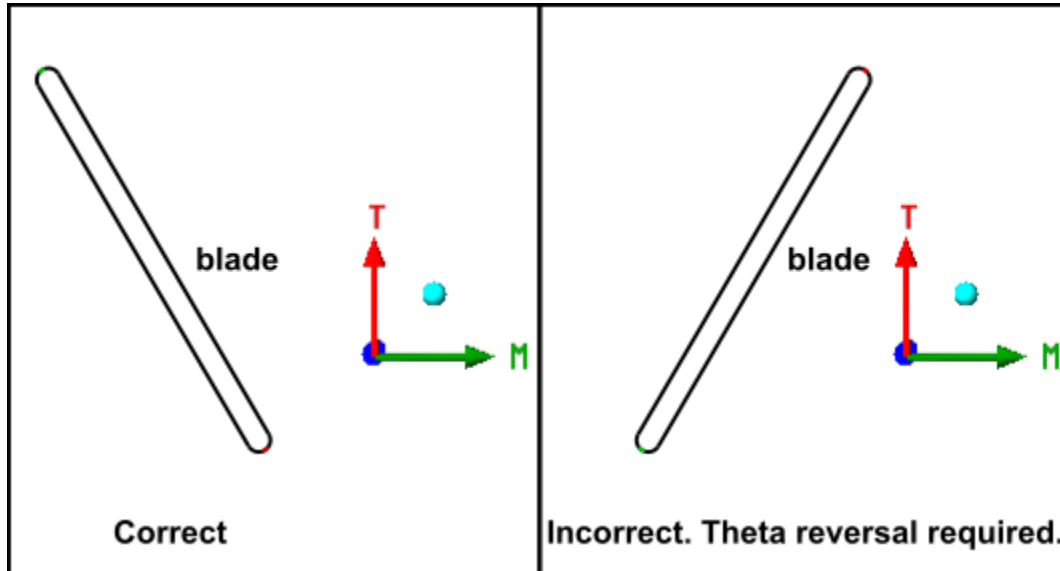
#### 10.4.5.1. Choosing the Appropriate TandemVaneAligned Template

The tandem vane template has a "Low" and "High" version. To determine which tandem vane template is appropriate:

1. Right-click in the viewer and select **Transformation > 3D Turbo (Theta-M'-Span)**.
2. Orient the view so that the meridional coordinate (M) increases to the right and Theta (T) increases upward.

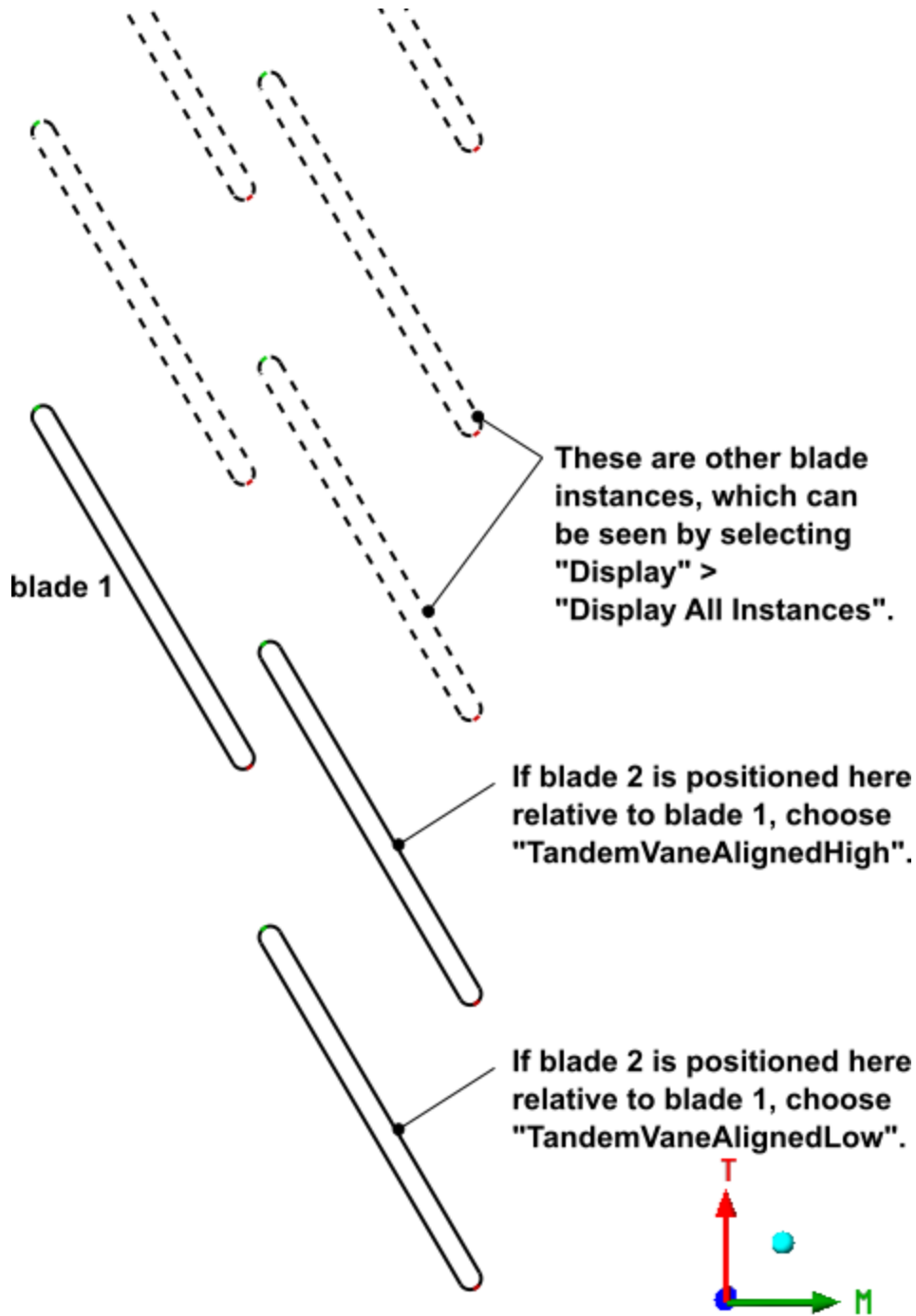
After you do this, the leading edges are on the left.

3. Proceed to the next step if Theta decreases towards the trailing edge (towards higher meridional coordinate). Otherwise, use the settings under **Geometry > Machine Data > Rotation** to reverse the direction of increasing Theta in relation to the geometry.

**Figure 10.12: Theta Definition Requirement**

4. If Theta increases from the first blade's trailing edge to the second blade's leading edge, select TandemVaneAlignedHigh. Otherwise, select TandemVaneAlignedLow.

**Figure 10.13: Choosing the Tandem Vane Template**

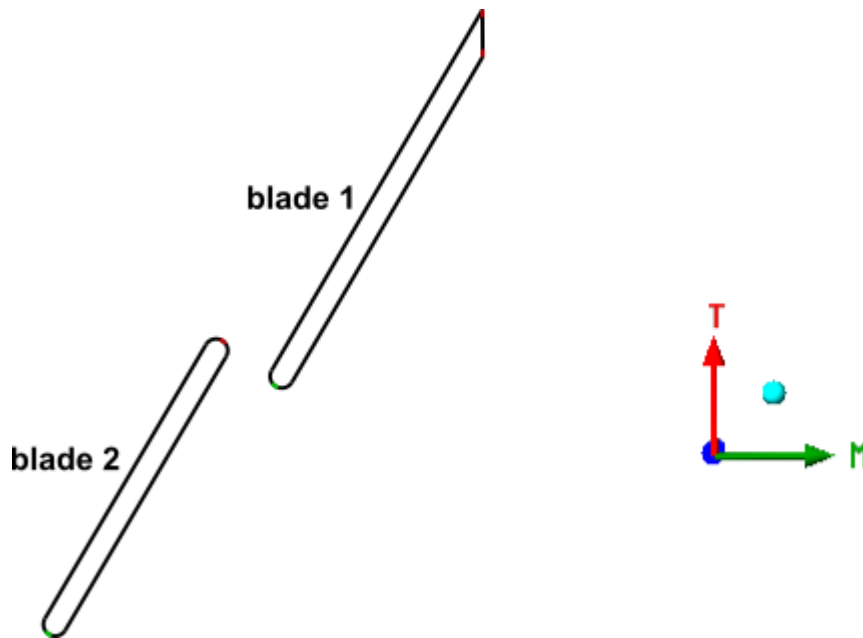


**Note:**

When running TurboGrid in stand-alone mode (as described in [Using the Ansys TurboGrid Launcher in the TurboGrid Introduction](#)), it is possible to rotate the second blade along the Theta direction, which can potentially change which template is suitable, as determined by the procedure above.

### 10.4.5.2. The TandemVanesCustomizedNr1 Template

An example geometry that is suitable for use with the TandemVanesCustomizedNr1 template is shown below:



### 10.4.6. Advanced Local Refinement Control

When TurboGrid initializes the ATM topology, it builds the topology in pieces and then conglomerates the block structure to minimize the number of blocks in the final topology. Because you can control only the number of elements along a given topology edge and not the distribution, having more blocks in specific areas, for example near the trailing edge, gives you better control of the mesh resolution in these areas. Preserving specific vertices from being conglomerated in these situations will give you better control of the local mesh resolution — the specific vertices will be dependent on the topology pieces used. For a list of the available topology pieces, see [Advanced Topology Control \(p. 154\)](#).

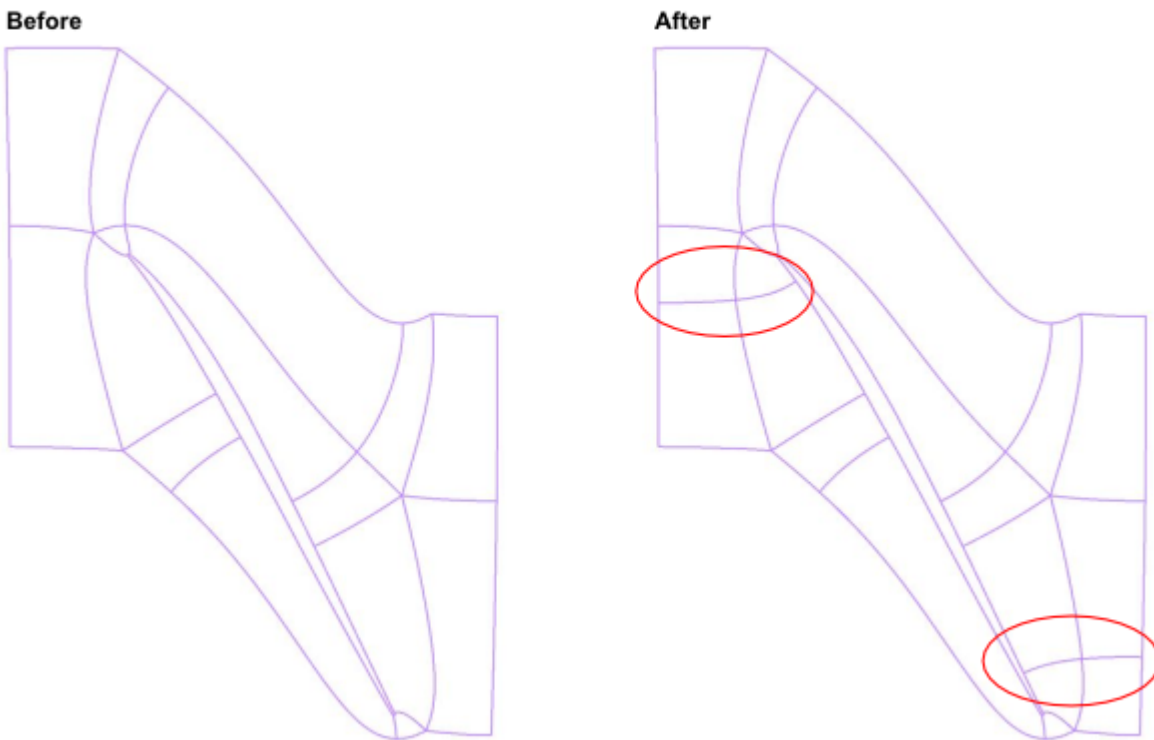
You can retain additional vertices with CCL of the following format: "template name", "vertex number". This must be repeated for each new vertex number, leading to the form:

```
TOPOLOGY SET:
  Override Conglomeration Retained Vertices List = true
  Conglomeration Retained Vertices List = "template name", "vertex number", "template name", "vertex number", ...
END
```

The following is an example of CCL that provides additional control near the leading and trailing edges for rounded blades:

```
TOPOLOGY SET:
  Override Conglomeration Retained Vertices List = true
  Conglomeration Retained Vertices List = LERoundedSymmetricStar, 0, TERoundedSymmetricStar, 0, \
  LERoundedSymmetricStar, 13, TERoundedSymmetricStar, 13
END
```

The figure below demonstrates the result of applying the example CCL.



You can apply edge splits using ATM Background Topology Edge Refinement to influence the passage topology background mesh count. One typical example is to apply edge splits on widely spaced blades. The following CCL is used to apply eight edge splits on edges 8 and 10 of PassageBlock1by4:

```
TOPOLOGY SET:
  ATM Background Topology Edge Refinement = PassageBlock1by4, 8, 8, PassageBlock1by4, 10, 8
END
```

For cases where ATM Background Topology Edge Refinement increases the passage topology background mesh count, the final mesh count at the specified edges also goes up. You may want to add an edge split on those edges in order to decrease the final mesh count in the blade-to-blade direction.

### 10.4.7. Span Location for Controlling Topology

The span is the distance between the hub and shroud, expressed as a fraction between 0 and 1. The span is calculated when  $m'$ -Theta coordinates are created. For details, see [Transformation Commands and Coordinate Systems](#) (p. 206).

Ansys TurboGrid uses a span parameter for several reasons related to topology, including:

- Obtaining blade metal angles so that Ansys TurboGrid can automatically set topology types after you have specified the topology method: H/J/C/L-Grid.
- Calculating an appropriate number of topology blocks between the blade and the periodic interface or interface between blades.

By default, the span location is set to 0.5 (that is, 50% of the span). To adjust the span parameter, called `Span Location`, edit the `Topology Set` object using the **Command Editor** dialog box.

### 10.4.8. ATM3D

ATM3D is an alternative meshing approach to ATM. Whereas the original ATM meshing process uses a quasi-3D mesh smoothing process, ATM3D uses a full 3D mesh smoothing process. The goal of ATM3D is to ultimately produce a higher quality mesh, particularly for cases that are challenging for ATM.

Initially, ATM3D is not active by default. To use it by default, select **Edit > Options**, then, in the **Options** dialog box, with `TurboGrid` selected in the tree, select **Use ATM3D Meshing By Default** and click **OK**. This preference takes effect the next time you start TurboGrid.

For a given case, you can activate ATM3D by opening `Topology Set` and setting **Mesh Generation Mode** to ATM3D.

#### Limitations of ATM3D

- **Geometry:** Cut-off blade geometry is not supported: If the geometry has a cut-off blade, ATM3D is turned off regardless of the **Use ATM3D Meshing By Default** preference setting. For details, see [Cut-off or square \(p. 130\)](#).
- **Layers:** Compared to ATM, ATM3D has less need to adjust the number of layers. When using ATM3D, the `Layers` object's **Layers** tab > **Intermediate Layers** > **Insertion Mode** setting can only be set to `Manual - Uniform`. You can only change the number of intermediate layers via the **Count** setting. For details, see [Defining Intermediate Layers \(p. 184\)](#).
- **Layers:** Unlike the ATM approach, ATM3D does not have a setting to change the type of the spanwise mesh interpolation guide curves that connect the layers. For details, see [Spanwise Mesh Interpolation Guide Curves \(p. 186\)](#).
- **Mesh Data:** Inserting/editing edge splits is not supported. For details, see [Changing the Number of Elements on a Selected Master Topology Edge \(p. 182\)](#).

Also see [Table 10.2: Mesh Generation Mode Comparison \(p. 157\)](#).

---

#### Note:

- If you encounter high aspect ratio elements along the leading/trailing edge, or elements with poor angles:
  - Try adjusting the number of layers.
  - Try increasing the value of parameter `Domain Meshing Smoothing Iterations After Remesh`. This parameter is accessed by right-clicking the `3D Mesh` object and selecting **Edit in Command Editor**.
- With ATM3D, layers show neither the refined boundary layer mesh around the blade nor the blade tip mesh. You can, as usual, inspect the mesh using the `3D Mesh > Show Mesh` object.

## 10.4.9. ATM v2

The ATM v2 mesh generation mode is intended to be an improved version of the ATM mesh generation mode, and can generally be used as an alternative to the ATM mesh generation mode.

ATM v2 supports meshing with high-fidelity geometry, to support geometric features such as complex blade ends. However, adding a complex blade end can limit support for other geometric features.

Some templates require ATM v2 mesh generation. For example, the following templates require ATM v2:

### Templates for Kaplan Turbines having a square trailing edge and a blade stagger angle around 90 degrees

- Single Round Square TE
- Single Refined Round Square TE (with topology refinement around the leading edge)


### Templates for Francis turbine blades having a square trailing edge

- Single Round Square TE Francis Low
- Single Splitter Round Square TE Francis Low

ATM v2 supports meshing with high-fidelity geometry. However, unless the selected template requires ATM v2, ATM3D is the preferred mesh generation mode for meshing with high-fidelity geometry.

A comparison of the mesh generation modes is given in [Table 10.2: Mesh Generation Mode Comparison \(p. 157\)](#).

## 10.5. Mesh Data

To define the mesh properties for the entire mesh, edit the `Mesh Data` object from the object selector or click *Edit Mesh Data* . To define or modify the mesh properties of a blade-specific portion of the mesh, edit the corresponding object stored under the `Mesh Data` object.

Defining the `Mesh Data` object does not create the mesh. The mesh data can only be set after the topology has been defined, but it is recommended that you set the required mesh size before you create the layers (discussed later), because the true mesh nodes are displayed on the topology layers as the topology is adjusted. For details on creating a mesh, see [Steps to Create a Mesh \(p. 101\)](#) and [Mesh Command \(p. 59\)](#).

### 10.5.1. The Mesh Data Objects

The `Mesh Data` object contains settings that affect the mesh globally.

The blade-specific mesh data objects contain a subset of the settings of the `Mesh Data` object, and affect the mesh for individual blades in the blade set.

The complex blade end `Mesh Data` objects affect the mesh for individual complex blade ends.

The following topics are discussed:

10.5.1.1. Mesh Size Tab

10.5.1.2. Passage Tab

10.5.1.3. Hub Tip/Complex Blade End and Shroud Tip/Complex Blade End Tabs

10.5.1.4. Distribution Settings in General

10.5.1.5. Inlet and Outlet Tabs

10.5.1.6. Mesh Around Blade Tab

10.5.1.7. Complex Blade End Mesh Data Settings

## 10.5.1.1. Mesh Size Tab

The **Mesh Size** tab is applicable to the Mesh Data object.

### 10.5.1.1.1. Lock Mesh Size Check Box

In some situations, for example when you are comparing minor geometry variations, you may want to keep the mesh dimensions the same from one case to the next. This will reduce the influence of the mesh on the CFD results. To lock the mesh dimensions, select the **Lock mesh size** check box. This setting will maintain the current mesh dimensions and prevent any subsequent changes to the global or local mesh controls. The lock status will be preserved from one session to the next if the project or state file is saved. To unlock the mesh, clear the **Lock mesh size** check box.

### 10.5.1.1.2. Method

The **Method** setting controls the mesh density, and can be set to one of the following:

- Target Passage Mesh Size

The Target Passage Mesh Size method sets a target for the number of nodes in the mesh passage. The pre-set options for the **Node Count** setting are Coarse (20000), Medium (100000), and Fine (250000). There is also an option, Specify, to specify the target number of nodes. In this mode, if you change the spanwise mesh size or the boundary layer refinement, or make any local refinements to the mesh, Ansys TurboGrid will adjust the mesh size factor (Global Size Factor) automatically to achieve the desired target mesh size.

Ansys TurboGrid attempts to reach the target number of nodes by:

- Adjusting the number of elements placed along each topology block edge. The number of elements can be viewed after pressing **Apply** by changing the method to Global Size Factor.
- Adjusting the number of elements from hub (or hub tip) to shroud (or shroud tip) if these have not been explicitly set.

- Global Size Factor

The Global Size Factor method defines the overall mesh size. To increase the resolution of the mesh, increase the size factor using the **Size Factor** setting. Note that the change in overall mesh size is not linear. In this mode, if you change the spanwise mesh size or the

boundary layer refinement, or make any local edge refinements to the mesh, the Global Size Factor will stay fixed and the overall mesh size may change. This factor, when used with proportional refinement, can be used to scale the mesh size for a mesh refinement study.

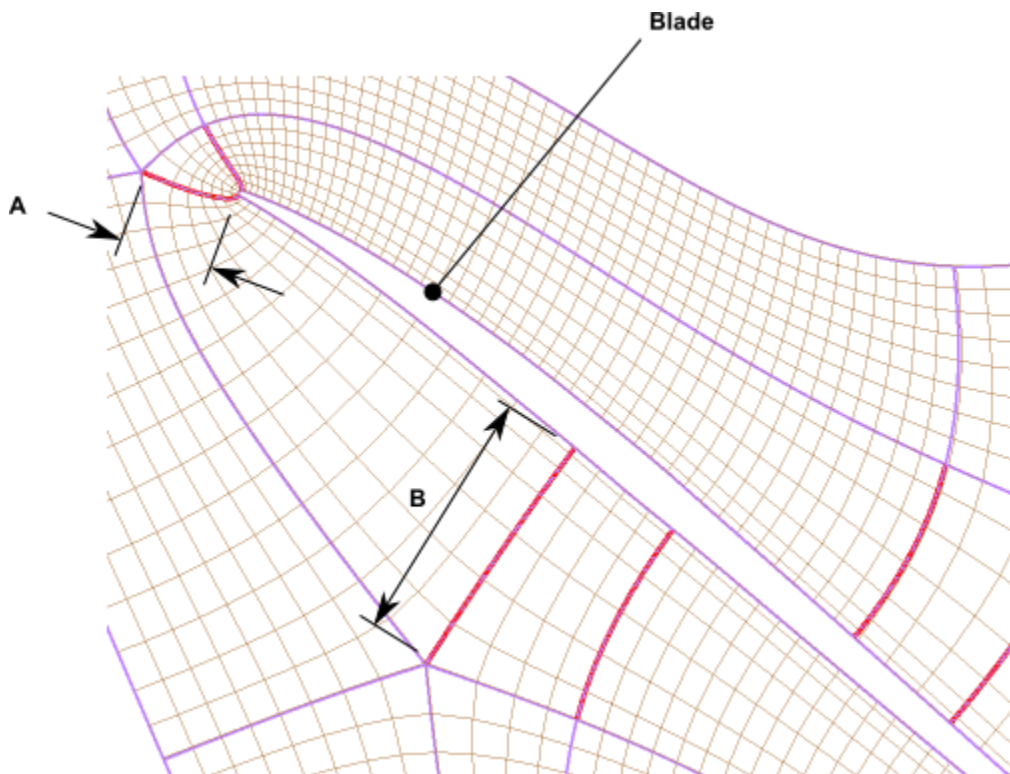
### 10.5.1.1.3. Boundary Layer Refinement Control

In the context of the **Boundary Layer Refinement Control** settings, the boundary layer region is represented by:

- The blade boundary layer region, which is the set of topology blocks along the sides of the blade, plus
- The mesh near the hub and shroud.

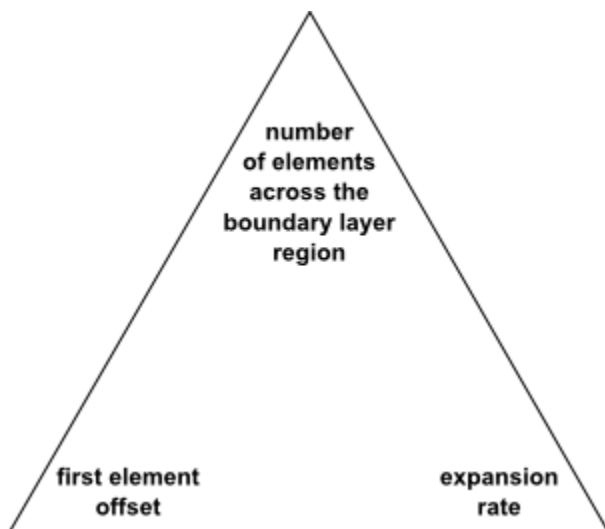
The blade boundary layer region varies in thickness (outward from the blade) around the blade profile, as indicated by distances A and B in [Figure 10.14: Variations in boundary layer region thickness \(p. 168\)](#).

**Figure 10.14: Variations in boundary layer region thickness**



Within the blade boundary layer region, TurboGrid uses various expansion rates to distribute the elements normal to the blade surface to assure a smooth transition to the passage mesh.

The blade boundary layer region thickness is a function of the variables shown in [Figure 10.15: Variables that control the blade boundary layer region distribution \(p. 169\)](#).

**Figure 10.15: Variables that control the blade boundary layer region distribution**

The number of elements outward across the blade boundary layer region is constant around the blade profile. Because the blade boundary layer region thickness (see [Figure 10.14: Variations in boundary layer region thickness \(p. 168\)](#)) changes around the blade, either the expansion rate or the first element offset (that is, the height of the first row of elements next to the blade) can be constant, but the other must change.

You can specify which properties of the boundary layer region to control by using the **Boundary Layer Refinement Control > Method** setting. The options are:

- Proportional to Mesh Size

This option controls the number of elements outward across the blade boundary layer region in proportion to the values specified for **Factor Base** and **Factor Ratio**. This method maintains similar expansion rates when the Global Size Factor is changed. If the blade has a cut-off edge, **Factor Ratio** also controls the number of elements along the cut-off edge.

Increasing the value of **Factor Base** or **Factor Ratio** increases the number of elements outward across the blade boundary layer region and along the cut-off edge (if applicable). Conversely, decreasing the value of **Factor Base** or **Factor Ratio** decreases the number of elements. The number of elements outward across the blade boundary layer region is calculated as  $\text{Base Count} * \text{Global Size Factor} * (\text{Factor Base} + \text{Factor Ratio} * \text{Global Size Factor})$ . The default values of Factor Base and Factor Ratio are 3 and 0 respectively.

- First Element Offset

This option directly controls the height of the first row of elements next to the blade.

- First Element Offset and Element Count

This option controls the height of the first row of elements next to the blade, hub, and shroud, via parameter **Offset** and the number of elements outward across the blade boundary layer region via parameter **Count**.

When using the First Element Offset and Element Count option, **Target Maximum Expansion Rate** (described in [Target Maximum Expansion Rate \(p. 171\)](#)) is not applicable.

- Edge Refinement Factor

This factor is multiplied with Base Count \* Global Size Factor to determine the number of elements outward across the blade boundary layer region.

When **Constant First Element Offset** is selected, you can influence the range of expansion rates using the **Target Maximum Expansion Rate** settings, described in [Target Maximum Expansion Rate](#) (p. 171).

When the topology is created, Ansys TurboGrid creates a `Boundary Layer Control` object under the `Mesh Data` object in the object selector. The `Boundary Layer Control` object shows information about the boundary layer.

---

**Note:**

- When **Boundary Layer Refinement Control > Method** option is set to `First Element Offset`, the offset value can be controlled either in the `Boundary Layer Control` object or the **Mesh Data > Mesh Size** tab.
- When **Boundary Layer Refinement Control > Method** option is set to `First Element Offset` and `Element Count`, the offset and count values can be controlled either in the `Boundary Layer Control` object or the **Mesh Data > Mesh Size** tab.

---

#### 10.5.1.1.3.1. Constant First Element Offset

When **Constant First Element Offset** is selected, TurboGrid uses a double-sided node distribution (a non-constant expansion rate) for the boundary layer. In this case, the near-wall expansion rates that are reported in the `Boundary Layer Control` object indicate the approximate minimum and maximum values based on a sample of the layer values.

When **Constant First Element Offset** is not selected, TurboGrid uses a single sided node distribution for the boundary layer; at any given position around the blade (in the blade-to-blade view), there is a constant expansion rate across the boundary layer. In this case, the user-specified first element offset is applied approximately as an average offset.

---

**Note:**

Mesh element aspect ratios (spanwise length to wall offset length) on the blade surface at the leading and trailing edges might be higher with **Constant First Element Offset** not selected compared to with **Constant First Element Offset** selected. With ATM3D, you can reduce these aspect ratios by increasing the spanwise mesh resolution or by increasing the boundary layer first element offset. If you see solver convergence issues related to high aspect ratios, try running a double precision solver.

---

#### 10.5.1.1.3.2. Cutoff Edge To Boundary Layer

The **Cutoff Edge To Boundary Layer** setting is available only for cut-off blades.

You can adjust **Cutoff Edge To Boundary Layer > Factor** to change the expansion rate used near the cut-off edge.

#### 10.5.1.1.3.3. Cutoff Edge Split Factor

The **Cutoff Edge Split Factor** setting is available only for cut-off blades.

In cases where the leading edge and/or trailing edge is cut-off, you can specify a factor for each cut-off edge to control the number of elements along that edge (as viewed in any given layer).

#### 10.5.1.1.3.4. Target Maximum Expansion Rate

The **Target Maximum Expansion Rate** setting is available only when **Constant First Element Offset** is selected.

Selecting **Target Maximum Expansion Rate** enables the specification of a target maximum expansion rate. TurboGrid attempts to prevent the expansion rate (at any place around the blade profile) from exceeding the specified maximum in different ways, depending on the **Boundary Layer Refinement Control > Method** option:

- Proportional to Mesh Size

The number of elements across the boundary layer is fixed. TurboGrid may increase the first element offset in order to reduce the maximum expansion rate as appropriate.

- First Element Offset

The first element offset is fixed. TurboGrid may increase the number of elements across the boundary layer in order to reduce the maximum expansion rate as appropriate.

---

#### Note:

**Target Maximum Expansion Rate** is available only when **Constant First Element Offset** is selected.

---

If reducing the maximum expansion rate to the specified maximum results in a sub-unity expansion rate (that is, a contraction rate) elsewhere, specifically such that the expansion rate is, at some place, less than the inverse of the specified maximum, then TurboGrid may balance the amounts by which:

- The maximum expansion rate is above the maximum
- The minimum expansion rate is below the inverse of the maximum

If the balance involves exceeding the rate by more than 15%, the problem will be indicated in the **Boundary Layer Control** object (which is under the **Mesh Data** object in the object selector).

#### 10.5.1.1.3.5. Near Wall Element Size Specification

The **Near Wall Element Size Specification** setting controls the method by which the near-wall node spacing is specified on the **Passage** and **Hub/Shroud Tip** tabs. The near-wall node spacing

is the distance between a wall (for example, hub, shroud, or blade) and the first layer of nodes from the wall.

The available **Method** options for calculating the near wall spacing are:

- `y+`

For details, see [Y Plus \(p. 172\)](#).

- `Absolute`

For details, see [Absolute \(p. 172\)](#).

#### 10.5.1.1.3.5.1. Y Plus

The `y+` method allows you to set the near wall spacing,  $\Delta y$ , in accordance with a target value of  $y^+$ . The target value of  $y^+$  may then be specified on the **Passage** and **Hub/Shroud Tip** tabs, as applicable (that is, when a near wall size is required by the specified distribution method).

The following formula relates the near wall spacing to  $y^+$ :

$$\Delta y = L \Delta y^+ \sqrt{80} Re_x^{1/4} \frac{1}{Re_L} \quad (10.1)$$

where  $L$  is the blade chord,  $\Delta y^+$  is the specified target  $y^+$  value,  $Re_x$  is the Reynolds number based on the distance along the chord (measured from the leading edge), and  $Re_L$  is the Reynolds number based on chord length. Ansys TurboGrid approximates  $L$  as the algebraic average of the chord lengths of each blade profile in the blade file. You must specify  $Re_L$ . Ansys TurboGrid approximates  $Re_x$  as being equal to the specified value of  $Re_L$ .

---

#### Note:

If you specify a near wall size (on the **Passage** or **Hub/Shroud Tip** tab) when the near wall method is not `y+`, you can switch the method to `y+` (at least temporarily) to see the estimated value of  $y^+$  as a setting on the **Passage** or **Hub/Shroud Tip** tab.

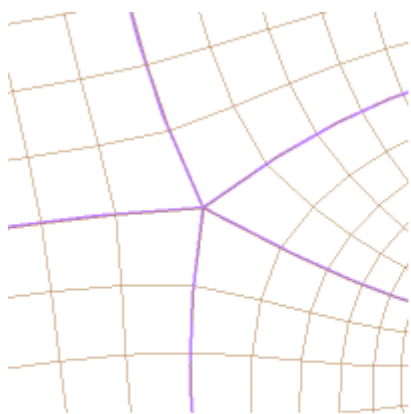
---

#### 10.5.1.1.3.5.2. Absolute

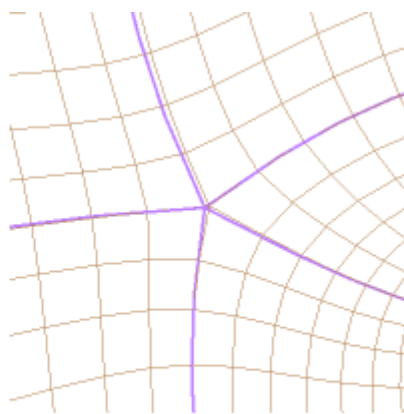
The `Absolute` method allows you to set the near wall spacing directly. Such a specification (that is, on the **Passage** or **Hub/Shroud Tip** tab) requires a dimensional value with units of distance.

#### 10.5.1.1.4. Five-Edge Vertex Mesh Size Reduction

An ATM-based mesh may contain one or more vertices that each join exactly five edges. At these locations, the mesh generation process will tend to pull the mesh away from the five-edge vertex to make the surrounding element edges more orthogonal, resulting in relatively larger elements surrounding the vertex than the rest of the mesh in their vicinity. The **Five-Edge Vertex Mesh Size Reduction > Factor** setting, when set to a value less than 1.0, will reduce this tendency, trading off between mesh orthogonality for sizing.



Factor = 1.0



Factor = 0.5

### 10.5.1.1.5. Inlet Domain and Outlet Domain Check Boxes

The **Inlet Domain** and **Outlet domain** check boxes determine whether or not the inlet and outlet domains are to be generated as part of the mesh. Settings that affect these grid regions are found on the **Inlet/Outlet** tab.

### 10.5.1.2. Passage Tab

The **Passage** tab is applicable to the Mesh Data object, and is used to specify distribution settings. Distribution settings are described generically in [Distribution Settings in General](#) (p. 176).

#### 10.5.1.2.1. Spanwise Blade Distribution Parameters

Use the **Spanwise Blade Distribution Parameters** section to control the distribution of mesh elements in the spanwise direction along the blade.

The **Method** setting and its associated settings are described in [Distribution Settings in General](#) (p. 176).

### 10.5.1.3. Hub Tip/Complex Blade End and Shroud Tip/Complex Blade End Tabs

The **Hub Tip** and **Shroud Tip** tabs of the Mesh Data object are available when their respective geometry objects (**Hub Tip** and **Shroud Tip**) exist. For details on defining those geometry objects, see [The Hub Tip and Shroud Tip Objects](#) (p. 141).

If you have defined any complex blade ends (see [Meshing High-Fidelity Geometry](#) (p. 219)), the respective tabs of the Mesh Data object are renamed; if there is a complex blade end at the hub, the **Hub Tip** tab becomes the **Hub Complex Blade End** tab; if there is a complex blade end at the shroud, the **Shroud Tip** tab becomes the **Shroud Complex Blade End** tab. The available settings vary by tab.

Setting descriptions follow:

10.5.1.3.1. [Hub Tip Distribution Parameters and Shroud Tip Distribution Parameters](#)

10.5.1.3.2. [Blade Tip Settings](#)

10.5.1.3.3. [Spanwise Distribution Parameters \(for Complex Blade Ends\)](#)

### 10.5.1.3.1. Hub Tip Distribution Parameters and Shroud Tip Distribution Parameters

Use these settings to control the distribution of mesh elements in the spanwise direction across the tip gap.

Set **Method** to one of the following:

- Match Expansion at Blade Tip
- Element Count and Size
- Uniform

These methods are described in [Distribution Settings in General](#) (p. 176).

### 10.5.1.3.2. Blade Tip Settings

The settings under **Blade Tip** (for cases with a single blade per blade set) or **Apply Blade Tip Parameters To All Blades** (for cases with multiple blades per blade set) control details of the mesh elements on the blade tip (and throughout the blade tip region of the mesh).

The settings that are available depend on the type of leading edge and the type of trailing edge:

[10.5.1.3.2.1. Blades with two rounded edges](#)

[10.5.1.3.2.2. Blades with one rounded edge and one cut-off edge](#)

[10.5.1.3.2.3. Blades with two cut-off edges](#)

#### 10.5.1.3.2.1. Blades with two rounded edges

The first element height in the tip mesh is matched with the first element height in the boundary layer region (outside the blade). The match may not be exact in rounded leading edge or trailing edge regions.

In the tip mesh, the mesh elements change height across the blade thickness according to locally applied near-wall expansion rates. These expansion rates are limited at the thickest part of the blade by a target maximum rate, which is determined by the first applicable method of the following methods:

- If you select **Override Target Maximum Expansion Rate**, the corresponding specified rate is used.
- If you select **Target Maximum Expansion Rate** (on the **Mesh Size** tab), the corresponding specified rate is used.
- The maximum achieved expansion rate in the boundary layer region (outside the blade) is used.

The expansion rate influences the number of mesh elements across the blade tip.

After the topology has been generated, TurboGrid reports the minimum and maximum achieved expansion rates, and the number of mesh elements across the blade tip, in the **Hub Tip** and **Shroud Tip** tabs.

### 10.5.1.3.2.2. Blades with one rounded edge and one cut-off edge

The distribution of elements across the blade tip mesh is governed by the distribution of elements across the cut-off edge. The latter is controlled indirectly by the **Cutoff Edge Split Factor** setting (on the **Mesh Size** tab) which controls the number of elements across the cut-off edge. For details, see [Cutoff Edge Split Factor](#) (p. 171).

### 10.5.1.3.2.3. Blades with two cut-off edges

The **Tip Centerline Location** setting is available when the following conditions are met:

- The corresponding blade tip, hub or shroud, exists.
- The tip topology is set to `H-Grid Not Matching`.
- The corresponding edge of the blade, leading or trailing, is cut-off.

The setting controls how the blade centerline behaves at the cut-off end. The options are:

- `Automatic`

Effectively selects one of the other two methods automatically, based on the geometry. If one of the corners of the cut-off edge is at a sufficiently acute angle (default is 45°), then the `Corner` option is selected, otherwise the `Middle` option is selected. The threshold angle can be set by defining the parameter

`GGI Tip Angle To Switch Mean Line Into Cut Off Corner` (to an angular value; default unit is [degree]) in the CCL for the blade-specific mesh data object.

- `Middle`

The centerline meets the cut-off blade edge in the middle (between the corners).

- `Corner`

The centerline meets the corner of the cut-off blade edge.

### 10.5.1.3.3. Spanwise Distribution Parameters (for Complex Blade Ends)

Use these settings to control the distribution of mesh elements near the complex blade end.

The **Spanwise Element Count** settings affect the number of hex elements placed spanwise across the complex blade end clearance gap.

**Method** can be set to:

- `Element Count`

Set **Number of Edge Splits** to the number of hex elements to place spanwise across the complex blade end clearance gap

- `Factor` (the default for new cases)

Set **Element Count Factor** to a value that multiplies with a computed suggested number of edge splits to yield the target number of edge splits.

The **Interface Element Size** settings control the element size over the interface between the structured hex mesh of the passage and the unstructured mesh block for the complex blade end.

**Method** can be set to:

- Size

Set **Normalized Interface Element Size** to the target element size over the interface between the structured hex mesh of the passage and the unstructured mesh block for the complex blade end.

- Size Factor (the default for new cases)

Set **Element Size Factor** to a value that multiplies with a computed suggested normalized interface element size to yield the normalized interface element size.

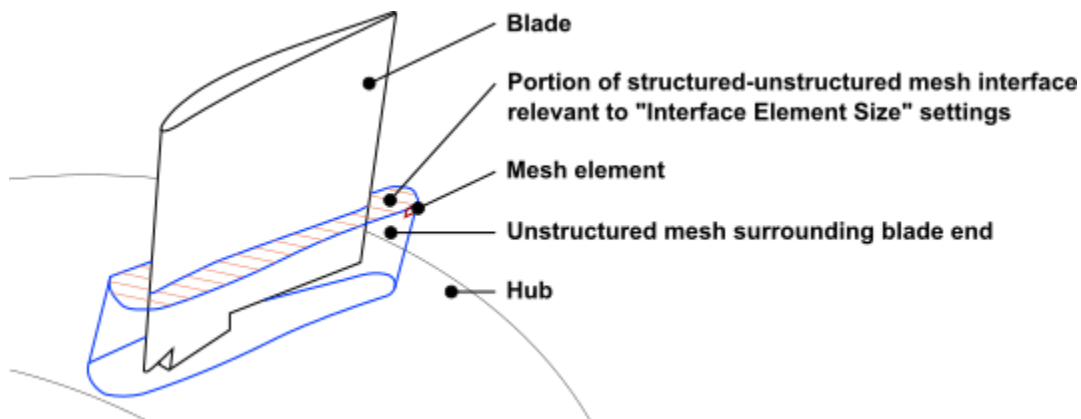
The default interface element size aims to make isotropic the mesh element in the position shown in the illustration below.

---

**Note:**

If the span varies significantly from leading to trailing edge, generation of the unstructured mesh might benefit from changes to:

- The interface element size, and
- The interface span position, which is controlled by the **Span Location** setting on the **Complex Blade End** tab for the applicable complex blade end Mesh Data object. For details, see [Setting Descriptions for Complex Blade End Mesh Data Objects](#) (p. 225).



#### 10.5.1.4. Distribution Settings in General

The distribution of elements along the blade (in the spanwise direction) is controlled by one of these methods:

- Element Count and Size

To use the `Element Count and Size` method, specify the number of elements that span the pertinent extent and whether the distribution is uniform or non-uniform. If the distribution is non-uniform, you must also specify the number of uniformly-distributed elements (**Const Elements**) and the element size next to the pertinent wall(s).

- `Boundary Layer`

To use the `Boundary Layer` method (not applicable for O-Grid settings), specify distribution parameters for three sections: boundary layer at the hub, boundary layer at the tip/shroud, and the section in between. **Layer Offset** means the thickness of the boundary layer and **Wall Offset** means the thickness of the first element next to the wall.

- `Expansion Rate`

To use the `Expansion Rate` method, specify an expansion rate and the element size next to the blade wall.

- `Uniform`

To use the `Uniform` method, specify the number of elements. Each element is the same size.

- `Proportional` (the only option when using a meridional splitter)

This is the default method when ATM topology is used. With this method, Ansys TurboGrid automatically computes the number and distribution of elements in the spanwise direction so that the near wall element heights on the hub and shroud match the maximum near wall element height on the blade. Ansys TurboGrid adjusts the spanwise node count in proportion to the blade aspect ratio, the square root of the streamwise node count, and the proportional **Factor**. Increasing the proportional **Factor** increases the mesh resolution in the spanwise direction.

When there is a meridional splitter, the **Passage Boundary Layer Refinement Control > Factor** setting is available, and scales the near wall element heights on the hub, meridional splitter, and shroud surfaces.

---

### Note:

The **Const Elements** value cannot be negative and must be at least 2 less than the **# of Elements** value for that region.

---

The distribution of elements across a tip gap (in the spanwise direction) is controlled by one of these methods:

- `Match Expansion at Blade Tip`

The `Match Expansion at Blade Tip` method is default. This method will apply the same expansion factor as that used along the blade span, and will match the size of the elements on each side of the tip clearance gap with the size of the first element along the blade. An appropriate number of elements is then derived.

- `Element Count and Size`

(described above)

- Uniform

(described above)

When using the `End Ratio` method, the element count and wall size(s) are updated accordingly in the **Element Count and Size** settings. (When two wall sizes are adjusted, they are adjusted to the same value.) When using the `Element Count and Size` method, the **End Ratio** setting is updated accordingly.

In the case of an O-Grid distribution, the **Blade End Ratio** is the size ratio of the passage mesh element nearest the O-Grid to the O-Grid element nearest the blade surface. A special feature of the distribution settings for O-Grids is that, if **# of Elements** is set to 0, the number of elements will be calculated automatically so that the best possible matching of element sizes at the interface between the O-Grid and the passage is ensured.

**Size of Elements Next to Wall (Normalized)** is normalized based on the total distance along the blade (not counting any tip clearance gaps). For example, on the **Hub Tip** tab, a value of 0.05 for **Size of Elements Next to Wall (Normalized) > Hub** represents an element size of five percent of the distance along the blade, applied on the elements next to the hub, and a value of 0.05 for the **Size of Elements Next to Wall (Normalized) > Tip** represents an element size of five percent of the distance along the blade, applied on the elements next to the hub end of the blade (both in the tip clearance gap and on the blade).

---

**Note:**

The **Size of Elements Next to Wall (Normalized)** value must be less than 1 (unity) divided by the number of elements for that region. If it is not, Ansys TurboGrid decreases it to meet this criterion.

To increase the quality of the mesh, try to minimize drastic changes in element size. Wherever possible, attempt to have gradual increases and decreases in element size in all directions.

---

### 10.5.1.5. Inlet and Outlet Tabs

The geometries of the inlet and outlet domains of the mesh are controlled by the hub and shroud curves, and the `Inlet` and `Outlet` geometry objects. For details, see [The Hub and Shroud Objects \(p. 120\)](#) and [The Inlet and Outlet Objects \(p. 143\)](#).

The **Inlet/Outlet** tab contains settings, as applicable, that affect the mesh in the inlet and outlet domains. The **Inlet Domain** and **Outlet domain** check boxes on the **Mesh Size** tab of the `Mesh Data` object determine whether or not the inlet and outlet domains are to be generated as part of the mesh.

The nodes of the inlet and outlet domains match one-to-one with the nodes of the passage domain where they meet at the interfaces. The rest of the mesh in the inlet or outlet domain is then as near to being isotropic and orthogonal as possible.

#### 10.5.1.5.1. Inlet Domain and Outlet Domain Settings

**Mesh Type** can be set to one of the following options:

- H-Grid (default)

If you have any hub and/or shroud regions defined (see [Hub Regions and Shroud Regions Tab \(p. 122\)](#)), and the applicable domain (**Inlet Domain** or **Outlet Domain**) is selected on the **Mesh Size** tab, then setting **Mesh Type** to H-Grid makes available the **Limit Aspect Ratio** and **Inlet Multi Segment Enabled/Outlet Multi Segment Enabled** settings (described below).

- H-Grid in Parametric Space

In this case, the resulting mesh tries to follow a parametric space created by an elliptic smoothing method. This type of mesh is particularly suitable for return channels.

Both options specify an H-Grid type mesh in the inlet/outlet domain that preserves the boundary layer mesh resolution at the hub and shroud and automatically matches the element size at the interface between the inlet/outlet domain and passage domain.

The streamwise distribution of elements is non-uniform and uses a constant expansion rate.

**Define By** can be set to one of the following options:

- Target Expansion Rate (default)

In this case, the number of elements in the inlet/outlet domain is determined by matching the element size at the passage interface and expanding the size at a prescribed rate for each successive element. The default expansion rate (element size ratio) is 1.2


- Target Number of Elements

The **# of Elements** setting controls the target number of elements in the inlet/outlet domain in the streamwise direction. The expansion rate is determined by the target number of elements, the streamwise length of the inlet/outlet domain, and the passage element size.

---

### Important:

Clicking the **Apply** button does not create the mesh, it only saves the Mesh Data object and updates the 2D mesh previews. To create the mesh, select **Insert > Mesh**

from the main menu or click *Create Mesh* .

---

If you have any hub and/or shroud regions defined (see [Hub Regions and Shroud Regions Tab \(p. 122\)](#)), and the applicable domain (**Inlet Domain** or **Outlet Domain**) is selected on the **Mesh Size** tab, and **Mesh Type** is set to H-Grid, then the **Limit Aspect Ratio** option is available. The **Limit Aspect Ratio** option enables you to specify a maximum target aspect ratio for mesh elements in the inlet/outlet domain by setting a value for **Request Max AR** value. TurboGrid may increase the element count in the streamwise direction in an attempt to reduce the maximum aspect ratio to (or below) the target value. The existing maximum aspect ratio is shown in the object editor beside the label **Current Max AR**. Note that the number of elements in the spanwise direction is controlled by the passage mesh.

If you have any hub and/or shroud regions defined (see [Hub Regions and Shroud Regions Tab \(p. 122\)](#)), and the applicable domain (**Inlet Domain** or **Outlet Domain**) is selected on the **Mesh Size** tab, and **Mesh Type** is set to H-Grid, then the **Inlet Multi Segment Enabled/Outlet**

**Multi Segment Enabled** option is available, and is selected by default. When this option is selected:

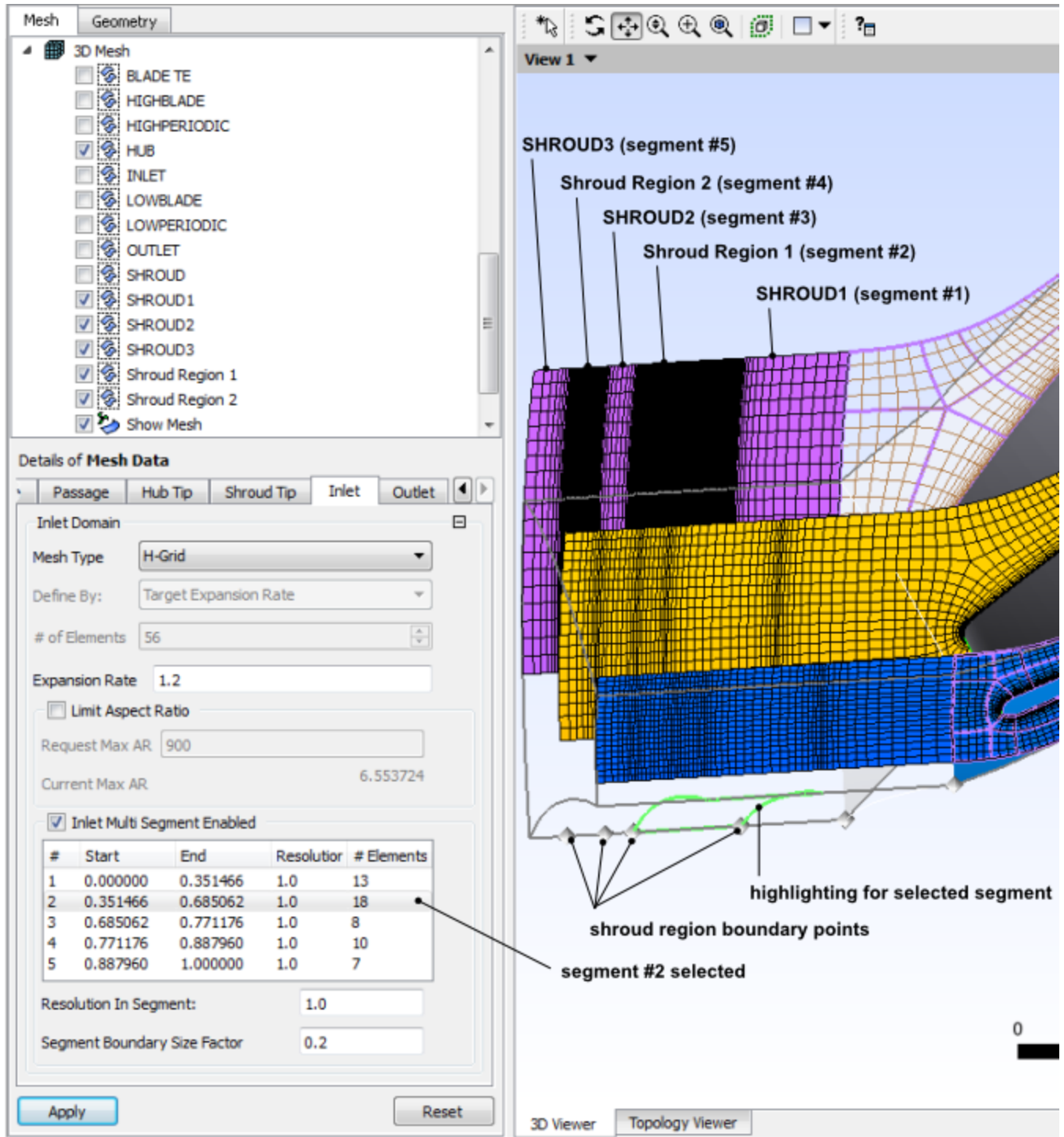
- Any defined hub/shroud regions are enabled (affecting the mesh and surface regions in the output), and
- Settings for controlling the mesh distribution within the hub/shroud regions become available.

If you select a segment (a defined hub/shroud region or a numbered surface region that is beside a defined hub/shroud region) in the list, the segment is highlighted in the **3D Viewer** and you can then change the resolution via the **Resolution in Segment** setting. The specified value directly scales the number of elements (**# Elements**) along the segment (in the streamwise direction). Note that the minimum number of elements along any segment is 4.

The **Segment Boundary Size Factor** setting directly scales the element thickness (as measured in the streamwise direction) on all segment boundaries except the outermost segment boundaries. Note that this setting indirectly affects the number of elements along each and every segment.

Figure 10.16: [Inlet Domain Segments \(p. 181\)](#) shows an example of a 3D Mesh object with 2D regions that correspond with the segments inside the inlet domain.

Figure 10.16: Inlet Domain Segments



In this example, there are:

- Two defined shroud regions in the inlet domain: "Shroud Region 1" and "Shroud Region 2". These regions can be renamed and otherwise redefined in the Shroud geometry object settings.
- Three other regions on the shroud in the inlet domain: "SHROUD1", "SHROUD2", and "SHROUD3". These regions are automatically generated in order to cover the rest of the shroud within the inlet domain.

- No defined hub regions. (Hub regions could potentially be added.)

### 10.5.1.6. Mesh Around Blade Tab

The **Mesh Around Blade** tab is applicable to the blade-specific mesh data objects.

The **Size of Elements Next to Wall (Normalized)** setting is described in [Distribution Settings in General](#) (p. 176).

The **Hub Tip**, and **Shroud Tip** settings are for controlling the number of elements across the hub tip gap or shroud tip gap. They are similar to the settings on the **Hub Tip** and **Shroud Tip** tabs for the `Mesh Data` object. For details, see [Hub Tip/Complex Blade End and Shroud Tip/Complex Blade End Tabs](#) (p. 173).

#### 10.5.1.6.1. Tip Centerline Location

For details, see [Blades with two cut-off edges](#) (p. 175).

### 10.5.1.7. Complex Blade End Mesh Data Settings

See [Setting Descriptions for Complex Blade End Mesh Data Objects](#) (p. 225).

## 10.5.2. Changing the Number of Elements on a Selected Master Topology Edge

You can control the number of elements along any master topology edge that is outside of the boundary layer region by right-clicking the edge in the viewer and selecting one of the following commands:

- **Insert Edge Split Control**

This option inserts an `Edge Split` object under the `Mesh Data` object, provided that the applicable master topology edge is outside, or along the outer perimeter of, the boundary layer region. You can then edit the `Edge Split` object to adjust the edge split factor.

---

**Note:**

`Edge Split` objects are not available when using the ATM3D meshing approach.

---

- **Increase Edge Refinement** and **Decrease Edge Refinement**

These options enable you to choose a preset amount by which to increase or decrease the edge split factor. An `Edge Split` object is automatically added for the selected edge if one has not been created already.

The `Edge Split` object displays the base and actual block edge split counts for the applicable master topology edge. You can remove the `Edge Split` object by right-clicking it in the object selector and selecting **Delete**.


## 10.6. Layers

A layer shows the topology projected onto a given span. By default, and as a minimum, there are two layers present: one at the hub and one at the shroud. In many cases, additional layers are required to improve mesh quality. The addition of layers improves the 3D mesh by adapting the topology to the local geometry before mesh generation. Creating (and adjusting if necessary) additional layers enhances the quality of the mesh by creating a curve for the mesh to follow between the hub and the shroud. The more complex the blade shape-change from hub to shroud, the more layers are required.

### 10.6.1. Adding Layers

Layers can be added from the editor for the `Layers` object (see [The Layers Object \(p. 184\)](#)), or by right-clicking a layer object in the selector, and selecting the appropriate command from the **Insert** submenu of the shortcut menu. Layers can also be added automatically at the time of mesh creation (default), depending on a setting found in the editor for the `Layers` object.

### 10.6.2. Deleting Layers

The object selector and the editor for the `Layers` object ([The Layers Object \(p. 184\)](#)) each have an icon to delete layers () as well as a shortcut menu option for deleting layers.

### 10.6.3. Editing the Settings of Layers

The editor for the `Layers` object ([The Layers Object \(p. 184\)](#)) and the editor for layer objects ([Layer Objects \(p. 187\)](#)) are used to control various properties of layers.

### 10.6.4. Layer Visibility

There are visibility check boxes next to each layer listed in the object selector. By right-clicking a layer object, a shortcut menu will appear, allowing the following visibility options as applicable:

- Show  
Makes the layer visible.
- Show + Hide All Siblings  
Makes the layer visible and turns off the visibility of all other layers.
- Hide  
Turns off the visibility of the layer.

The visibility of a layer can also be controlled by the visibility settings in the individual layer objects.

To change which parts of a layer are visible, select the layer(s) from the object selector, then right-click the selection and select an appropriate rendering option from the shortcut menu. For details, see [Master Topology Visibility \(p. 187\)](#), [Topology Visibility \(p. 187\)](#), and [Refined Mesh Visibility \(p. 187\)](#).

## 10.6.5. The Layers Object

The `Layers` object is used to control the individual layer objects. You can access the `Layers` objects in the object selector.

### 10.6.5.1. Layers Tab

A number of possible operations are available for a layer by clicking it in the list box and then selecting one of the icons on the right of the list box, or by right-clicking the object and then selecting an operation from the shortcut menu.

#### 10.6.5.1.1. Defining Intermediate Layers

To ensure that the mesh is adequately guided in the spanwise direction, a mesh might require more layers than just a Hub layer and a Shroud layer. You can set up intermediate layers in different ways, as controlled by the **Insertion Mode** option:

- `Manual - Uniform`

TurboGrid creates the specified number (**Count**) of layers and positions them uniformly in the spanwise direction from hub to shroud.

Existing layers are not preserved when you apply the `Manual Uniform` option.

- `Automatic - Adaptive` (default for the ATM meshing approach; unavailable for the ATM3D meshing approach)

The `Automatic - Adaptive` option causes TurboGrid to automatically add the "recommended number of additional Layers" (as displayed in the **Layers** object editor) upon topology generation. The recommended number might be zero, in which case no new layers are added. Whenever new layers are added, a notification appears in the status bar in the lower-left corner. In this mode you have no control over the number of intermediate layers or the positions of the intermediate layers. Switch to the `Manual` option (described below) if more control is needed.

---

**Note:**

The recommended number of layers is a recommendation based on blade shape, size of the topology around the blade, proximity of inlet/outlet, and so on. As such the recommendation is not always reliable.


---


- `Manual` (unavailable for the ATM3D meshing approach)

You can control the number and positioning of intermediate layers.

Existing layers are preserved when you switch to the `Manual` option. This enables you to, for example, make adjustments after using the `Manual - Uniform` option.

You can add one layer at a time either:

- wherever TurboGrid chooses (right-click any Layer in the list and select **New Layer** or click *New Layer* ) , or
- "after" (on the Shroud side of) any particular existing layer (right-click a layer in the list and select **Insert Layer After**).

You can add the recommended number of layers all at once, letting TurboGrid choose the layer positions (right-click a layer in the list and select **Auto Add Layers** or click *Auto Add Layers* ) .

You can select one or (using **Shift** or **Ctrl**) more layers, then delete them (right-click the layer or selection of layers and select **Delete Selected Layer(s)** or click **Delete Selected Layer(s)** ) .

---

**Note:**

When the `Manual` option is applied, new layers are not generated upon topology generation or 3D mesh generation. If you want TurboGrid to automatically add its "recommended number of additional Layers" (as displayed in the **Layers** object editor) upon topology generation or 3D mesh generation, switch to the `Automatic - Adaptive` option.

---


#### 10.6.5.1.2. Count

(applies when **Insertion Mode** is set to `Manual - Uniform`)

The number of intermediate layers to be generated and placed uniformly in the spanwise direction between the Hub and Shroud layers is specified in the **Count** box.


#### 10.6.5.1.3. New Layer

(applies when **Insertion Mode** is not set to `Manual - Uniform`)

Clicking *New Layer*  (or right-clicking a layer and selecting **New Layer**) creates a new layer at the most suitable span location as determined by TurboGrid.

#### 10.6.5.1.4. Delete Selected Layers


(applies when **Insertion Mode** is not set to `Manual - Uniform`)

Clicking *Delete Selected Layer(s)*  removes the selected layer(s). Hold **Shift** or **Ctrl** to select multiple layers to delete. You cannot delete the hub or the shroud layers.

#### 10.6.5.1.5. Inserting Layers Automatically

(applies when **Insertion Mode** is not set to `Manual - Uniform`)

Selecting **Insert > Layer Automatically** from the shortcut menu for any layer object will cause Ansys TurboGrid to insert one layer automatically using built-in heuristics to determine an appropriate location. The position of the inserted layer does not depend on which layer object you right-click to access the shortcut menu.

Clicking *Auto Add Layers*  in the editor for the `Layers` object, or selecting **Insert > Layers Automatically** from the shortcut menu for a layer object, causes Ansys TurboGrid to insert layers automatically, using built-in heuristics to determine an appropriate number of layers to insert, and the location of each layer. The number of layers that Ansys TurboGrid proposes to add is shown in the editor for the `Layers` object.

To have layers inserted automatically upon topology generation or 3D mesh generation, set **Insertion Mode** to `Automatic - Adaptive`.

#### 10.6.5.1.6. Inserting Layer After Selected Layer

(applies when **Insertion Mode** is not set to `Manual - Uniform`)

The **Insert Layer After** and **Insert > Layer After** shortcut menu items cause a new layer to be created at the span halfway between the selected layer and the next layer. For example, if the only existing layers are on the hub and the shroud, and **Insert > Layer After** is selected from the hub, Ansys TurboGrid inserts the new layer at a span of 0.5.

#### 10.6.5.1.7. Span Location

(applies when **Insertion Mode** is not set to `Manual - Uniform`)

The span location of a layer can be modified in the **Span Location** box.

#### 10.6.5.1.8. Spanwise Mesh Interpolation Guide Curves

Topological guide curves are constructed in the spanwise direction. Such curves pass through corresponding topology points on each layer. The curves can be constructed using line segments (piece-wise linear) or curves (Bspline curves). The **Spanwise Mesh Interpolation Guide Curves > Type** setting, which is available when not using ATM3D, controls the type of curve, and has the following options:

- `Automatic`

Chooses one of the other options automatically. TurboGrid chooses `Bspline` unless either of the following conditions are met, in which case it chooses `Piece-wise linear`:

- The blade is ruled in the spanwise direction.
- In the `Inlet/Outlet` object, **Interface Specification Method > Points** is selected and more than two inlet/outlet points are defined in the list of points.

- `Piece-wise linear`
- `Bspline`

---

This option may produce adverse effects in cases with many layers.

---

**Note:**

If you have negative mesh element volumes near the leading or trailing edge of the blade, switching the curve type might correct the problem. If the problem is not corrected by switching the curve type then it is recommended that you add more layers and use the `Piece-wise linear` option.

---

## 10.6.6. Layer Objects

You can access the layer objects under the `Layers` object in the object selector.

### 10.6.6.1. Data Tab

#### 10.6.6.1.1. Master Topology Visibility

Clicking in the box will toggle the visibility of the master topology. The master topology is shown as violet line segments.

You can also change the **Master Topology Visibility** setting from the shortcut menu after selecting and right-clicking the layer(s) in the object selector, or right-clicking a layer in the viewer.

#### 10.6.6.1.2. Topology Visibility

Clicking in the box will toggle the visibility of the topology. The topology is shown as yellow line segments.

You can also change the **Topology Visibility** setting from the shortcut menu after selecting and right-clicking the layer(s) in the object selector, or right-clicking a layer in the viewer.

#### 10.6.6.1.3. Refined Mesh Visibility

Selecting **Refined Mesh Visibility** causes the refined mesh to appear in the viewer. On the first and last layers, the refined mesh shows the 3D mesh. On intermediate layers, the refined mesh shows an approximation of the 3D mesh.

You can also change the **Refined Mesh Visibility** setting from the shortcut menu after selecting and right-clicking the layer(s) in the object selector, or right-clicking a layer in the viewer.

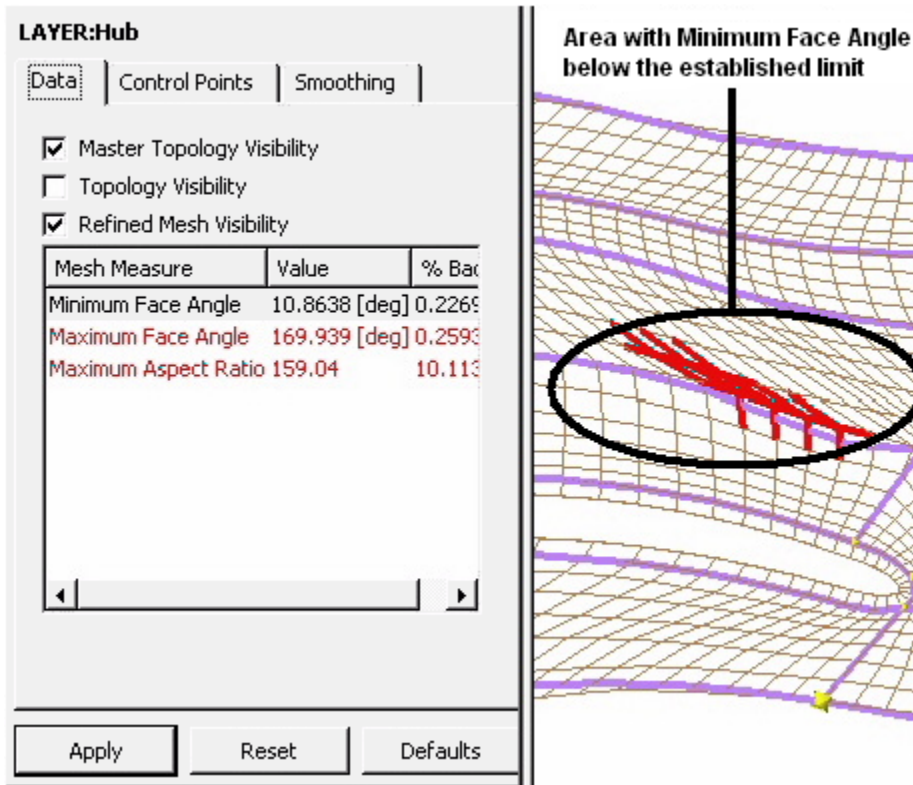
When **Refined Mesh Visibility** is selected, the following refined-mesh quality statistics—Mesh Measure—are available for viewing:

- Minimum Face Angle
- Maximum Face Angle
- Maximum Aspect Ratio

To display problem areas in the refined mesh, double-click one of the **Mesh Measure** statistics in the list. [Figure 10.17: Refined Mesh Showing Areas of Unacceptable Minimum Face Angle \(p. 188\)](#)

shows an example of problem areas, shown with thick red lines, for the `Minimum Face Angle` statistic.

**Figure 10.17: Refined Mesh Showing Areas of Unacceptable Minimum Face Angle**



The `Mesh Limits` object holds the criteria that determine if the **Mesh Measure** statistics are acceptable.

If, for a given layer, **Refined Mesh Visibility** is turned on and either the `Minimum Face Angle` or `Maximum Face Angle` is outside the applicable limit, the layer will be listed in red text in the selector tree.

## 10.7.3D Mesh

The 3D mesh generated by Ansys TurboGrid has objects associated with it. These objects are useful for generating and examining the mesh.


### 10.7.1. The 3D Mesh Object

You can open the **3D Mesh** object editor to see node and element counts for the passage, inlet, outlet, and the whole mesh.

By default, the `3D Mesh` object is initially unsuspending, so mesh generation happens automatically whenever the `Layers` object is (re)processed.

To reduce the update delay for large meshes, you can suspend the `3D Mesh` object (right-click it in the **Mesh** workspace tree and select **Suspend Object Updates**). By suspending the `3D Mesh` object, you prevent it from being updated in response to changes to other objects.

If the 3D Mesh object is suspended, you can generate the 3D mesh in any of the following ways:

- Unsuspend the 3D Mesh object by right-clicking it in the **Mesh** workspace tree and clearing **Suspend Object Updates**.
- Click **Insert > Mesh** from the main menu.
- Click the Mesh  icon.
- Click the **Generate** button in the **3D Mesh** object editor.

The **Generate** button is enabled whenever it is possible to generate (or refresh) a 3D mesh.

---

**Note:**

The **Undo** command cannot properly undo a suspension change (for example, to the Topology Set object); attempting to undo a suspension change can lead to an inconsistent state.

---

## 10.7.2. Surface Group and Turbo Surface Objects

The 3D Mesh branch contains objects that display the mesh on geometry surfaces and turbo surfaces. Of these display objects, only the turbo surface object Show Mesh is visible by default. To view another surface, select the visibility check box for that object.

The mesh can be displayed only after it has been created. Changes made to the Geometry, Topology, or Mesh Data objects will cause the mesh, and the surface objects found under the 3D Mesh object, to be deleted.

### 10.7.2.1. 3D Mesh Turbo Surfaces

The turbo surface is a flexible object that allows you to closely examine every area of the mesh. One turbo surface exists in the 3D Mesh branch of the object selector after you create a mesh. You can create other turbo surfaces. For details, see [Turbo Surface Command \(p. 65\)](#).

### 10.7.2.2. 3D Mesh Surface Groups

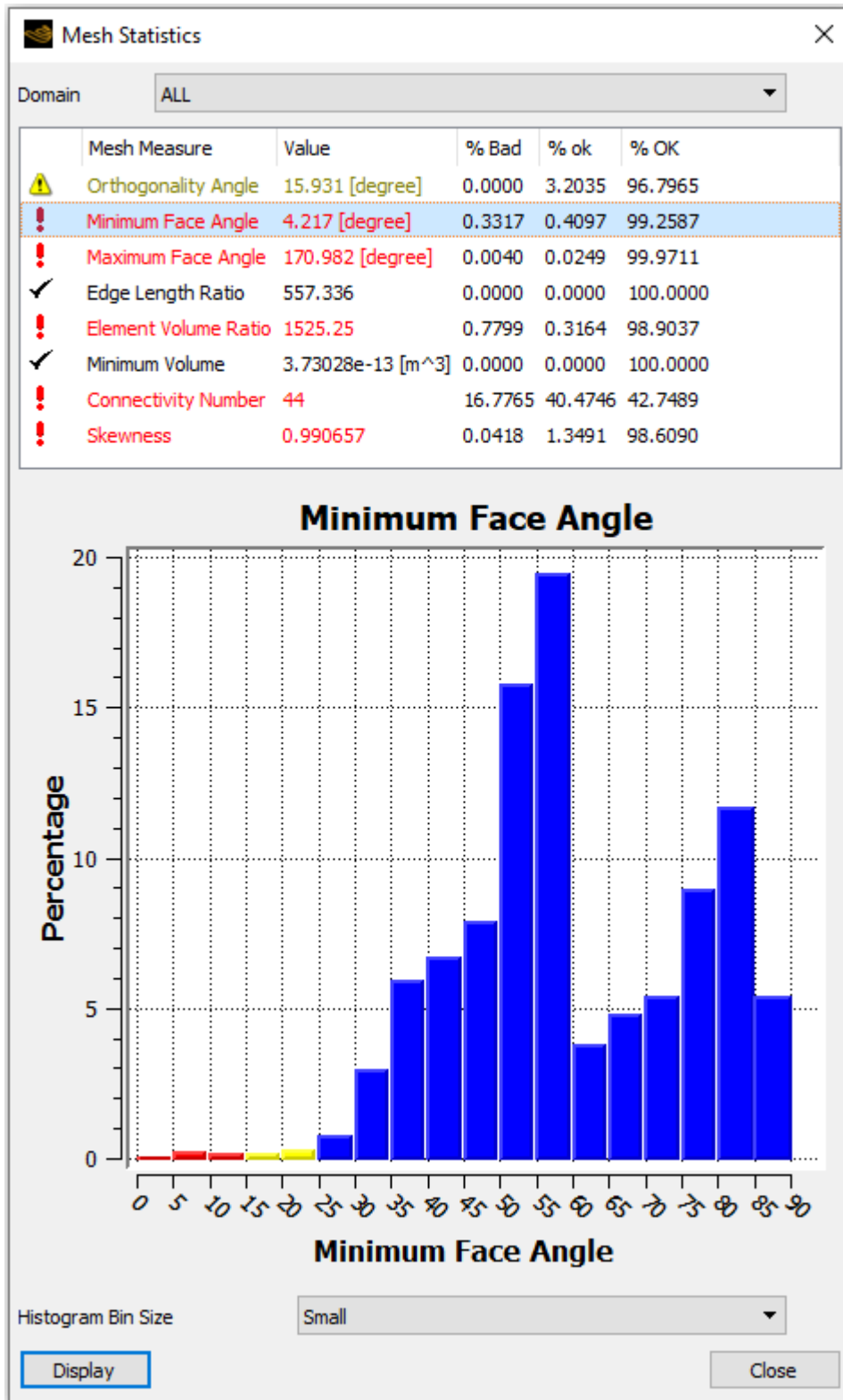
A collection of surface groups exists in the 3D Mesh branch of the object selector after you create a mesh. For details, see [Surface Groups \(p. 70\)](#).

## 10.8. Mesh Analysis

---

The Mesh Analysis branch offers a variety of tools to analyze the quality of the mesh. The mesh can be analyzed only after it has been created. If changes are made to the Geometry, Topology, Layers, or Mesh Data objects, the mesh cannot be analyzed again until it has been recreated. For details, see [Mesh Command \(p. 59\)](#).

### 10.8.1. Mesh Statistics



The Mesh Statistics object provides details about the quality of the current mesh. The Mesh Statistics object cannot be viewed until the mesh is created. For details, see [Steps to Create a](#)

[Mesh \(p. 101\)](#). To view the mesh statistics, double-click the `Mesh Analysis` object or `Mesh Statistics` object in the object selector.

The `Mesh Statistics` object does not open in the object editor like the other objects in the object selector. Instead, it appears as the `Mesh Statistics` dialog box, so it can stay open while other objects are edited in the object editor. This allows for constant monitoring of the statistics.

Mesh analysis variables include `Minimum Face Angle`, `Maximum Face Angle`, and `Element Volume Ratio`. The preferences control which mesh analysis variables are available. For details, see [Mesh Quality Measures \(p. 50\)](#).

Statistics displayed in yellow or red are partially or fully outside the limits defined in the `Mesh Analysis > Mesh Limits` object. For details, see [Mesh Limits \(p. 191\)](#). To display the regions where the mesh statistics are outside the limits, select the mesh measure, then click **Display**. Alternatively, double-click the mesh analysis variable in the list.

Each calculated mesh measure is added to the list of available variables for creating new plots (for example, a contour plot).

The **Mesh Statistics** dialog box can show a histogram. To produce a histogram, select a mesh statistic from the list and click **Display**. You can control the resolution of the histogram using the **Histogram Bin Size** setting.

The rankings "Bad", "ok", and "OK", which are reflected in both the columns and histogram colors in the **Mesh Statistics** dialog box, are judged according to the criteria set in the `Mesh Limits` object, described in [Mesh Limits \(p. 191\)](#).

## 10.8.2. Mesh Limits

The `Mesh Limits` object defines the range of acceptable values for each mesh analysis variable. For each variable, the type of range depends on the **Limits Type** option:

- **Maximum**

Set **Max Value** to the high end of the range.

Set **OK Value** to the low end of the range.

Values above **Max Value** are considered to be "Bad".

Values between **Max Value** and **OK Value** are considered to be "ok" (not very good but still acceptable).

Values below **OK Value** are considered to be "OK" (very good).

- **Minimum**

Set **OK Value** to the high end of the range.

Set **Min Value** to the low end of the range.

Values above **OK Value** are considered to be "OK" (very good).

Values between **OK Value** and **Min Value** are considered to be "ok" (not very good but still acceptable).

Values below **Min Value** are considered to be "Bad".

- Range

Set **Max Value** to the high end of the range.

Set **Min Value** to the low end of the range.

Values above **Max Value** are considered to be "Bad".

Values between **Max Value** and **Min Value** are considered to be "OK" (very good).

Values below **Min Value** are considered to be "Bad".

The rankings "Bad", "ok", and "OK", are reflected in both the columns and histogram colors in the **Mesh Statistics** dialog box, described in [Mesh Statistics \(p. 190\)](#).

The default limits are intended to work for a broad variety of cases, and should highlight any problem areas of the mesh. You can change the default limits in the preferences; for details, see [Mesh Quality Measures \(p. 50\)](#).

The mesh analysis variables are described in the following sections:

[10.8.2.1. Maximum Face Angle](#)

[10.8.2.2. Minimum Face Angle](#)

[10.8.2.3. Connectivity Number](#)

[10.8.2.4. Element Volume Ratio](#)

[10.8.2.5. Minimum Volume](#)

[10.8.2.6. Edge Length Ratio](#)

[10.8.2.7. Skewness](#)

[10.8.2.8. Orthogonality Angle](#)

### **10.8.2.1. Maximum Face Angle**

This is the greatest face angle at a given node. For each face touching a given node, the angle between the two edges of the face that connect to the node is calculated. The largest of all such angles at the node is returned. The maximum face angle can be considered to be a measure of skewness.

### **10.8.2.2. Minimum Face Angle**

This is the smallest face angle at a given node.

### **10.8.2.3. Connectivity Number**

`Connectivity Number` is the number of elements that are connected to a given node. For an unstructured solver, this value is not important. However, high connectivity numbers in much of the mesh can have an adverse effect on the speed of a structured solver.

### 10.8.2.4. Element Volume Ratio

Element Volume Ratio is defined as the volume ratio of the maximum-volume element that connects to a given node to the minimum-volume element that connects to the same node.

### 10.8.2.5. Minimum Volume

Minimum Volume is used to ensure that no negative volumes are created within the passage. The value returned for a given node is the minimum volume of any element connected to that node.

### 10.8.2.6. Edge Length Ratio

For each face that is connected to a given node, the ratio

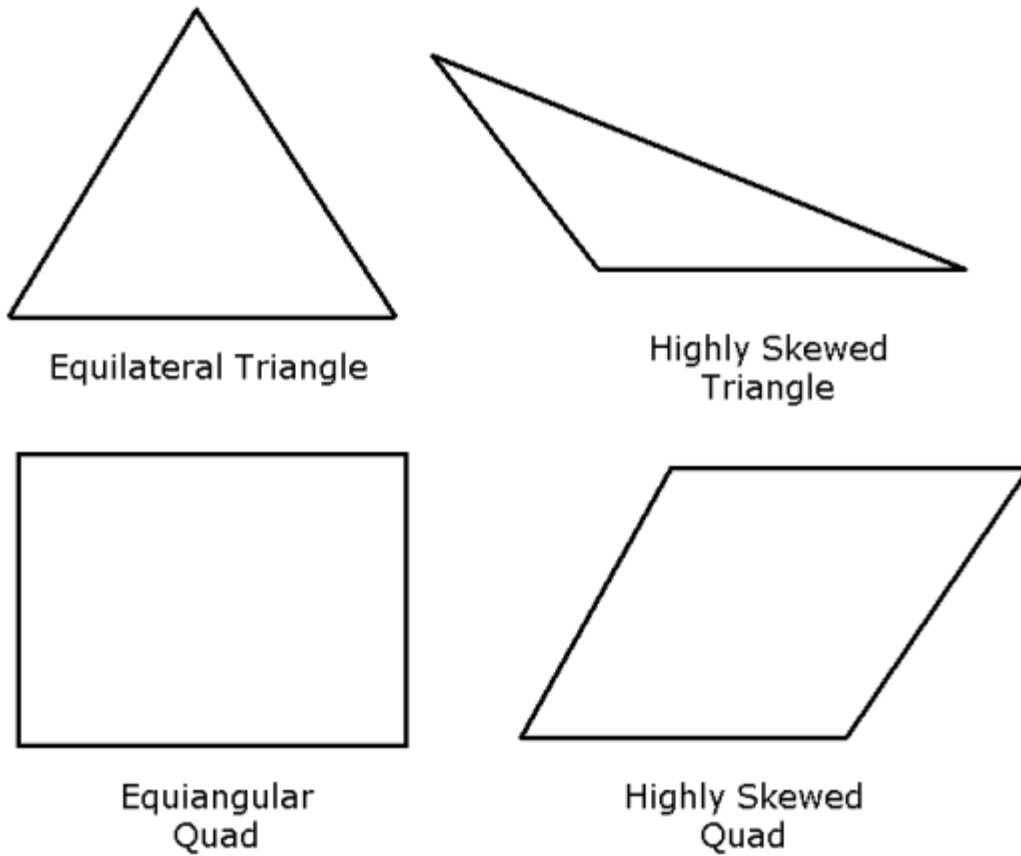
$$\frac{\max(l_1, l_2)}{\min(l_1, l_2)}$$

is calculated for the two face edges that are connected to the node. The returned value is the largest such ratio calculated for the node. The edge length ratio can be considered to be a measure of aspect ratio.

### 10.8.2.7. Skewness

Skewness is one of the primary quality measures for a mesh. Skewness determines how close to ideal (that is, equiangular) an element is in terms of its interior angles:

**Figure 10.18: Ideal and Skewed Triangles and Quadrilaterals**



The following table lists the range of skewness values and the corresponding element quality.

**Table 10.4: Skewness Ranges and Element Quality**

Skewness	Element Quality
>1	non-convex; especially problematic for FEM solvers; not necessarily problematic for finite volume solvers.
1	degenerate
0.9–<1	bad (sliver)
0.75–0.9	poor
0.5–0.75	fair
0.25–0.5	good
0–0.25	excellent
0	equiangular

According to the definition of skewness, a value of 0 indicates an equiangular element (best quality) and a value of 1 indicates a completely degenerate element. Degenerate elements (slivers) are characterized by faces that are nearly coplanar.

Highly skewed elements should be avoided because they can lead to less accurate results.

### 10.8.2.8. Orthogonality Angle

For details, see [Mesh Orthogonality in the CFX-Solver Modeling Guide](#).

### 10.8.3. Mesh Statistics Parameters - Order Of Importance

Generally, the mesh statistics can be ranked as follows (most important to least important)

1. `Minimum Volume` - these **MUST** be fixed before the mesh would be usable.
2. `Maximum Face Angle/Minimum Face Angle` - these should be improved until they fall within the constraints (minimum of 15° and maximum of 165°), if possible. Values close to, but just outside the constraints may still be acceptable for your simulation.
3. `Edge Length Ratio` - this can often be fixed by increasing the number of elements from hub to shroud. The default limit is 100, so values close to this will normally be acceptable.
4. `Element Volume Ratio` - depending on the mesh, it may not be possible to satisfy this constraint.
5. `Connectivity Number` - may or may not be pertinent depending on the type of solver used. For details, see [Connectivity Number \(p. 192\)](#).

### 10.8.4. Volume

The `Mesh Analysis > Histogram Selection and Show Limits` volume objects provide convenient ways to examine the mesh. These function exactly the same as the volume object in the **Insert > User Defined** menu, except that the only variables available are the mesh statistics variables. For details, see [Volume Command \(p. 65\)](#).

---

**Note:**

The visibility of the volume object is off by default.

---

## 10.9. User Defined Objects

---

The `User Defined` branch in the object selector is initially empty. Any new objects created using the **Insert** menu will be displayed in this branch. Examples include points and legends. For details, see [Insert Menu \(p. 59\)](#).

## 10.10. Default Instance Transform

---

The default instance transform is initially applied to all objects for which instance transforms are possible. As a result, editing the definition of the default instance transform causes all such plots and objects to be transformed. Additional instance transform objects can be created. For details, see [Instance Transform Command \(p. 73\)](#). A different instance transform can be applied to an object using its **Render** tab. For details, see [Applying Instance Transforms to Objects \(p. 26\)](#).

## 10.11. Shortcut Menu Commands

---

You can perform many actions from shortcut menus; simply right-click one of the following:

- The viewer background (that is, a blank area in the viewer)
- An object in the viewer
- An object listed in the object selector (in the **Mesh** or **Geometry** workspace)
- An object listed in the object editor (in the **Mesh** or **Geometry** workspace)

The following sections describe all of the shortcut menu commands that can appear:

10.11.1. Assign [Family | Families | Geometry Only] To Topology Instance

10.11.2. Auto Add Layers and Insert Layers Automatically Commands

10.11.3. Clear Selected Entities Command

10.11.4. Color Command

10.11.5. Create Entity and Assign Family

10.11.6. Create Mesh Command

10.11.7. Create New View Command

10.11.8. Delete Complex Blade End Command

10.11.9. Delete New View Command

10.11.10. Delete Secondary Passage Command

10.11.11. Delete Selected Entity Command

10.11.12. Edit Blade Topology Command

10.11.13. Edit Command

10.11.14. Edit Complex Blade End Topology Command

10.11.15. Edit in Command Editor Command

10.11.16. Edit Render Properties Command and Edit Render Properties For All Entities Command

10.11.17. Fit View Command

10.11.18. Hide Command

10.11.19. Insert Blade CAD Feature [feature object name] (Mesh Workspace)

10.11.20. Insert Blade Command (Mesh Workspace)

10.11.21. Insert Blade Command (Geometry Workspace)

10.11.22. Insert Complex Blade End

10.11.23. Insert Edge Split Control Command

10.11.24. Insert Hub Complex Blade End Command

10.11.25. Insert Hub Tip Line

10.11.26. Insert Layer After and Insert Layer After Selected Layer Commands

10.11.27. Insert Layer Automatically Command

10.11.28. Insert Secondary Passage Boundary Command

10.11.29. Insert Secondary Passage Command

- 10.11.30. Insert Shroud Complex Blade End Command
- 10.11.31. Insert Shroud Tip Line
- 10.11.32. Insert USER DEFINED Object Command
- 10.11.33. Predefined Camera Commands
- 10.11.34. Projection Commands
- 10.11.35. Render Properties Edit Options Command
- 10.11.36. Render Properties Show Curves Command
- 10.11.37. Render Properties Show Surfaces Command
- 10.11.38. Render Properties Topology and Refined Mesh Visibility Commands
- 10.11.39. Save Picture Command
- 10.11.40. Show and Hide All Siblings Command
- 10.11.41. Show Free Edges
- 10.11.42. Show Object and Show Commands
- 10.11.43. Show Surface Mesh
- 10.11.44. Show Tip Shell Curves
- 10.11.45. Show Tip Shell Mesh
- 10.11.46. Span Curve Visibility Commands
- 10.11.47. Suspend Object Updates Command
- 10.11.48. Toggle Axis Visibility Command
- 10.11.49. Toggle Ruler Visibility Command
- 10.11.50. Transformation Commands and Coordinate Systems
- 10.11.51. Unassign Selected Geometries Command
- 10.11.52. Viewer Options Command

### 10.11.1. Assign [Family | Families | Geometry Only] To Topology Instance

Each of these commands associates the applicable CAD entity with a topological feature. For details, see [Associating CAD Objects in TurboGrid's Geometry Workspace](#) (p. 211).


### 10.11.2. Auto Add Layers and Insert Layers Automatically Commands

For details, see [Inserting Layers Automatically](#) (p. 185).

### 10.11.3. Clear Selected Entities Command

This command resets the settings for the topological entities (such as *LeadingCutoffSurface* and *LeadingEdge1*) that you have selected (using **Ctrl** or **Shift** while clicking). In particular, CAD geometry is unassigned.

### 10.11.4. Color Command

The **Color** command provides a subset of the functionality offered by the **Color** tab. The latter is available by selecting an eligible object from the object selector, then clicking *Rendering* . For details, see [Color Tab \(p. 23\)](#).

### 10.11.5. Create Entity and Assign Family

This command creates a topological feature for the object that is selected via the submenu. This command then associates the applicable CAD family with that new topological feature. For details, see [Associating CAD Objects in TurboGrid's Geometry Workspace \(p. 211\)](#).

### 10.11.6. Create Mesh Command

The **Create Mesh** command is available when the topology has been created. It is the same as selecting **Insert > Mesh**. For details, see [Mesh Command \(p. 59\)](#).

### 10.11.7. Create New View Command

Adds a new view based on the current state of the viewer.

### 10.11.8. Delete Complex Blade End Command

The **Delete Complex Blade End** command deletes a Blade Set > [blade name] > ComplexBladeEnd object in the **Geometry** workspace. For details, see [Complex Blade End \(p. 223\)](#).

### 10.11.9. Delete New View Command

Deletes an existing user-defined view.

### 10.11.10. Delete Secondary Passage Command

The **Delete Secondary Passage** command deletes a secondary flow path object from the Topological Entity Instances > Secondary Flow Paths object in the **Geometry** workspace.

### 10.11.11. Delete Selected Entity Command

This command deletes the selected entity.

### 10.11.12. Edit Blade Topology Command

The **Edit Blade Topology** command opens the blade settings in the object editor.

### 10.11.13. Edit Command

The **Edit** command opens the object editor for the selected object.

### 10.11.14. Edit Complex Blade End Topology Command

The **Edit Complex Blade End Topology** command enables you to edit a `Blade Set > [blade name] > ComplexBladeEnd` object in the **Geometry** workspace. For details, see [Complex Blade End](#) (p. 223).

### 10.11.15. Edit in Command Editor Command

The **Edit in Command Editor** command opens the **Command Editor** dialog box for the selected object. For details, see [Command Editor Command](#) (p. 96).

### 10.11.16. Edit Render Properties Command and Edit Render Properties For All Entities Command

The **Edit Render Properties** command enables you to adjust color settings for the selected object.

The **Edit Render Properties For All Entities** command enables you to adjust color settings for all entities that belong to a blade.

### 10.11.17. Fit View Command

The **Fit View** command centers the view on the displayed objects and sets the zoom to an appropriate level for viewing all of the visible objects. Only the active viewport is affected by this command.

---

**Note:**

Objects that have their visibility check box cleared do not necessarily fit into the viewer. Only visible objects are considered.

---

### 10.11.18. Hide Command

The **Hide** command turns off the visibility of the selected object(s).

To turn the visibility back on for an object in the **Mesh** workspace, find the object in the object selector and select its check box, or right-click the object in the object selector and select **Show**, or use the appropriate **Show Object** command available in the shortcut menu that appears when right-clicking the viewer background.

To turn the visibility back on for an object in the **Geometry** workspace, find the object in the object selector and select its check box, or right-click the object in the object selector and select **Show**.

### 10.11.19. Insert Blade CAD Feature [feature object name] (Mesh Workspace)

The **Insert > Blade CAD Feature > [feature object name]** command opens the **Blade CAD Features** object editor, which is used to specify CAD features, such as a blends and partial tips, for generated CAD. For details, see [Blade CAD Features](#) (p. 132).

### 10.11.20. Insert Blade Command (Mesh Workspace)

The **Insert > Blade** command adds a blade to the blade set. After selecting this command, Ansys TurboGrid prompts you for a name for the blade, then generates the new blade object. If you already have a blade loaded, most settings for the new blade will default to those from the first blade. Simply change the **File Name** setting to indicate the file for the added blade and click **Apply**.

### 10.11.21. Insert Blade Command (Geometry Workspace)

The **Insert Blade** command adds a blade to the `Blade Set` group of topological entity instances. After selecting this command, Ansys TurboGrid prompts you for a name for the blade, the leading edge type, and the trailing edge type, then generates a new blade object.

### 10.11.22. Insert Complex Blade End

The **Insert Complex Blade End** command adds a `Blade Set > [blade name] > ComplexBladeEnd` object in the **Geometry** workspace. The `ComplexBladeEnd` object enables you to define the geometry of one or two complex blade ends by specifying the constituent CAD entities. For details, see [Complex Blade End \(p. 223\)](#).

### 10.11.23. Insert Edge Split Control Command

The **Insert Edge Split Control** command is available by right-clicking (on a visible layer, in the viewer when in the **Mesh** workspace) a master topology line, a topology line, or a refined mesh line. This command creates/edits an edge split control object for the nearest master topology segment. Edge split control objects appear in the object selector under `Mesh Data`. For details, see [Changing the Number of Elements on a Selected Master Topology Edge \(p. 182\)](#).

### 10.11.24. Insert Hub Complex Blade End Command

The **Insert Hub Complex Blade End** command enables you to add a hub complex blade end child object to the `Blade Set > [blade name] > ComplexBladeEnd` object in the **Geometry** workspace. For details, see [Complex Blade End \(p. 223\)](#).

### 10.11.25. Insert Hub Tip Line

The **Insert Hub Tip Line** command enables you to select a hub tip line from the CAD geometry. This command is offered only when the hub curve has not been defined.

### 10.11.26. Insert Layer After and Insert Layer After Selected Layer Commands

For details, see [Inserting Layer After Selected Layer \(p. 186\)](#).

### 10.11.27. Insert Layer Automatically Command

For details, see [Inserting Layers Automatically \(p. 185\)](#).

## 10.11.28. Insert Secondary Passage Boundary Command

The **Insert Secondary Passage Boundary** command enables you to add a boundary entity to a secondary flow path object. For details, see [Associating CAD Objects in TurboGrid's Geometry Workspace](#) (p. 211).

## 10.11.29. Insert Secondary Passage Command

The **Insert Secondary Passage** command enables you to define the geometry of a secondary flow path by specifying the constituent boundary entities. For details, see [Associating CAD Objects in TurboGrid's Geometry Workspace](#) (p. 211).

## 10.11.30. Insert Shroud Complex Blade End Command

The **Insert Shroud Complex Blade End** command enables you to add a shroud complex blade end child object to the `Blade Set > [blade name] > ComplexBladeEnd` object in the **Geometry** workspace. For details, see [Complex Blade End](#) (p. 223).

## 10.11.31. Insert Shroud Tip Line

The **Insert Shroud Tip Line** command enables you to select a shroud tip line from the CAD geometry. This command is offered only when the shroud curve has not been defined.

## 10.11.32. Insert USER DEFINED Object Command

The **Insert > USER DEFINED Object** command creates a new object. It functions like the **Insert** menu. For details, see [Insert Menu](#) (p. 59).

## 10.11.33. Predefined Camera Commands

The **Predefined Camera** commands allow you to set the viewing angle according to one of the built-in presets. These commands also set the zoom level automatically after adjusting the viewing angle.

## 10.11.34. Projection Commands

### 10.11.34.1. Orthographic Command

The **Orthographic** command sets an orthographic view.

### 10.11.34.2. Perspective Command

The **Perspective** command sets a view with a fixed amount of perspective.

## 10.11.35. Render Properties Edit Options Command

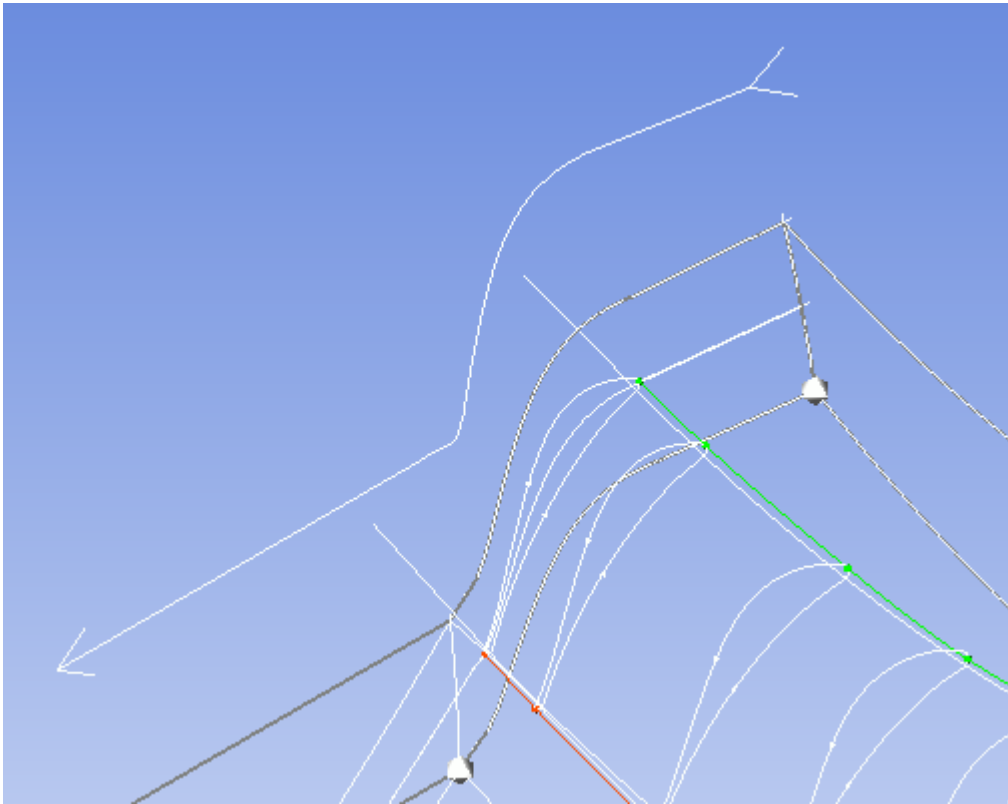
The **Render (Properties) > Edit Options** command enables you to adjust color and render settings for the selected object. For details, see [Color Tab](#) (p. 23) and [Render Tab](#) (p. 25).

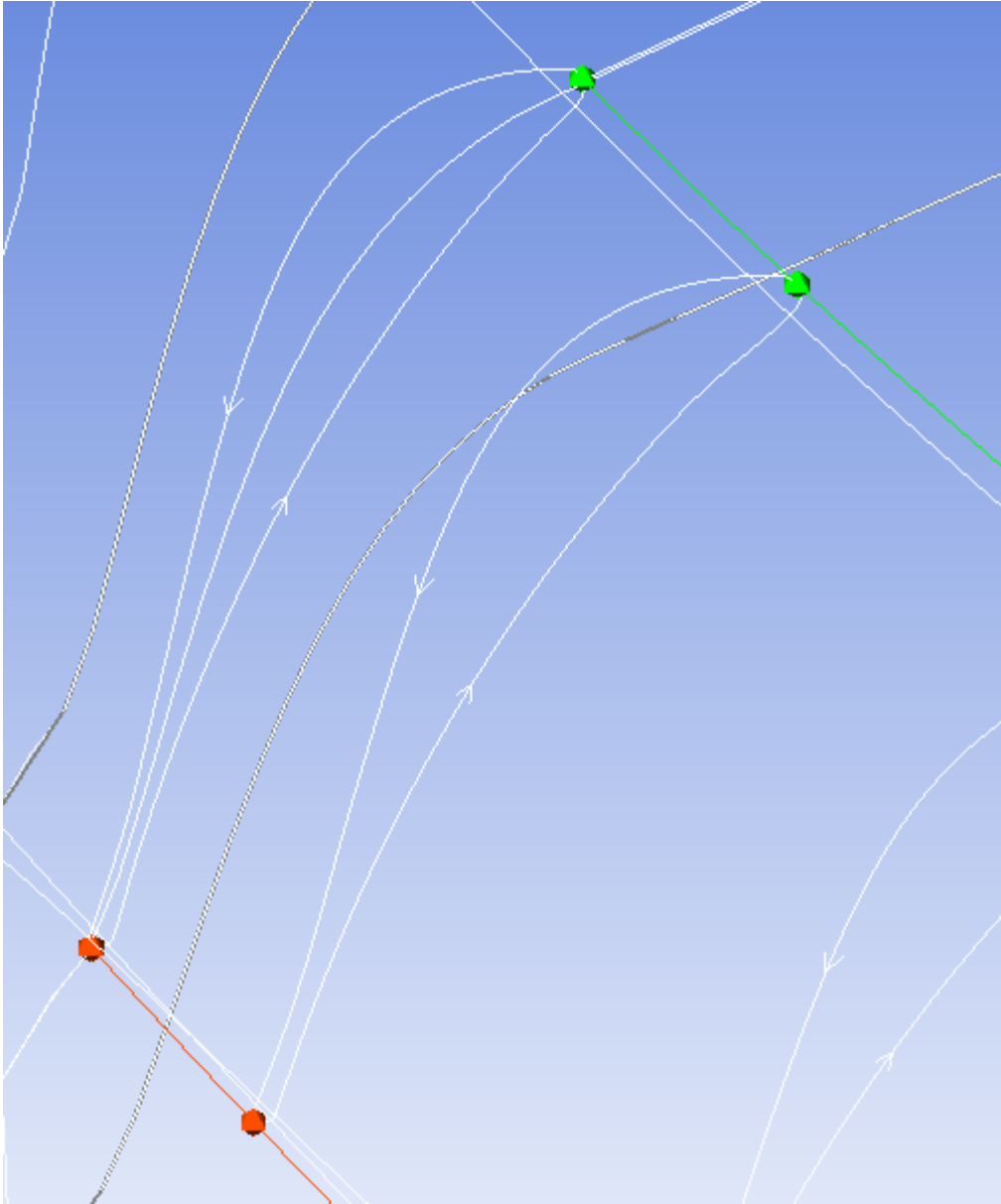
## 10.11.36. Render Properties Show Curves Command

The **Render (Properties) > Show Curves** command is available by right-clicking an object (for example, the hub, shroud or blade surface). Use this command to display curves instead of surfaces. If, in the geometry browser, **Input Mode** is set to `Profile Points` or `Profile Points With Secondary CAD`, then the curves displayed on the blade correspond to the raw curve file data. For other input modes, the curves represent the underlying CAD surfaces.

For curves displayed on the blade that correspond to the raw curve file data, and for the hub and shroud curves, arrowheads are added to show the direction of increasing point index number. The figures below show curves with arrowheads.

**Figure 10.19: Hub Curve With Direction Arrows**



**Figure 10.20: Blade Profiles With Direction Arrows**

### 10.11.37. Render Properties Show Surfaces Command

The **Render (Properties) > Show Surfaces** command is available by right-clicking the hub or shroud curve, or one of the blade curves. Use this command to view the surface instead of the raw curve file data.

---

**Note:**

The blade surface and resulting mesh can be adversely affected when a blade shroud tip is defined by the second-last profile and the last profile has a shape that is inconsistent with where the blade would exist if the blade were extended in the spanwise direction while ignoring the last profile. To avoid this problem, you can either ensure that the last

profile is consistent with the second last profile, or manually remove the last profile from the curve file. A similar statement applies for the hub end of a blade having a hub tip.

---

### 10.11.38. Render Properties Topology and Refined Mesh Visibility Commands

The **Render (Properties) > Master Topology Visibility, Topology Visibility, and Refined Mesh Visibility** commands toggle the visibility of the corresponding items for the selected layer(s).

### 10.11.39. Save Picture Command

The **Save Picture** command is available by right-clicking the background. It is the same as selecting **File > Save Picture**.

### 10.11.40. Show and Hide All Siblings Command

The **Show + Hide All Siblings** command turns on the visibility of the object selected in the object selector, and turns off the visibility of every sibling object in the tree.

### 10.11.41. Show Free Edges

The **Show Tip Shell Curves** command toggles the visibility of the free edges in the surface mesh of the applicable complex blade end `Mesh Data` object. Showing the free edges can indicate holes in the CAD geometry. This command is available only when meshing with high-fidelity geometry fails.

### 10.11.42. Show Object and Show Commands

The **Show Object** commands are available by right-clicking the background in the viewer when in the **Mesh** workspace. These commands turn on the visibility of any geometry, volume, or mesh data object.

The **Show** command turns on the visibility of the object(s) selected in the object selector.

As an alternative, you can select the check box next to an item in the object selector to show that item in the viewer.

### 10.11.43. Show Surface Mesh

The **Show Surface Mesh** command toggles the visibility of the surface mesh of the applicable complex blade end `Mesh Data` object. This command is available only when meshing with high-fidelity geometry fails.

### 10.11.44. Show Tip Shell Curves

The **Show Tip Shell Curves** command toggles the visibility of the tip shell curves of the applicable complex blade end `Mesh Data` object.

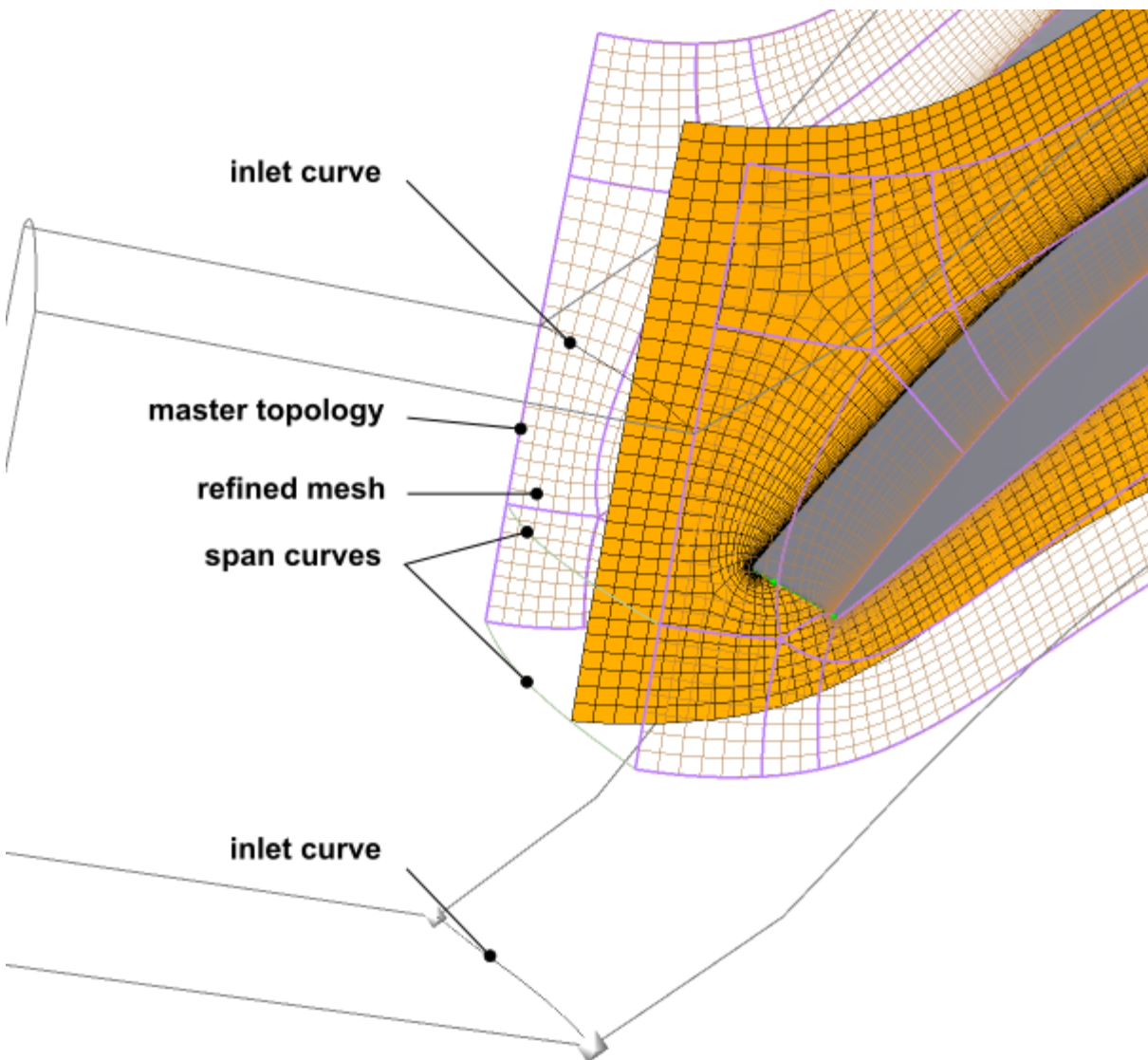
## 10.11.45. Show Tip Shell Mesh

The **Show Tip Shell Mesh** command toggles the visibility of the tip shell mesh of the applicable complex blade end Mesh Data object.

## 10.11.46. Span Curve Visibility Commands

The **Span Curve Visibility** commands are available by right-clicking (on a visible layer, in the viewer when in the **Mesh** workspace) a master topology line, a topology line, or a refined mesh line. These commands control the visibility of the span curves. A span curve is a line that passes through corresponding master topology nodes across layers. Span curves are used to construct the 3D mesh. A span curve that is sharply curved can indicate a potential problem with mesh quality.

**Figure 10.21: Span Curves**



The **Toggle span curve visibility** command toggles the visibility of the span curve that passes through the nearest node of the master topology.

The **Turn on all span curves** and **Turn off all span curves** commands control the visibility of all span curves at once.

### 10.11.47. Suspend Object Updates Command

You can save time by appropriately suspending objects such as the `Topology Set` and `3D Mesh` objects. The **Suspend Object Updates** command toggles the state of suspension of an object. By suspending an object, you prevent it (and objects that depend on it) from being updated in response to changes to other objects.

For example, the `Topology Set` object depends on the `Geometry` object, and, when not suspended, will be reprocessed each time the geometry is changed. If you want to make several changes to the geometry, you can skip the intermediate processing of the topology by suspending the `Topology Set` object. Note that the `Topology Set` object is initially suspended when you start a new case.

To process a suspended object, unsuspend it.

An icon overlay indicates whether or not an object is suspended. Icon overlays are illustrated in [Table 1.1: Icon Overlays and Text Styles \(p. 20\)](#).

---

**Note:**

The **Undo** command cannot properly undo a suspension change (for example, to the `Topology Set` object); attempting to undo a suspension change can lead to an inconsistent state.

---

### 10.11.48. Toggle Axis Visibility Command

The **Toggle Axis Visibility** command is available by right-clicking the background. It controls the visibility of the axis orientation indicator that appears in the lower-right corner of the viewer.

### 10.11.49. Toggle Ruler Visibility Command

The **Toggle Ruler Visibility** command is available by right-clicking the background. It controls the visibility of the ruler that appears near the lower edge of the viewer.

### 10.11.50. Transformation Commands and Coordinate Systems

The **Transformation** commands are available by right-clicking the background.

You can use different coordinate systems to display various views of the geometry and mesh objects in the viewer window. This is useful when closely examining every aspect of the mesh before saving it for use in a CFD solver.

#### 10.11.50.1. Cartesian

The Cartesian coordinate system is used by default in the viewer window when there is only one viewport. The 3 axis coordinates for the Cartesian coordinate system are X, Y and Z.

### 10.11.50.2. Blade-to-Blade

The blade-to-blade coordinate system is used to display the geometry in the  $m'$  -  $\theta$  conformal space, which is familiar to blade designers. The  $m'$  -  $\theta$  coordinate space is angle-preserving and minimizes the effect of changing radius on viewing and manipulation. By utilizing the blade-to-blade coordinate system, a wide variety of machine types, from axial to radial, can be treated similarly.

$m'$  is defined in [Meridional](#) (p. 207).

---

#### Note:

Due to the fact that  $m'$  is ill-conditioned at  $r=0$ , you can expect to see different behavior in cases for which the geometry extends to the machine axis.

---

### 10.11.50.3. Meridional

The meridional coordinate system is one transformation used by blade designers. It is useful for viewing the flow path as well as the upstream and downstream extents of the mesh. The three axis coordinates for the meridional coordinate system are A (axial), R (radial) and  $\theta$  (Theta). The viewer shows only the A and R coordinates.

Variable	Description
$a$	Axial location
$r$	Radius or radial location
$s$	Normalized distance along meridional curve (for example, from 0 to 1)
$S$	The particular value of $s$ that corresponds to the point location for which $m$ and $m'$ are to be computed
$m$	Meridional coordinate (distance along meridional curve)
$m'$	Normalized meridional coordinate (radius normalized distance along meridional curve)

$$m = \int_0^S \sqrt{\left(\frac{dr}{ds}\right)^2 + \left(\frac{da}{ds}\right)^2} ds$$

$$m' = \int_0^S \left[ \frac{\sqrt{\left(\frac{dr}{ds}\right)^2 + \left(\frac{da}{ds}\right)^2}}{r} \right] ds$$

### 10.11.50.4. 3D Turbo

The three axis coordinates for the 3D Turbo Coordinate System are M ( $m'$ ), T (Theta) and S (span).

## 10.11.51. Unassign Selected Geometries Command

Unassigns the applicable geometries.

## 10.11.52. Viewer Options Command

The **Viewer Options** command is available by right-clicking the background. It is the same as selecting **Edit > Options** and then navigating to **TurboGrid > Viewer**. For details, see [Viewer \(p. 49\)](#).

---

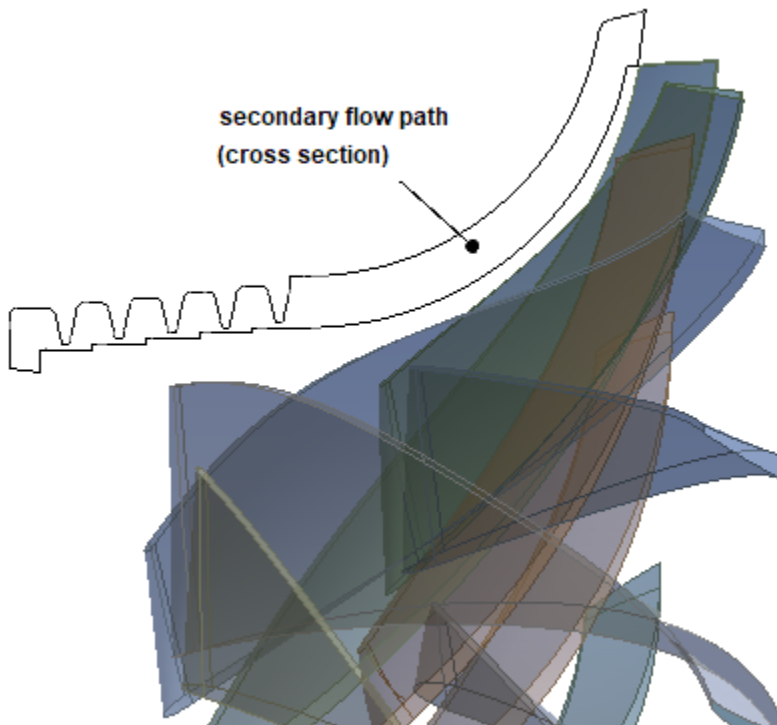
# Chapter 11: Secondary Flow Paths

---

Some turbomachinery designs have secondary flow paths that are outside the main passage.

TurboGrid enables you to add one or more secondary flow paths to a case.

**Figure 11.1: Secondary Flow Path for a Compressor**



Each secondary flow path is axisymmetric, located outside the main flow passage, and has one or more interfaces with either the hub or shroud.

The following topics are discussed:

- [11.1. Creating Geometry for Input to TurboGrid](#)
- [11.2. Specifying Secondary Flow Path Geometry in TurboGrid's Geometry Workspace](#)
- [11.3. Specifying Secondary Flow Path Details in TurboGrid's Mesh Workspace](#)
- [11.4. Specifying Secondary Flow Paths By Importing NDF Files](#)

## 11.1. Creating Geometry for Input to TurboGrid

---

The shape of a secondary flow path is mainly defined by a curvilinear closed loop on an axial-radial plane. The loop is made up of several segments, known here as boundary entities, that are connected end-to-end.

The following topics are discussed:

[11.1.1. Source of Geometry](#)

[11.1.2. Interfaces with the Main Passage](#)

### 11.1.1. Source of Geometry

In order to be able to create secondary flow paths in TurboGrid, the geometry must be defined from a CAD source. TurboGrid has no support for creating secondary flow paths for geometry that is defined from profile points. For details on defining geometry in TurboGrid using these different sources of geometric data, see [Geometry \(p. 103\)](#).

Note that BladeEditor can serve as a source of CAD geometry for TurboGrid. In particular, native BladeEditor blades (blades that were produced by the Blade or Splitter feature, without subsequent modification to the blade or splitter) with added 2D sketches of the secondary flow path(s) can be sent to TurboGrid as CAD data by making a cell connection in Workbench from the Geometry cell (that represents BladeEditor) to the Turbo Mesh cell (that represents TurboGrid).

To prepare the geometry in BladeEditor:

1. Create line bodies for the sketches: at least one line body for each boundary entity that will be made in TurboGrid.
2. Create Attributes (**Tools > Attribute**): as many as one for each line body; at least one for each boundary entity that will be made in TurboGrid.

The `Attribute Name` property values (that is, the attribute text names) must be prefixed with "NamedSelection:". The remainder of the name is what will appear in downstream applications. Note: Unicode is not supported for names in TurboGrid. Set the attribute `Geometry` property by selecting edges from the defined wire bodies.

### 11.1.2. Interfaces with the Main Passage

Any given secondary flow path is permitted to have one or more interfaces with the main passage's hub and shroud boundaries.

At each interface, the curvilinear closed loop that defines the secondary flow path must be near or cross either the hub or shroud. The curvilinear closed loop is adjusted as needed so that it follows the hub or shroud along the interface.

## 11.2. Specifying Secondary Flow Path Geometry in TurboGrid's Geometry Workspace

---

Your input is required in order for TurboGrid to be able to recognize, within a given loaded CAD geometry, each secondary flow path and its various boundary entities (walls and hub or shroud interfaces).

You can provide this input through TurboGrid's **Geometry** workspace.

The following topics are discussed:

[11.2.1. Associating CAD Objects in TurboGrid's Geometry Workspace](#)

## 11.2.1. Associating CAD Objects in TurboGrid's Geometry Workspace

This section describes how to set up secondary flow path geometry in TurboGrid using the **Geometry** workspace.

With the CAD geometry already loaded into TurboGrid, you can specify the geometry for each secondary flow path as follows:

1. In the **Geometry** workspace tree, right-click `Topological Entity Instances > Secondary Flow Paths` and select **Insert Secondary Passage**.

The **Create New Secondary Passage Topology** dialog box appears.

2. Enter a name for the secondary flow path and click **OK**.

A new, incomplete, secondary flow path object appears in the tree under `Secondary Flow Paths`. This object requires boundary entity definitions.

3. Define boundary entities to complete the secondary flow path object using one or both of the following procedures:

### Create a boundary entity and then assign it one or more CAD objects

1. Right-click the new secondary flow path object and select **Insert Secondary Passage Boundary**.


The **Create New Secondary Passage Boundary** dialog box appears.

2. Enter a name for a new boundary entity for the secondary flow path object and click **OK**.

A new, incomplete, boundary entity appears in the tree under the secondary flow path object that is being defined. This boundary entity requires an association with a curvilinear line selected from the `CAD Families` branch.

3. Assign CAD objects to the boundary entity using either or both of the following methods:

- Preferred Method — Open the boundary entity in the object editor, then make CAD assignments:

- By using the *CAD Entity Selector*  icon (beside the **Location** field) to open the **CAD Region Entities** dialog box, which you can use to select CAD objects, and/or
- By typing a comma-delimited list of CAD object names into the **Location** field.

- Alternative Method — Right-click one CAD object (or optionally a selection of multiple CAD objects made using **Shift/Ctrl**) and select **Assign [Family | Families | Geometry Only] To Topology Instance > [Name of the secondary flow path object that has an incomplete boundary entity] > [Name of the incomplete boundary entity that requires an association with the CAD object(s)]**.

### Create a boundary entity directly from a CAD object

1. Right-click a CAD family (a high-level item listed directly under `CAD Families`) and select **Create Entity and Assign Family > [Name of the secondary flow path object that requires a new boundary entity]**.

The **Create New Secondary Passage Boundary** dialog box appears.

2. Enter a name for a new boundary entity for the secondary flow path object that is being defined and click **OK**.

A new, complete, boundary entity is created and appears in the tree under the secondary flow path object that you chose in the previous step. This boundary entity is associated with the CAD family that you right-clicked in the previous step.

---

#### Note:

CAD objects include families and their constituent primitives. If you assign a CAD family to a boundary entity, the entire family is associated with the boundary entity. Any upstream geometry updates to the family will be propagated to the assigned topological entity. If you assign a selection of CAD primitives (curves or surfaces) to a boundary entity, the family association will be lost and only the specified primitives will be maintained during upstream geometry updates.

This also applies for all other topological entities.

---

## 11.3. Specifying Secondary Flow Path Details in TurboGrid's Mesh Workspace

---

With the geometry loaded and with the secondary flow paths recognized by TurboGrid (as reflected in the **Geometry** workspace), you can use the **Mesh** workspace to specify meshing details of the secondary flow paths (as well as the main passage).

In the **Mesh** workspace tree, the secondary flow path objects are listed under three locations, as detailed in the following sections:

[11.3.1. Geometry Branch](#)

[11.3.2. Mesh Data Branch](#)

[11.3.3. 3D Mesh Branch](#)

### 11.3.1. Geometry Branch

The secondary flow path objects listed under `Geometry > Secondary Flow Paths` provide the following functions:

- You can toggle the visibility of the geometry of a secondary flow path, or any of its boundaries, by toggling the corresponding visibility check box.

- You can edit any boundary of a secondary flow path in order to change its type via the **Boundary Type** setting. The available types are: `Wall`, `Hub Interface`, `Shroud Interface`. The type specification affects meshing details near the boundary.

A type setting of `Hub Interface` or `Shroud Interface` makes the **Automatically Manage Opening Region** setting available. Selecting the latter causes the hub or shroud (as applicable) to be divided into regions; in particular, one of the regions is made to match (line up with) the interface. The resulting passage mesh is therefore able to have nodes that are aligned with the interface edges, facilitating mesh refinement within the passage mesh near the interface edges. The regions that are created in this process can be reviewed in the object editor for `Geometry > Hub` on the **Hub Regions** tab, or the corresponding tab for the shroud, as applicable.

Note that, when using the **Automatically Manage Opening Region** option, the interface must be fully within the inlet or outlet block.

## 11.3.2. Mesh Data Branch

The secondary flow path objects listed under `Mesh Data` provide the following functions:

- The object editor for a secondary flow path object provides controls that affect the mesh within the bulk of the secondary passage.
- The object editor for a given boundary of a secondary flow path object provides similar settings that affect the mesh near the boundary. These settings enable you to override, for the applicable boundary, the default settings that are established for the secondary flow path (parent object).

### 11.3.2.1. Setting Descriptions for the Secondary Flow Path Object

- **Triangle Refinement Factor**

As this value increases, the element size in the mesh interior decreases.

- **Default Boundary Edge Refinement Factor**

As this value increases, the node spacing along the boundary decreases.

- **Inflation Options**

- **Default Number of Boundary Layers**

The number of rows of hexahedral elements in the inflation layer.

- **Default Inflation Growth Ratio**

Height ratio between a row of mesh elements in the inflation layer and the adjacent row closer to the boundary.

- **Advanced Options**

- **Default Last Layer Height Ratio**

This is the length scale ratio between the boundary layer mesh elements and the interior mesh elements where the boundary layer elements meet the interior elements.

- **Default Maximum Feature Angle** and **Default Curvature Angle**

Decreasing the **Maximum Feature Angle** reduces the "detection" of boundary curve details.

Increasing the **Curvature Angle** reduces the number of elements along a curved section of the boundary.

Note that the **Maximum Feature Angle** and **Curvature Angle** values must sum to a value greater than 180 degrees.

- **Default Cells Per Gap (Proximity Refinement)**

This parameter controls the mesh proximity. Increasing this value increases boundary and interior mesh refinement in narrow gaps.

- **3D Options** (available only for the secondary flow path object, not a boundary object)

A secondary flow path is typically modeled as a sector of the full 360° geometry. The appropriate pitch (angular extent about the rotation axis) of the sector, and number of elements across the pitch, depend on details of the intended CFD simulation:

- If you intend to connect the main passage to the secondary flow path using a mixing plane interface (to model swirling flow with no circumferential variation) then the secondary flow path can be modeled using a thin, almost 2D, mesh that has a minimum of 2 layers of mesh elements across the pitch.
- If you want to model circumferential flow variations in the secondary flow path (for example using a `Frozen Rotor` or `Transient Rotor-Stator` frame change model in CFX) then the secondary flow path should have an appropriate pitch.

The settings under **3D Options** are:

- **Number of Extrusions**

Specify the number of rows of elements to generate (minimum 2) in the circumferential direction.

- **Theta Extrusion Angle**

Specify the sector angle (pitch) for the secondary flow path mesh.

- **Theta Extrusion Base**

Specify the starting value of Theta for the mesh extrusion.

### 11.3.2.2. Setting Descriptions for the Secondary Flow Path's Boundary Object

- **Inflation Zone**

This setting controls whether or not inflation is applied at the specified boundary. A value of `Default` applies inflation if the boundary is a wall; it applies no inflation if the boundary is an interface.

- **First Layer Height**

A read-only display of the average height of the row of mesh elements next to the boundary.

---

**Note:**

There is no setting to directly control the first element height in an inflation region. However, you can see the average height in the object editor for a boundary. To increase/decrease the height, decrease/increase the number of boundary layers. Other mesh factors affect the height, including the **Global Size Factor** (see [Method \(p. 167\)](#)).

---

- **Use Override Boundary Settings (available only for a boundary object, not the secondary flow path object)**

Select this option to override defaults set for the secondary flow path object:

- **Freeze Edges**

Try changing this option if there is a mesh quality problem at a transition between a boundary with an inflation layer and a boundary without an inflation layer.

### 11.3.3. 3D Mesh Branch

The secondary flow path objects listed under 3D Mesh provide the following functions:

- You can toggle the visibility of a mesh surface associated with a secondary flow path by toggling the corresponding visibility check box.
- You can edit any boundary of a secondary flow path in order to change its color and/or rendering settings.

For details, see [Surface Groups \(p. 70\)](#).

## 11.4. Specifying Secondary Flow Paths By Importing NDF Files

---

Importing an NDF file into TurboGrid involves generating and then loading CAD geometry. For details, see [Defining Geometry from Generated CAD: CAD From NDF \(p. 112\)](#).

In the NDF file, the specification for the flow path, <flow-path>, can include <cavity-objects> specifications for one or more cavities that form secondary flow paths. Format details are given in [Appendix 2: BNF Rule List in the TurboSystem User's Guide](#).

In order for a secondary flow path to be set up in the **Mesh** workspace upon importing an NDF file, ensure that each interface and wall boundary that forms part of the outline of the secondary flow path is in its own sketch, and that the sketches collectively form a closed loop.

The following is an example <cavity-objects> specification that could be found in a <flow-path> specification of an NDF file:

---

**Note:**

The <manage-opening> specifications currently have no effect.

---

```
<cavity-objects>
  <cavity>
    <cavity-name>ShrCavity</cavity-name>
    <cavity-type>Shroud</cavity-type>
    <boundary-objects>
      <boundary>
        <boundary-name>Inner</boundary-name>
        <sketch-reference>SecPassInnerWall</sketch-reference>
        <boundary-type>Wall</boundary-type>
        <manage-opening>>false</manage-opening>
      </boundary>
      <boundary>
        <boundary-name>Outer</boundary-name>
        <sketch-reference>SecPassOuterWall</sketch-reference>
        <boundary-type>Wall</boundary-type>
        <manage-opening>>false</manage-opening>
      </boundary>
      <boundary>
        <boundary-name>LE</boundary-name>
        <sketch-reference>SecPassOpenLE</sketch-reference>
        <boundary-type>Interface</boundary-type>
        <manage-opening>>false</manage-opening>
      </boundary>
      <boundary>
        <boundary-name>TE</boundary-name>
        <sketch-reference>SecPassOpenTE</sketch-reference>
        <boundary-type>Interface</boundary-type>
        <manage-opening>>false</manage-opening>
      </boundary>
    </boundary-objects>
  </cavity>
  <cavity>
    <cavity-name>IGVHubCavity</cavity-name>
    <cavity-type>Hub</cavity-type>
    <boundary-objects>
      <boundary>
        <boundary-name>Inner</boundary-name>
        <sketch-reference>IGVSecPassInner</sketch-reference>
        <boundary-type>Wall</boundary-type>
        <manage-opening>>false</manage-opening>
      </boundary>
      <boundary>
        <boundary-name>Outer</boundary-name>
        <sketch-reference>IGVSecPassOuter</sketch-reference>
        <boundary-type>Wall</boundary-type>
        <manage-opening>>false</manage-opening>
      </boundary>
      <boundary>
        <boundary-name>LE</boundary-name>
        <sketch-reference>IGVOpeningLE</sketch-reference>
        <boundary-type>Interface</boundary-type>
        <manage-opening>>false</manage-opening>
      </boundary>
      <boundary>
        <boundary-name>TE</boundary-name>
        <sketch-reference>IGVOpeningTE</sketch-reference>
        <boundary-type>Interface</boundary-type>
        <manage-opening>>false</manage-opening>
      </boundary>
    </boundary-objects>
  </cavity>
</cavity-objects>
```

```
</cavity>  
</cavity-objects>
```



---

## Chapter 12: Meshing High-Fidelity Geometry

---

Some turbomachinery designs have complex geometric features within the main passage.

TurboGrid has support for meshing complex geometric features at blade ends. You can use the complex blade end high-fidelity geometric feature to mesh various complex tips, including partial tips, squealer tips, and true blade blends.

Figure 12.1: [Inlet Guide Vane with Partial Tip Clearance](#) (p. 219) shows an example of a blade with partial tip clearance.

**Figure 12.1: Inlet Guide Vane with Partial Tip Clearance**

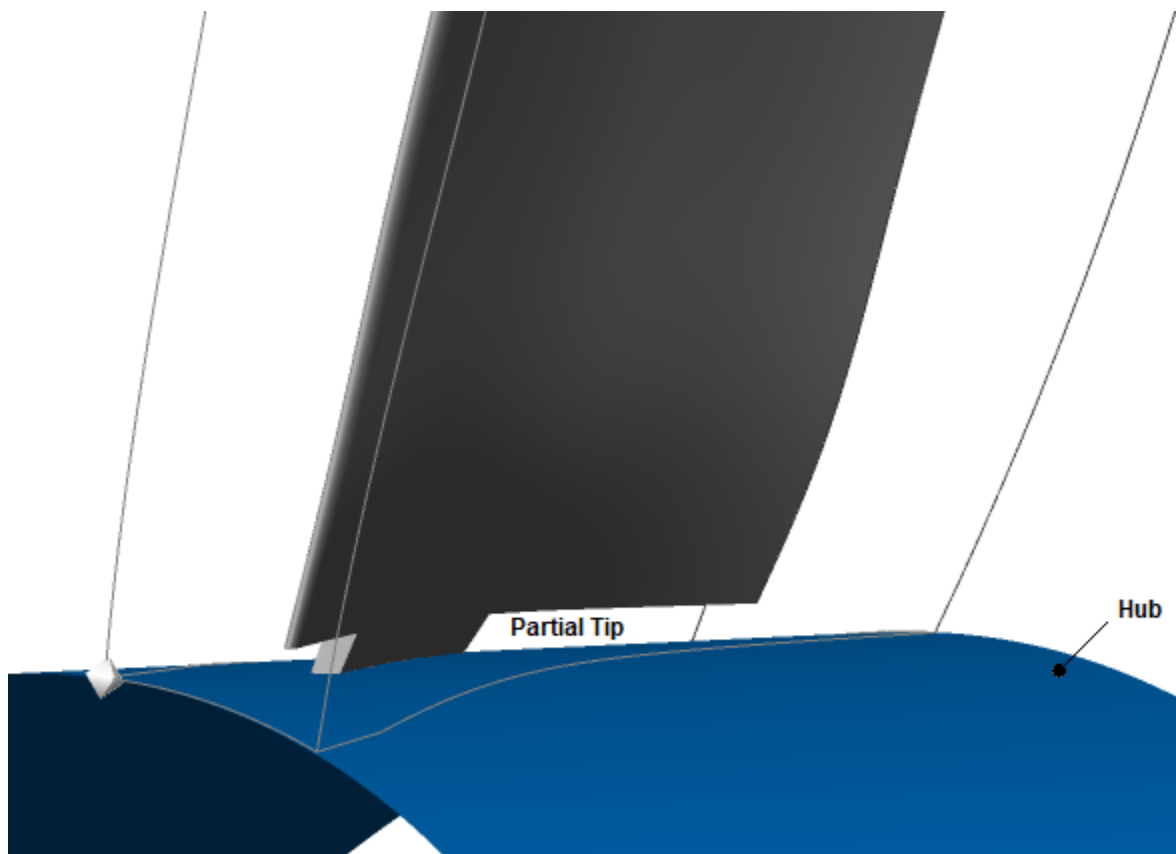
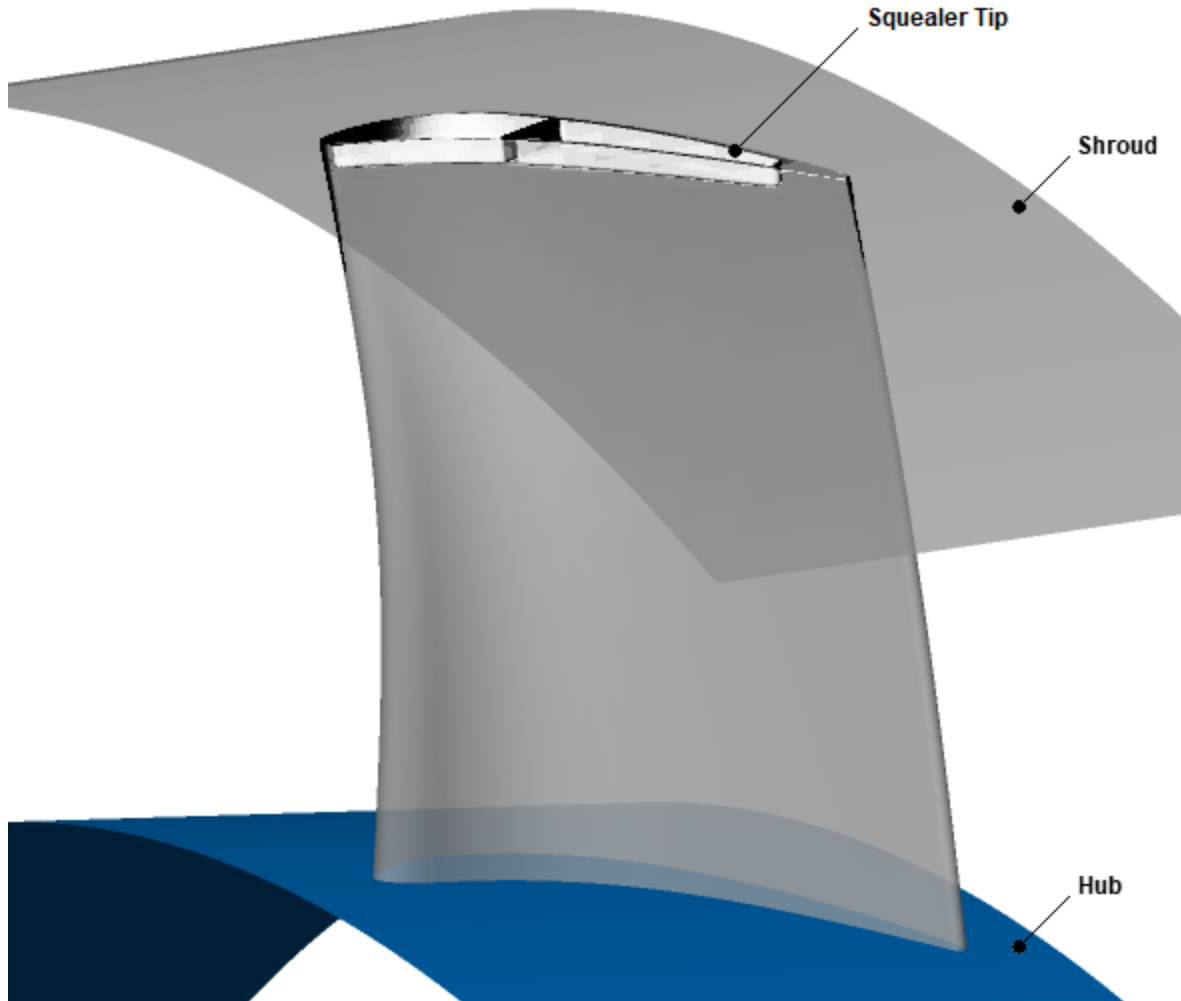


Figure 12.2: [Turbine Rotor with Squealer Tip](#) (p. 220) shows an example of a blade with a squealer tip.

**Figure 12.2: Turbine Rotor with Squealer Tip**



Meshing complex blade ends involves using a high-fidelity geometry definition of the blade, hub, and shroud, provided by CAD data.

Meshing with high-fidelity geometry involves a hybrid meshing approach, with blocks of unstructured meshes surrounding high-fidelity geometric features and blocks of structured meshes filling the remainder of the blade passage.

---

**Note:**

Meshing with high-fidelity geometry is not currently supported for multi-blade cases.

---

The following topics are discussed:

- [12.1. General Procedure for Meshing of Low-Fidelity and High-Fidelity Geometry](#)
- [12.2. Sources of High-Fidelity Geometry for TurboGrid](#)
- [12.3. Associating CAD Objects in TurboGrid's Geometry Workspace](#)
- [12.4. Using ATM3D or ATM v2](#)
- [12.5. Working with Meshes for High-Fidelity Geometry in TurboGrid's Mesh Workspace](#)

12.6. Analyzing the Mesh

12.7. Saving the Mesh

## 12.1. General Procedure for Meshing of Low-Fidelity and High-Fidelity Geometry

---

1. Import CAD or generate CAD. The resulting CAD geometry contains both low-fidelity and high-fidelity representations of the geometry.

For details, see [Sources of High-Fidelity Geometry for TurboGrid \(p. 221\)](#).

2. For imported CAD: In the **Geometry** workspace, specify topological entities, including at least one that is for the high-fidelity geometric feature.

For generated CAD: The topological entities are automatically specified.

3. Activate ATM3D.

4. Adjust the Mesh Data settings as needed.

The Mesh Data settings include settings for each unstructured mesh block surrounding a high-fidelity geometric feature.

5. Analyze the mesh to ensure quality by checking the mesh statistics and using 3D Mesh and other visualization objects as needed.

6. Save the mesh.

At any time after the high-fidelity topological entities are defined, you can view the low-fidelity geometry, the high-fidelity geometry, or both at the same time.

When generating a mesh, you can suppress the unstructured mesh block for some or all high-fidelity geometric features, for example to reduce mesh generation time when doing preliminary design work.

## 12.2. Sources of High-Fidelity Geometry for TurboGrid

---

High-fidelity geometry, together with low-fidelity geometry, may be obtained by:

- **Importing CAD**

Outside of the unstructured mesh blocks surrounding high-fidelity geometric features, the low-fidelity geometry and high-fidelity geometry must be the same to avoid kinks or jumps at the interfaces between structured and unstructured mesh blocks.

Note that BladeEditor can serve as a source of CAD geometry for import into TurboGrid. In particular, BladeEditor blades can be sent to TurboGrid as CAD data by making a cell connection in Workbench from the Geometry cell (that represents BladeEditor) to the Turbo Mesh cell (that represents TurboGrid).

- **Generating CAD by Importing an NDF File**

You can import an NDF file, causing a CAD definition to be generated then loaded. The selected blade row is loaded at the end of the import process; any other blade rows in the CAD definition remain available to be loaded. For details, see [Defining Geometry from Generated CAD: CAD From NDF \(p. 112\)](#).

- **Generating CAD from Profile Points and Optional Blade CAD Feature Specifications**

You can generate CAD based on profile points. For details, see [Defining Geometry from Generated CAD: CAD From Profile Points \(p. 112\)](#).

CAD generated based on profile points may contain blade CAD feature specifications such as blade blends and partial tips. For details, see [Blade CAD Features \(p. 132\)](#).

## 12.3. Associating CAD Objects in TurboGrid’s Geometry Workspace

---

This section describes how to set up high-fidelity geometry for CAD imported into TurboGrid. Your input is required in order for TurboGrid to be able to recognize the high-fidelity geometric features such as complex blade ends. Specifically, you must create one or more topological entities for the high-fidelity geometric features and associate those entities with CAD families or other CAD objects.

With the CAD geometry already imported into TurboGrid, you can set up a high-fidelity geometric feature as follows:

1. In the **Geometry** workspace tree, right-click the appropriate object under `Topological Entity Instances` and select a command from the shortcut menu to add a topological entity for the high-fidelity geometric feature.

A new, incomplete, entity appears in the tree. This object requires CAD entity definitions.


For example, in the **Geometry** workspace tree, right-click `Topological Entity Instances > Blade Set > [blade name]` and select **Insert Complex Blade End**.

In this example, a new, incomplete, complex blade end topological entity named `ComplexBladeEnd` appears in the tree. This object requires CAD entity definitions for two or three of its three child objects: `HighFidelityBladeSurface`, `HighFidelityHubSurface`, `HighFidelityShroudSurface`.

2. Complete the topological entity definition for the high-fidelity geometric feature as follows:

1. Assign CAD objects to a topological entity using either or both of the following methods:

- Preferred Method — Open the topological entity, for example `HighFidelityBladeSurface`, in the object editor, then make CAD assignments:

- By using the *CAD Entity Selector*  icon (beside the **Location** field) to open the **CAD Region Entities** dialog box, which you can use to select CAD objects, and/or
- By typing a comma-delimited list of CAD object names into the **Location** field.

- Alternative Method — Right-click the appropriate CAD object (or optionally a selection of multiple CAD objects made using **Shift/Ctrl**) and use the **Assign Family To Topology Instance/Assign Geometry Only To Topology Instance** shortcut menu command.

- Repeat the previous step as needed until the required entity definitions have been completed. For example, for a complex blade end topological entity, assign CAD families as needed to define two or three of the child objects: `HighFidelityBladeSurface`, `HighFidelityHubSurface`, `HighFidelityShroudSurface`.

---

### Note:

CAD objects include families and their constituent primitives. When you assign a CAD family to a topological entity, the entire family is associated with the topological entity. Any upstream geometry updates to the family will be propagated to the assigned topological entity.

---

The only available high-fidelity geometric feature is the complex blade end. For details, see [Complex Blade End](#) (p. 223).

### 12.3.1. Complex Blade End

To add a `ComplexBladeEnd` topological entity, right-click `Topological Entity Instances > Blade Set > [blade name]` and select **Insert Complex Blade End**.

The `ComplexBladeEnd` topological entity requires CAD entity definitions for two or three of its three child objects:

- `HighFidelityBladeSurface` (required)

This entity must be assigned CAD objects that cumulatively cover all blade surfaces that are exposed to the fluid flow in the blade passage.

- `HighFidelityHubSurface` (optional)

This entity, if present, must be assigned CAD objects that cumulatively cover all parts of the hub surface that are exposed to the fluid flow in the blade passage.

- `HighFidelityShroudSurface` (optional)

This entity, if present, must be assigned CAD objects that cumulatively cover all parts of the shroud surface that are exposed to the fluid flow in the blade passage.

To control whether there is a hub complex blade end, a shroud complex blade end, or both, you can edit the `ComplexBladeEnd` topological entity:

- In the tree, open the object editor for the `ComplexBladeEnd` object either by double-clicking the object or by right-clicking the object and then selecting **Edit Complex Blade End Topology**.
- In the object editor, on the **Topology Selection** tab, select **Hub Complex Blade End**, **Shroud Complex Blade End**, or both, to specify which blade ends are complex blade ends.
- Click **Apply**.

Child objects of `ComplexBladeEnd` are added or deleted accordingly.

Alternatively, you can:

- Insert a complex blade end child object by right-clicking the `ComplexBladeEnd` object and selecting **Insert Hub Complex Blade End** or **Insert Shroud Complex Blade End**.
- Delete a complex blade end child object by right-clicking it and selecting **Delete Selected Entity**.

To delete a `ComplexBladeEnd` topological entity, right-click it and select **Delete Complex Blade End**.

## 12.4. Using ATM3D or ATM v2

---



In preparation for generating a mesh that conforms to a high-fidelity geometric feature, for example a complex blade end, you need to enable ATM3D or ATM v2. This can be done as follows:

1. In the **Mesh** workspace, open `Topology Set`.
2. Set **Mesh Generation Mode** to ATM3D or ATM v2.
3. Click **Apply**.

Before setting up high-fidelity geometry, you can optionally generate a mesh for the low-fidelity geometry, adjusting the topology and mesh settings as required. The mesh for the low-fidelity geometry normally requires less time to generate than the mesh for the high-fidelity geometry, enabling you to save time while adjusting mesh settings for the mesh outside the high-fidelity regions.

There is a pair of **Display** menu commands and a pair of corresponding toolbar icons for selecting the type of geometry, high-fidelity or low-fidelity, for which to generate a mesh.

The **Display** menu commands are **Use Mesh for Low-Fidelity Geometry** and **Use Mesh for High-Fidelity Geometry**; only one command appears at a time depending on the type of geometry currently set for meshing. For example, if the displayed command is **Use Mesh for Low-Fidelity Geometry** then high-fidelity geometry is currently set for meshing, and using this command would switch to low-fidelity geometry for meshing.

The toolbar icons *Use Mesh for Low-Fidelity Geometry*  and *Use Mesh for High-Fidelity Geometry*  operate in the same way.

These commands and icons affect any existing mesh, any mesh subsequently generated, and any mesh subsequently saved.

With the type of geometry for meshing set appropriately, you can generate the topology and 3D mesh by right-clicking **Topology Set** and turning off **Suspend Object Updates**. You can subsequently toggle the type of geometry for meshing as needed.

## 12.5. Working with Meshes for High-Fidelity Geometry in TurboGrid's Mesh Workspace

---

With the geometry loaded and with the topology of the high-fidelity geometric features recognized by TurboGrid (as reflected in the **Geometry** workspace), you can use the **Mesh** workspace to specify meshing details for the high-fidelity geometric features.

In the **Mesh** workspace tree, the high-fidelity feature objects are listed under three locations, as detailed in the following sections:

#### 12.5.1. Geometry Branch

#### 12.5.2. Mesh Data Branch

#### 12.5.3. 3D Mesh Branch

### 12.5.1. Geometry Branch

The high-fidelity feature objects under the `Geometry` branch each have a visibility check box that can be used to control visualization of the high-fidelity geometry.

The visibility of the low-fidelity geometry is controlled by the usual applicable visibility check boxes.

The aforementioned check boxes operate independently.

### 12.5.2. Mesh Data Branch

The high-fidelity feature objects under the `Mesh Data` branch each have a visibility check box that can be used to control visualization of a partial outline of the unstructured mesh block surrounding the applicable high-fidelity geometric feature.

These objects can be edited in the object editor to control the applicable unstructured mesh block.

Settings related to particular `Mesh Data` objects are described in:

#### 12.5.2.1. Setting Descriptions for Complex Blade End Mesh Data Objects

### 12.5.2.1. Setting Descriptions for Complex Blade End Mesh Data Objects

On the **High-Fidelity Geometry** tab:

- The **Enable High-Fidelity Feature** check box provides a way to effectively ignore the applicable complex blade end feature when generating a mesh. You might want to do this to save computational time in some situations.

When this check box is selected, the applicable complex blade end can affect the mesh and will affect the mesh provided that the global control (in the **Display** menu and in the toolbar) for geometric fidelity is set to enable high-fidelity geometry.

When this check box is cleared, the applicable complex blade end is ignored during mesh generation, regardless of the global control setting.

- **Mesh Size Factor** controls the overall number of mesh elements used in the mesh near the applicable complex blade end.
- **Growth Rate** controls how quickly the element size changes with respect to distance from a nearby wall in the mesh near the applicable complex blade end. This setting affects boundary layer discretization and indirectly affects the number of mesh elements used in the mesh near the applicable complex blade end.

On the **Complex Blade End** tab:

- The **Span Location** setting specifies the normalized span location that approximately bounds the mesh for the complex blade end. The actual boundary is the nearest constant- $\kappa$  surface in the structured mesh, with an integer value of variable  $\kappa$ .

---

**Note:**

If the span varies significantly from leading to trailing edge, generation of the unstructured mesh might benefit from changes to:

- The interface element size, which is controlled by the **Interface Element Size** settings on the **Hub Complex Blade End** tab (or **Shroud Complex Blade End** tab) for the Mesh Data object. For details, see [Spanwise Distribution Parameters \(for Complex Blade Ends\)](#) (p. 175).
  - The interface span position, which is controlled by the **Span Location** setting.
- 

- The **Elements Per Gap** setting applies to the portion of the mesh associated with the complex blade end. This setting controls the number of elements used in close proximity to gaps.
- The **Edge Length Refinement Factor** setting controls mesh refinement at feature edges. This setting can vary from 0 to 1, with 0 tending towards a coarse mesh and 1 tending towards a fine mesh.

A target mesh element edge length is calculated from the specified value of **Edge Length Refinement Factor** as detailed below. The mesh element size at any given location is influenced by this target element edge length, along with:

- The proximity of nearby surfaces to each other
- The curvature of nearby surfaces
- The size of any nearby feature edges
- Minimum and maximum allowable element edge lengths, determined as the minimum and maximum element edge lengths found in the edges of the tip shell, which is the interface between the structured hex mesh of the passage and the unstructured mesh block for the complex blade end.

Note that the **Edge Length Refinement Factor** setting might have little effect for elements already constrained due to other factors. As an example, a blade with a squealer tip has close proximity between part of the blade tip and the shroud, causing some elements in the tip gap to have an edge length near the aforementioned minimum allowable element edge length. In this case, increasing the **Edge Length Refinement Factor** setting has little effect for those elements in the tip gap.

The target element edge length is calculated using the aforementioned minimum and maximum allowable element edge lengths, and the specified value of **Edge Length Refinement Factor**. Using the terms "target", "min", "max", and "factor":

$$\text{target} = \max\left(\frac{\text{min}}{\text{max}}\right)^{\text{factor}}$$

- To provide immediate feedback on how the complex blade end `Mesh Data` settings are likely to affect the mesh, **Estimated Max. Edge Length** shows the predicted maximum edge length of elements in the vicinity of feature edges in the unstructured mesh block for the complex blade end.
- The **Inflation Layer Count Factors** settings control, within the unstructured mesh block for the complex blade end, the number of inflation layers off the blade or wall as per the setting name: **Blade**, **Hub**, or **Shroud**. The allowed values range from 0 to 1, with the default value being 0.5. Larger values produce a greater number of inflation layers and a greater thickness of the inflation region.
- The **Named Selections to Join in Mesh** settings create named selections that might be required, depending on the case, to generate the unstructured mesh block for the complex blade end.

The tip shell, which is the interface between the structured hex mesh of the passage and the unstructured mesh block for the complex blade end, is required to contact a single 2D region on the blade. If there is no such single region around the blade then you must specify, beside **For Blade**, a list of named selections (CAD families and/or primitives) that can be joined to produce the required single region on the blade.

Similarly, the tip shell is required to contact a single 2D region on the hub/shroud. If there is no such single region on the hub/shroud then you must specify, beside **For Hub** or **For Shroud**, a list of named selections that can be joined to produce the required single region on the hub/shroud.

The resulting 3D Mesh includes regions that represent the joined named selections (trimmed to just the portion in contact with the unstructured mesh).

- When available, the **Tip Shell Selection Options** settings affect the extent of the tip shell. These settings are not available when the complex blade end has high-fidelity features that cross periodic boundaries (see [Complex Blade End Tab](#) (p. 139)).

The tip shell is the interface between the structured hex mesh of the passage and the unstructured mesh block for the complex blade end. If necessary, you can extend the tip shell so that it fully encompasses the high-fidelity geometric features of the complex blade end.

Select **Use Full Passage Topology** to maximize the size of the tip shell, or select a combination of the following four settings: **Extend to Inlet**, **Extend to Outlet**, **Extend to Low Periodic**, **Extend to High Periodic**.

Select **Remesh Extended Surfaces If Possible** to remesh the faces of any and all sides of the tip shell that are relocated due to the extension settings mentioned above, unless there is an inlet and/or outlet block in which case the inlet and/or outlet sides of the tip shell, respectively, are not remeshed. Remeshing is done using triangular faces in an attempt to improve the mesh quality on, and near, the applicable tip shell sides.

---

**Note:**

For some geometry, it might be necessary to shift the topology to one side of the blade or the other so that the topology encompasses the high-fidelity feature. This can be done by specifying the periodic start vertex by editing the `Topology Set` object in the **Command Editor** dialog box.

For example:

```
TOPOLOGY SET:
  ATM Periodic Start Vertex = LERoundedSymmetricStar, 11
END
```

The appropriate setting can be determined by using the **Topology Viewer** (see [Details Tab for the Topology Set Object \(p. 157\)](#)).

- The **Trim Surfaces Before Meshing** check box, which is selected by default, reduces mesh generation time by ignoring parts of the high-fidelity geometry that are deemed to be well outside of the blade passage. Typically this option has a subtle effect on the resulting mesh because the effective geometry differs in extent. This option is not compatible with some geometry, particularly when the geometry protrudes outside of a bounding box placed around the tip shell. If the geometry is incompatible, using this option results in failure to generate a mesh.
- When available, the **Tip Shell Extension Factor** setting affects the extent of the tip shell. This setting is available when the complex blade end has high-fidelity features that cross periodic boundaries (see [Complex Blade End Tab \(p. 139\)](#)).

The tip shell is the interface between the structured hex mesh of the passage and the unstructured mesh block for the complex blade end. If necessary, you can extend the tip shell so that it fully encompasses the high-fidelity geometric features of the complex blade end.

Adjust **Tip Shell Extension Factor** as needed to extend the tip shell in the spanwise direction.

- When available, the **Tip Shell Upstream Extension Factor** setting affects the extent of the tip shell. This setting is available when the complex blade end has high-fidelity features that cross periodic boundaries (see [Complex Blade End Tab \(p. 139\)](#)) and that extend upstream of the inlet end of the passage domain.

As mentioned above, the tip shell is the interface between the structured hex mesh of the passage and the unstructured mesh block for the complex blade end. If necessary, you can extend the tip shell so that it fully encompasses the high-fidelity geometric features of the complex blade end.

Adjust **Tip Shell Upstream Extension Factor** as needed to extend the tip shell in the upstream direction.

The **Tip Shell Downstream Extension Factor** setting is analogous, and applies in the downstream direction.

### 12.5.3. 3D Mesh Branch

The high-fidelity feature objects listed under **3D Mesh** provide the following functions:

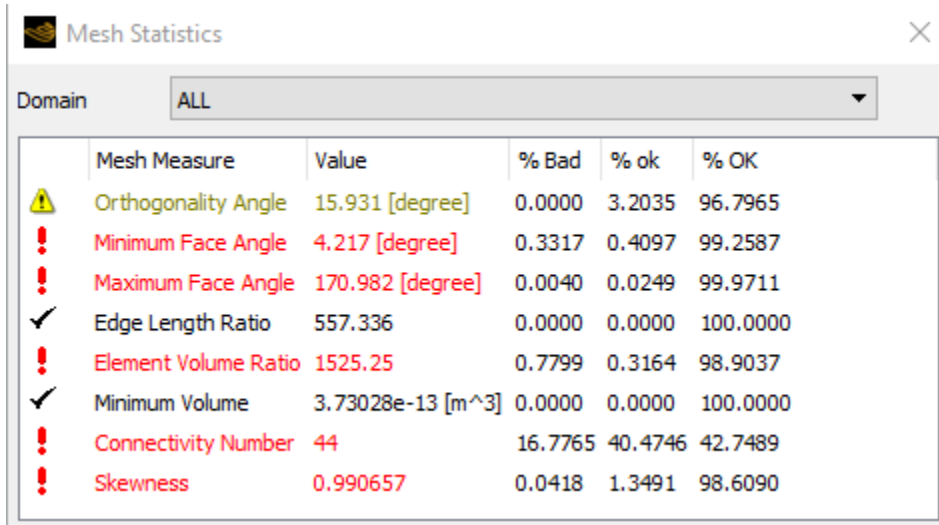
- You can toggle the visibility of a mesh surface associated with a high-fidelity geometric feature by toggling the corresponding visibility check box.
- You can edit any boundary of a high-fidelity geometric feature in order to change its color and/or rendering settings.

For details, see [Surface Groups](#) (p. 70).

## 12.6. Analyzing the Mesh

You can check the mesh statistics by opening **Mesh Analysis** or **Mesh Analysis > Mesh Statistics**.

**Figure 12.3: Typical Mesh Statistics when Meshing High-Fidelity Geometry**



The screenshot shows a window titled "Mesh Statistics" with a "Domain" dropdown set to "ALL". Below is a table with the following data:

	Mesh Measure	Value	% Bad	% ok	% OK
⚠	Orthogonality Angle	15.931 [degree]	0.0000	3.2035	96.7965
❗	Minimum Face Angle	4.217 [degree]	0.3317	0.4097	99.2587
❗	Maximum Face Angle	170.982 [degree]	0.0040	0.0249	99.9711
✓	Edge Length Ratio	557.336	0.0000	0.0000	100.0000
❗	Element Volume Ratio	1525.25	0.7799	0.3164	98.9037
✓	Minimum Volume	3.73028e-13 [m^3]	0.0000	0.0000	100.0000
❗	Connectivity Number	44	16.7765	40.4746	42.7489
❗	Skewness	0.990657	0.0418	1.3491	98.6090

Some of the mesh statistics might have apparent problems based on the usual quality criteria for meshes based on low-fidelity geometry. However, if you double-click any of the statistics that indicate a problem, you will most likely see in the **3D Viewer** that the indicated elements are related to the unstructured mesh surrounding the high-fidelity geometric features. The quality criteria are different for the unstructured mesh. For example, small face angles that would be problematic for a flow solver in the hex mesh might be acceptable in the unstructured mesh.

## 12.7. Saving the Mesh

When you save a mesh, the resulting saved mesh reflects either the high-fidelity geometry or the low-fidelity geometry, whichever is currently set according to:

- The **Display** menu commands **Use Mesh for Low-Fidelity Geometry** and **Use Mesh for High-Fidelity Geometry**, or equivalently
- The corresponding toolbar icons.

To save the mesh, use the **File > Save Mesh** or **File > Save Mesh As** command. For details, see [Save Mesh Command](#) (p. 38).

