Realize Your Product Promise®

Module 05: Mesh Control

Introduction to ANSYS Mechanical

Release 18.0

Module 05 Topics

This module covers introductory topics for meshing and mesh control:

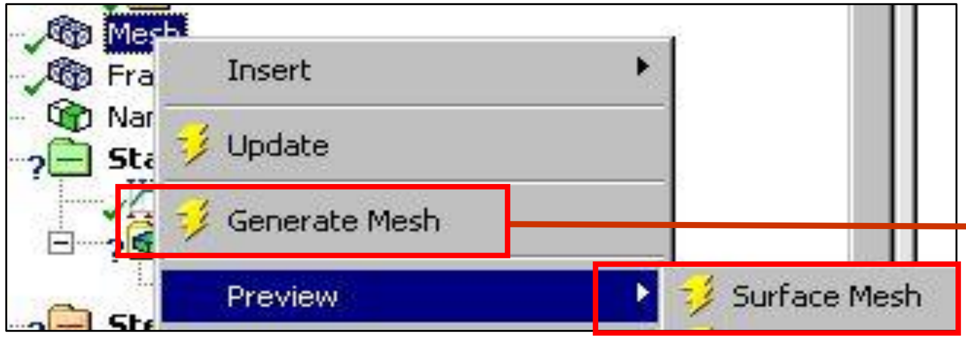
1. Meshing in Mechanical
2. Global Mesh Controls
3. Local Mesh Controls
4. Troubleshooting
5. Virtual Topology
6. Direct Meshing
7. Mesh Quality Criteria
8. Workshop 05.1: Mesh Creation
9. Workshop 05.2: Mesh Control
10. Appendix 05.1: Mesh Quality Criteria
11. Appendix 05.2: Model Assembly

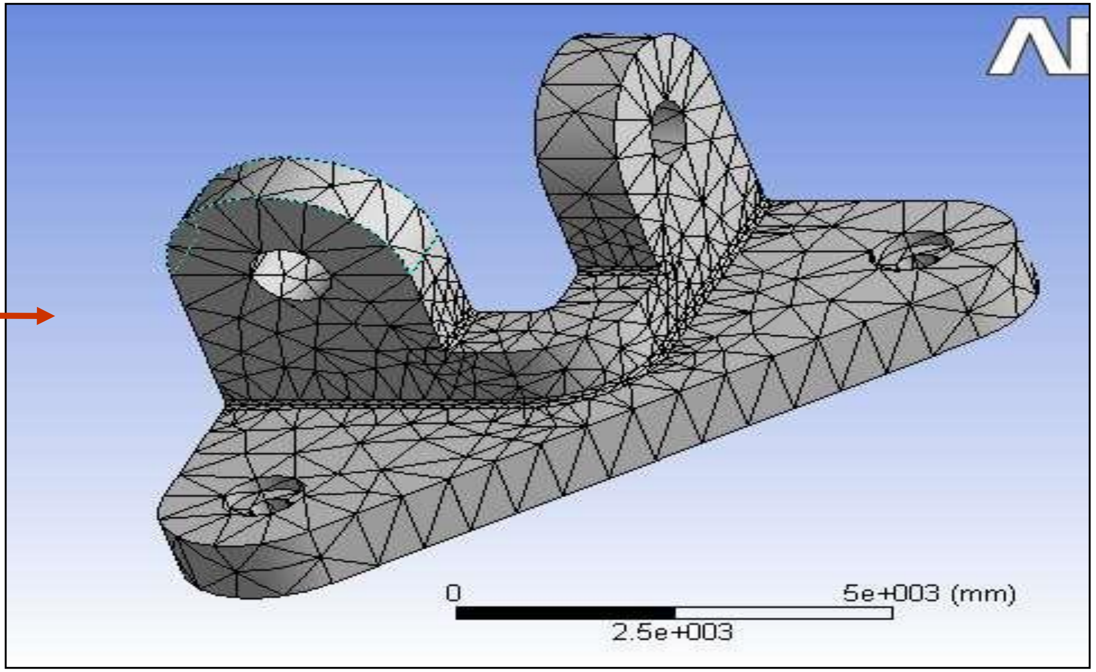
ANSYS

05.01 Meshing in Mechanical

The nodes and elements representing the geometry model make up the mesh:

- A "default" mesh is automatically generated during a solution.
- It is generally recommended that additional controls be added to the default mesh before solving.
- A finer mesh produces more precise answers but also increases CPU time and memory requirements.





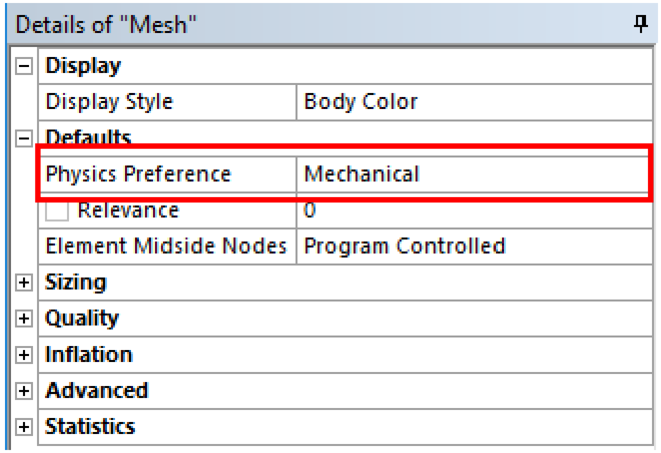
Generate the mesh or preview the surface of the mesh before solving (previewing the surface mesh is faster than generating the entire mesh).

05.02 Global Mesh Controls

Physics-Based Meshing allows the user to specify the metrics used in measuring element quality to be based on the kind of analysis being done.

Physics preferences are:

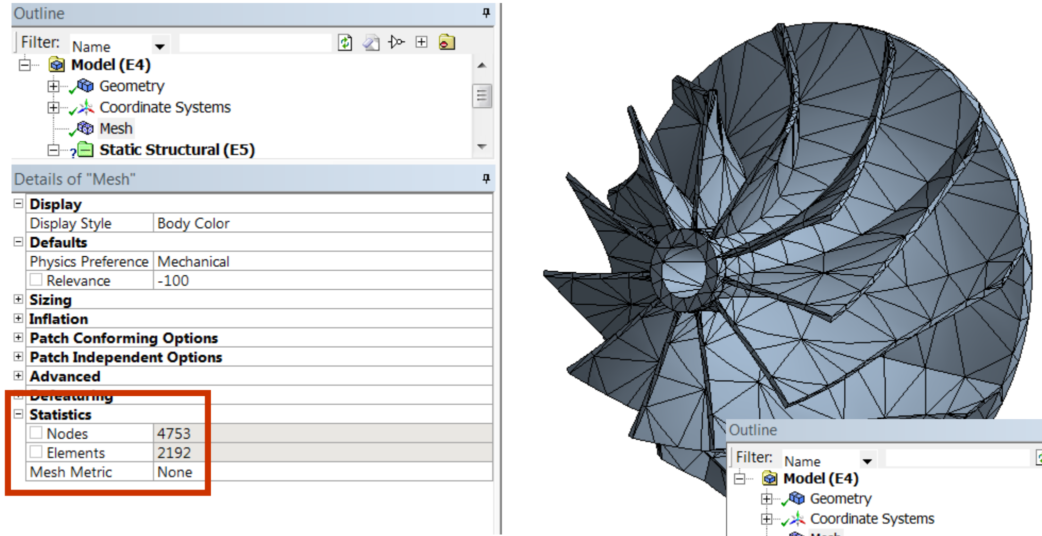
- Mechanical
- Nonlinear Mechanical
- Electromagnetics
- CFD
- Explicit
- Hydrodynamics



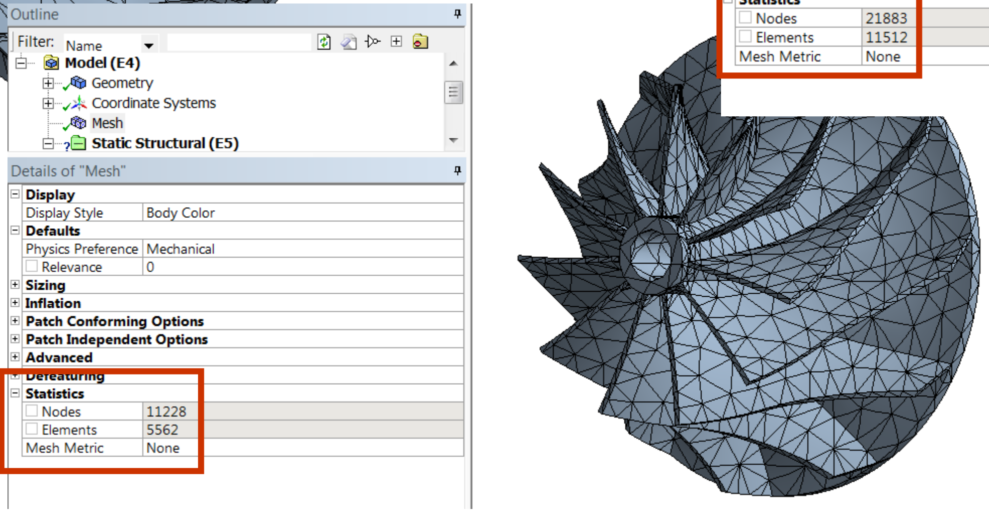
Different analysis types define acceptable or favorable element shapes differently. For this course we limit the discussion to Mechanical.

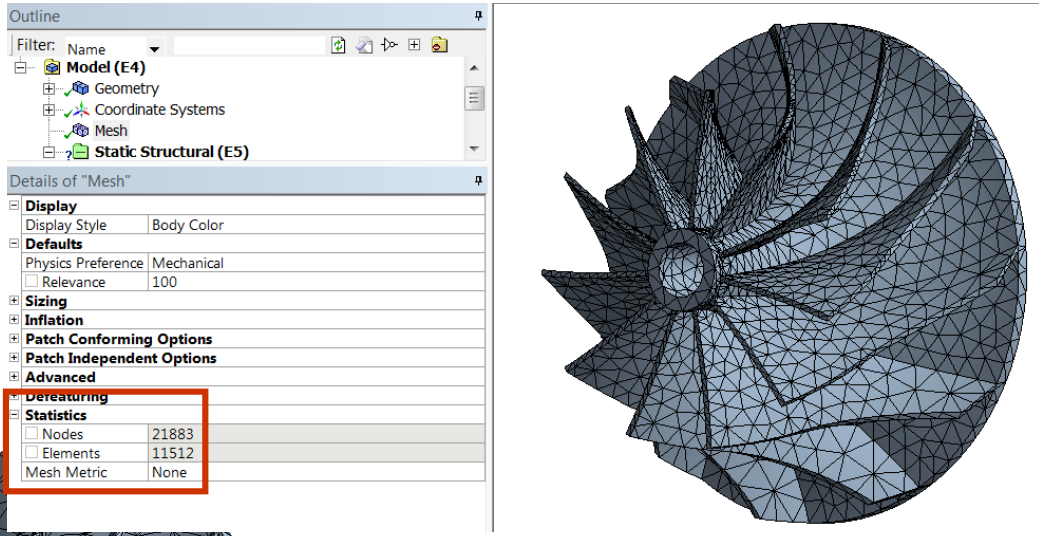
05.02 Global Mesh Controls

- **Relevance is the most basic global size control and is set in the “Defaults” area of the mesh details.**
- **Relevance is set between –100 and +100 (zero = default).**



– Relevance = coarse mesh



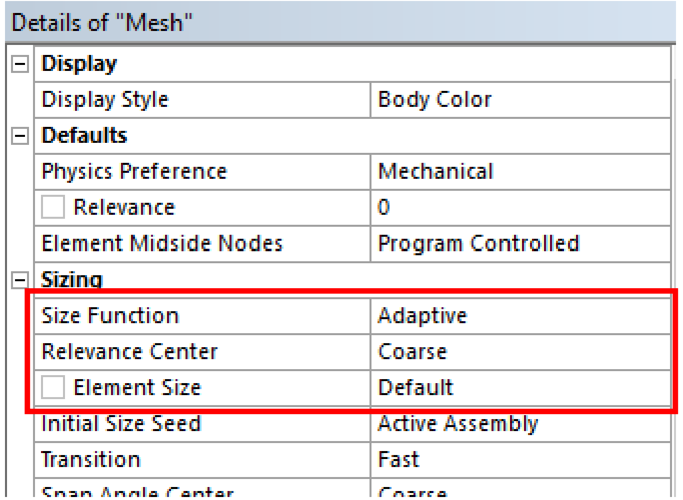


+ Relevance = fine mesh

05.02 Global Mesh Controls

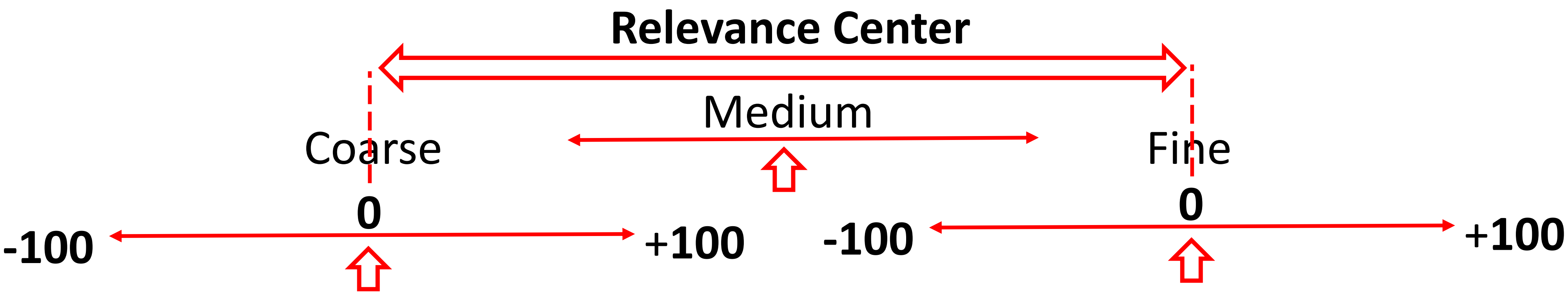
Sizing Control:

- The points below assume that Size Function is set to “Adaptive.” (Size Function is discussed further on the next slide.)
- Relevance Center sets the midpoint of the “Relevance” global control.
- Element Size defines the maximum element size used for the entire model.
- For most static structural applications, the default values for the remaining global controls are usually adequate.



Details of "Mesh"

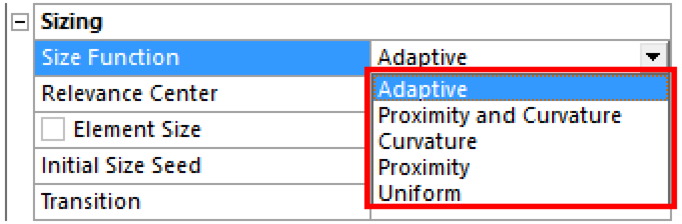
Display	
Display Style	Body Color
Defaults	
Physics Preference	Mechanical
Relevance	0
Element Midside Nodes	Program Controlled
Sizing	
Size Function	Adaptive
Relevance Center	Coarse
Element Size	Default
Initial Size Seed	Active Assembly
Transition	Fast
Span Angle Center	Coarse



05.02 Global Mesh Controls

Size Function provides several advanced automated control options over the global mesh sizing. It is controlled in the Mesh Details view.

- **While many of these controls are beyond the scope of an introductory course, we'll review some of the advanced size controls here. As stated earlier, linear static analysis types usually do not share the same meshing demands as more advanced analysis types, and so advanced size controls are not often necessary.**
- **"Adaptive" is the default setting for the Mechanical Physics Preference, and provides the highest level of automation.**
- **Four advanced size functions can be employed: Proximity and Curvature, Curvature, Proximity, and Uniform.**

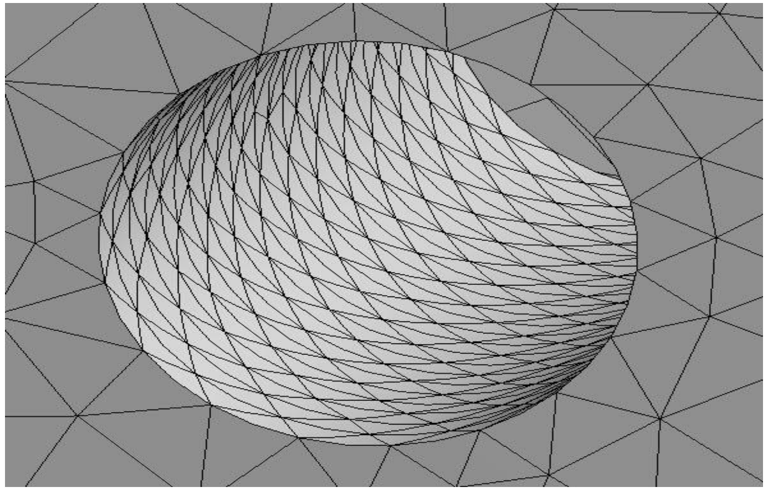


05.02 Global Mesh Controls

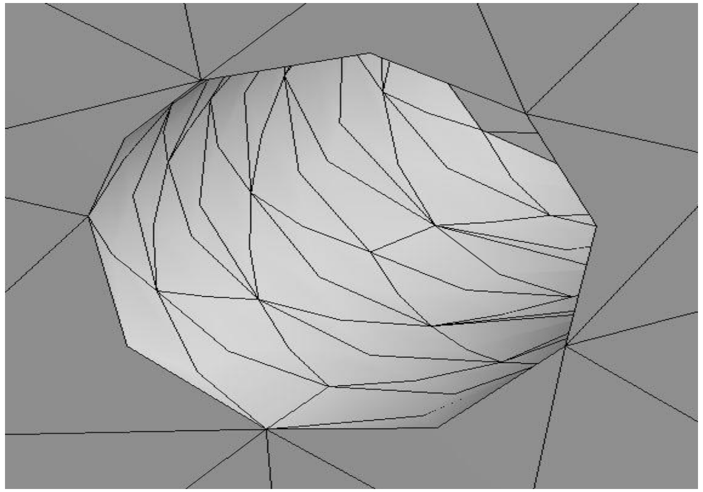
The Uniform Size Function provides control based solely on the user-input parameters Min Size, Max Face Size, Max Tet Size, and Growth Rate.

The Curvature Size Function, as the name implies, is driven by the curvature encountered in the geometry: the higher the curvature, the higher the mesh density. For models dominated by lots of curved features this control provides a way to refine the mesh over much of the model without using numerous local controls. For models composed of mostly straight features, this control will have a lesser impact.

Sizing	
Size Function	Curvature
Relevance Center	Coarse
Initial Size Seed	Active Assembly
Transition	Fast
Span Angle Center	Coarse
Curvature Normal Angle	Default (70.3950 °)
Min Size	Default (0.296840 mm)
Max Face Size	Default (29.6840 mm)
Max Tet Size	Default (59.3680 mm)
Growth Rate	Default (1.850)
Automatic Mesh Based Defeaturing	On
Defeature Size	Default (0.148420 mm)
Minimum Edge Length	5.2108e-002 mm



Curvature On



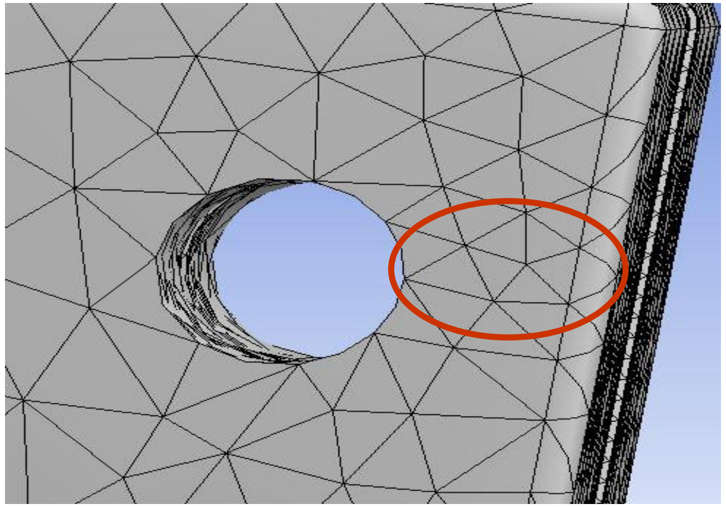
Curvature Off

05.02 Global Mesh Controls

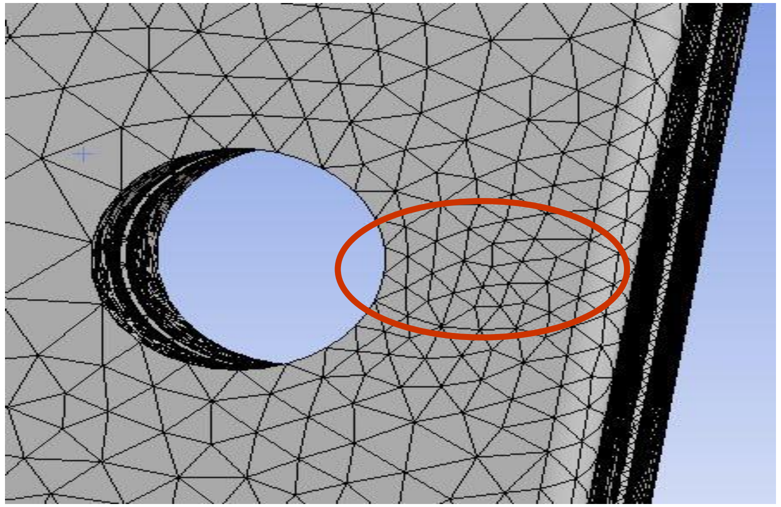
The Proximity Size Function provides a means to control the mesh density in regions of the model where features are located more closely together. In cases where the geometry contains a number of such regions, this control provides a way to refine the mesh over much of the model without using numerous local controls.

As mentioned above, proximity and curvature can be combined.

Sizing	
Size Function	Proximity
Relevance Center	Coarse
Initial Size Seed	Active Assembly
Transition	Fast
Span Angle Center	Coarse
Num Cells Across Gap	Default (3)
Proximity Size Function Sources	Faces and Edges
Proximity Min Size	Default (0.296840 mm)
Max Face Size	Default (29.6840 mm)
Max Tet Size	Default (59.3680 mm)
Growth Rate	Default (1.850)
Automatic Mesh Based Defeaturing	On
Defeature Size	Default (0.148420 mm)
Minimum Edge Length	5.2108e-002 mm



Num Cells = 4



Num Cells = 12

05.02 Global Mesh Controls

Element Midside Nodes:

- Program Controlled (default), Dropped, or Kept (see below).

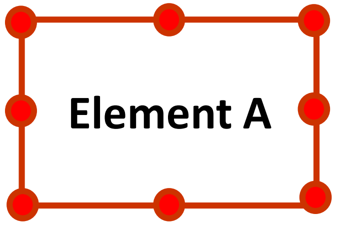
Error Limits for the “Mechanical” Physics Preference:

- Standard Mechanical is recommended for linear stress, modal, and thermal analyses.
- Aggressive Mechanical is recommended for analyses involving large deformations and/or material nonlinearities.

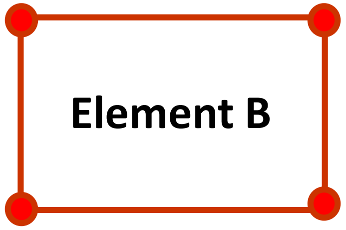
Number of Retries: If poor quality elements are detected, the mesher will try again using finer mesh controls. This setting limits the number of possible retries.

Details of "Mesh"

Display	
Defaults	
Physics Preference	Mechanical
Relevance	0
Element Midside Nodes	Program Controlled
Sizing	
Quality	
Check Mesh Quality	Yes, Errors
Error Limits	Standard Mechanical
Target Quality	Default (0.050000)
Smoothing	Medium
Mesh Metric	None
Inflation	
Advanced	
Number of CPUs for Parallel Part ...	Program Controlled
Straight Sided Elements	No
Number of Retries	Default (4)
Rigid Body Behavior	Dimensionally Reduced
Mesh Morphing	Disabled
Triangle Surface Mesher	Program Controlled
Topology Checking	Yes
Pinch Tolerance	Please Define
Generate Pinch on Refresh	No
Statistics	



Kept



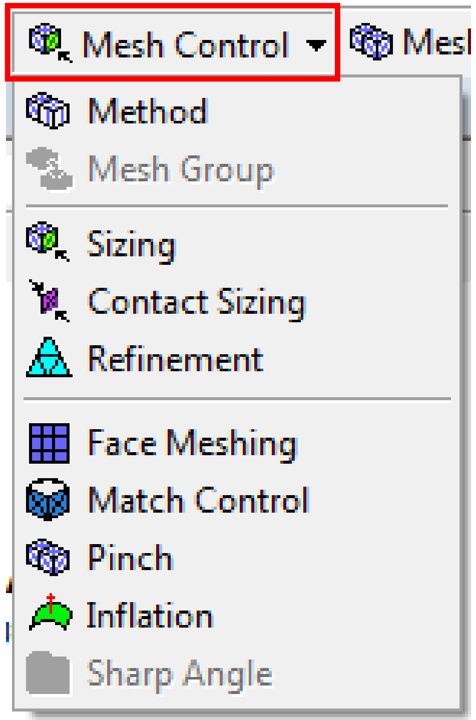
Dropped

ANSYS

05.03 Local Mesh Controls

Local Mesh Controls can be applied to either a Geometry Selection or a Named Selection. These are available only when the mesh branch is highlighted. Some of the available controls include :

- Method Control
- Sizing Control
- Contact Sizing Control
- Refinement Control
- Face Meshing Control
- Inflation Control
- Pinch Control

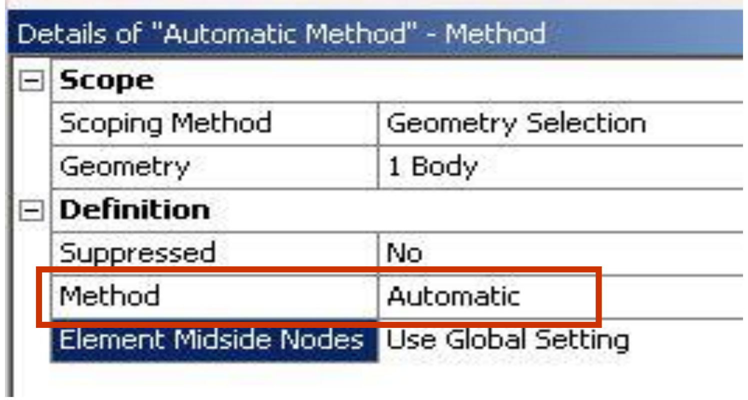


05.03 Local Mesh Controls

Method Control: provides the user with options for the meshing algorithm to be used.

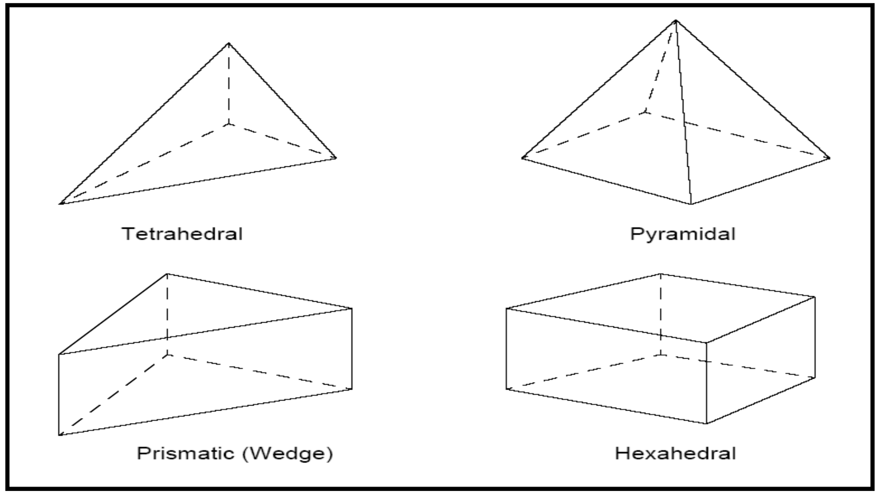
Method = Automatic (default):

- **Body will be swept if possible, which results in either all or most of the elements being hexahedral. If a swept mesh is not possible, the “Patch Conforming Tetrahedron” mesher will be used. Both of these methods are discussed further in subsequent slides.**

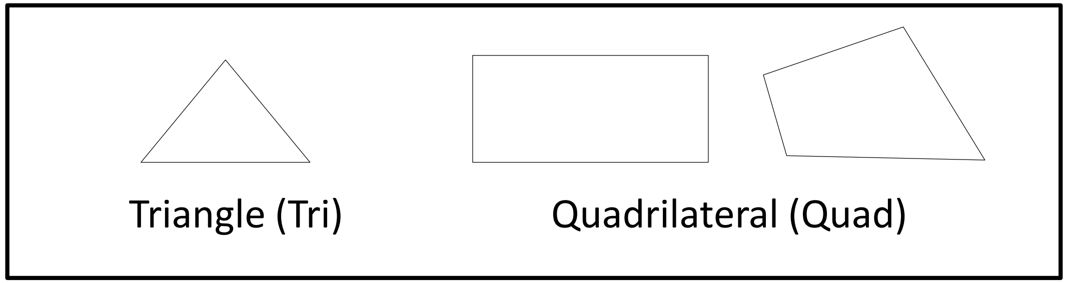


05.03 Local Mesh Controls

- **Mesh Methods for 3D bodies:**
  - Automatic
  - Tetrahedrons
    - Patch Conforming
    - Patch Independent
  - MultiZone
  - Hex Dominant
  - Sweep



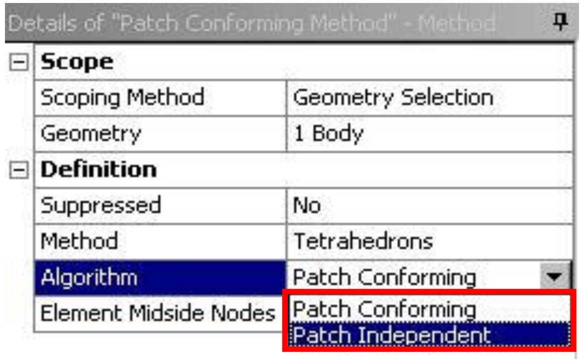
- **Mesh Methods for 2D bodies:**
  - Automatic Method (Quad Dominant)
  - Triangles
  - Uniform Quad/Tri
  - Uniform Quad



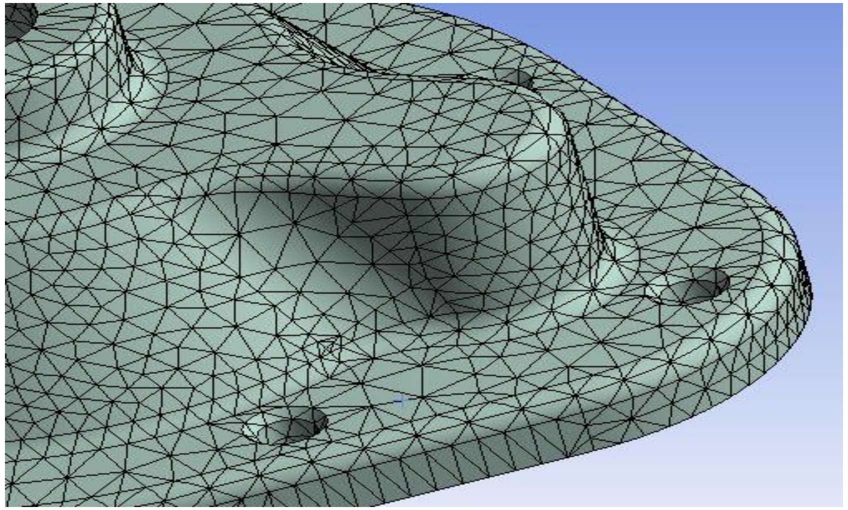
05.03 Local Mesh Controls

Method = Tetrahedrons:

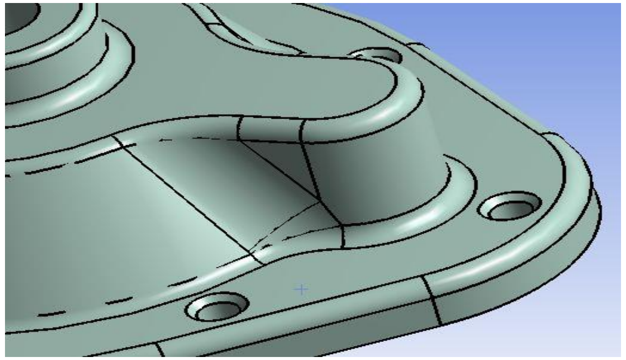
- An all-tetrahedron mesh is generated.
- Can use Patch Conforming or Patch Independent Meshing algorithms.



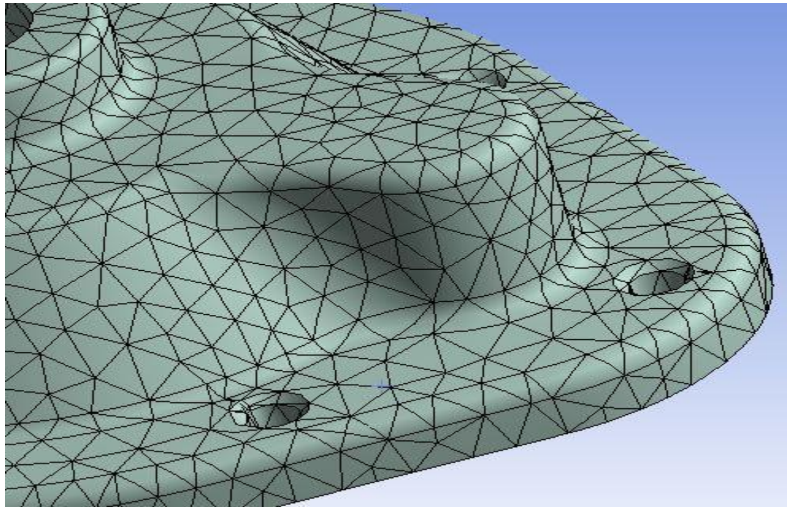
Patch Conforming



Underlying Geometry



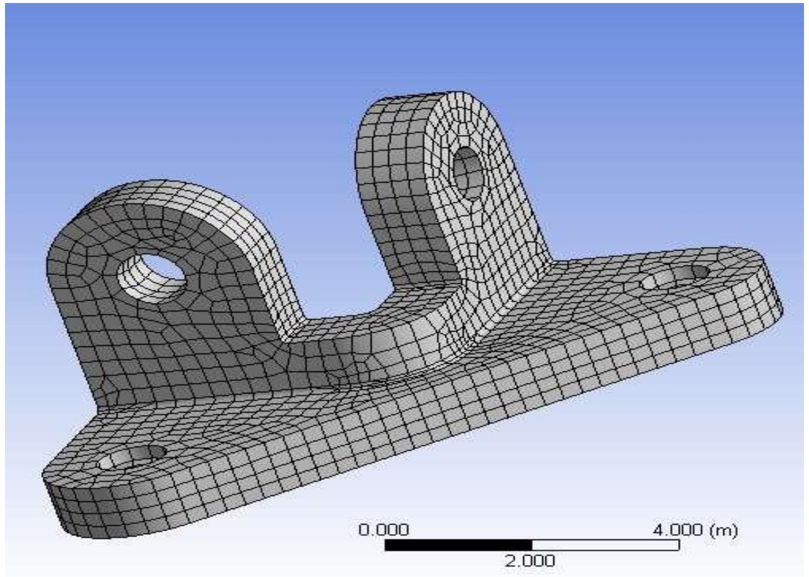
Patch Independent



05.03 Local Mesh Controls

Method = Hex Dominant: Creates a free hex-dominant mesh

- **Recommended for meshing bodies with large interior volumes.**
- **Not recommended for thin or highly complex shapes.**
- **Useful for meshing bodies that cannot be swept.**



Details of "Hex Dominant Method" - Method	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	Hex Dominant
Element Midside Nodes	Use Global Setting
Free Face Mesh Type	Quad/Tri
Control Messages	No

Solid Model with Hex dominant mesh (approximate percentages):

Tetrahedrons – 443 (9.8%)

Hexahedron – 2801(62.5%)

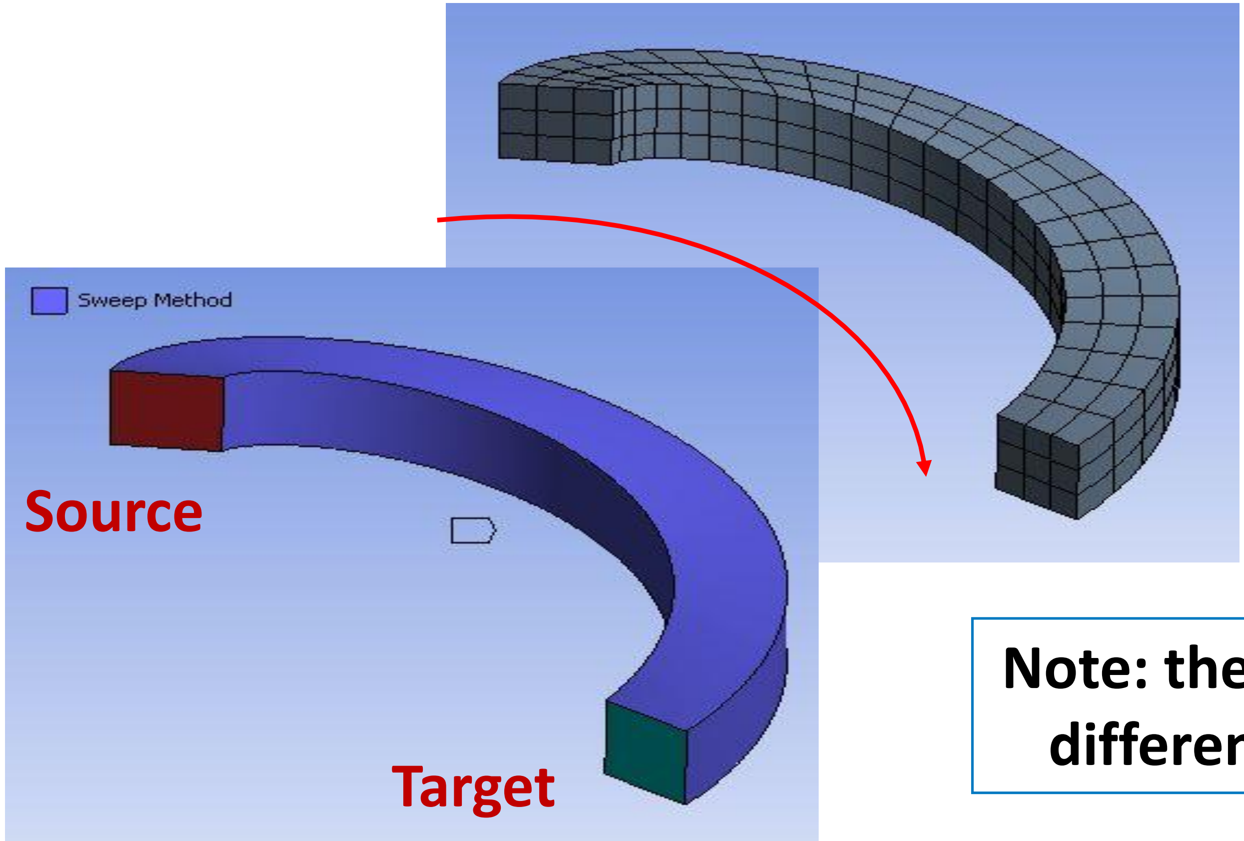
Wedge – 124 (2.7%)

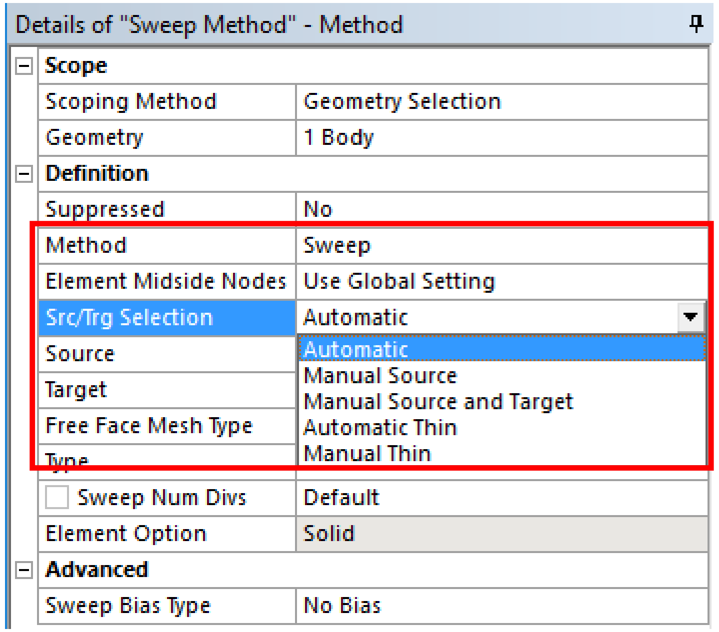
Pyramid – 1107 (24.7%)

05.03 Local Mesh Controls

Method = Sweep (hex and possibly wedge shapes):

- Source/Target Selection: Manually select the start/end faces for sweeping or allow the mesher to choose.
- Can include size controls and/or biasing along sweep.





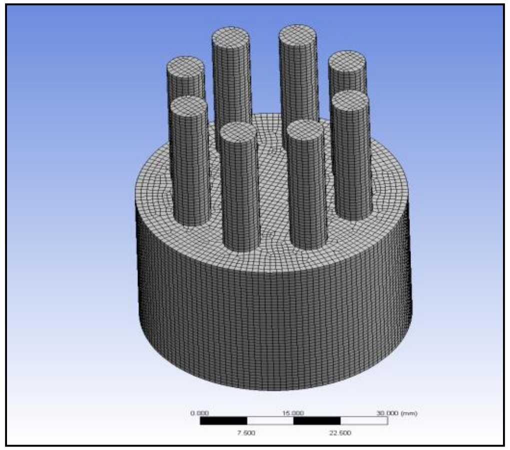
Note: the geometry shown here has six different possible sweep directions.

05.03 Local Mesh Controls

Method = MultiZone:

- Based on blocking approach used in ANSYS ICEM CFD Hexa
- Automatically decomposes geometry
- Uses structured and unstructured blocks
- Can have multiple source and target faces
- Depends on settings of Free Mesh Type
- Structured blocks are meshed with Hexa or Hexa/Prism
- If Free Mesh Type is set to other than Not Allowed, then unstructured blocks are meshed with Tetra, Tetra/Pyramid, Hex Dominant, or Hex Core, based on the selected method
- Program Controlled inflation

MultiZone Mesh



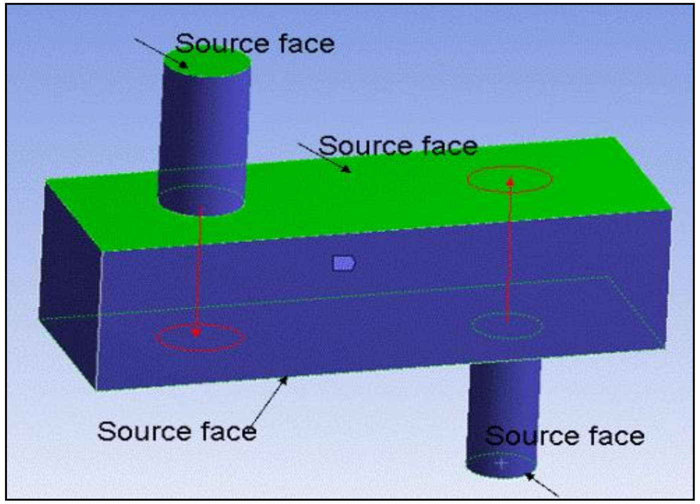
Details of "MultiZone" - Method

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	MultiZone
Mapped Mesh Type	Hexa
Surface Mesh Method	Program Controlled
Free Mesh Type	Not Allowed
Element Midside Nodes	Use Global Setting
Src/Trg Selection	Automatic
Source Scoping Method	Program Controlled
Source	Program Controlled
Sweep Size Behavior	Sweep Element Size
Sweep Element Size	Default
Advanced	
Preserve Boundaries	Protected
Mesh Based Defeaturing	Off

05.03 Local Mesh Controls

Method = MultiZone—Controls:

– Src/Trg Selection = Automatic

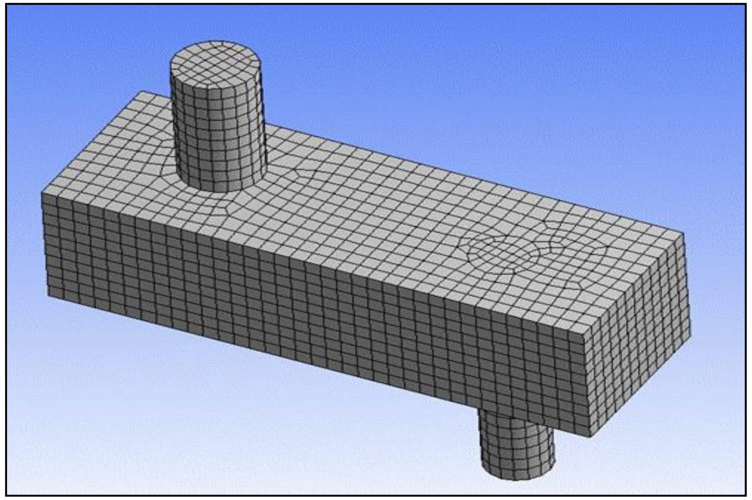


Geometry for MultiZone Meshing

Details of "MultiZone" - Method

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	MultiZone
Mapped Mesh Type	Hexa
Surface Mesh Method	Program Controlled
Free Mesh Type	Not Allowed
Element Midside Nodes	Use Global Setting
Src/Trg Selection	Automatic
Source Scoping Method	Program Controlled
Source	Program Controlled
Sweep Size Behavior	Sweep Element Size
Sweep Element Size	Default

Details View of MultiZone Method

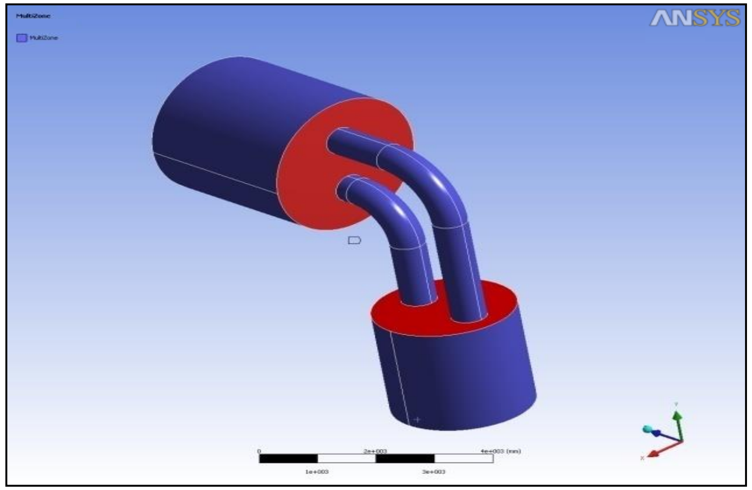


MultiZone Mesh

05.03 Local Mesh Controls

Method = MultiZone—Controls:

– Src/Trg Selection = Manual

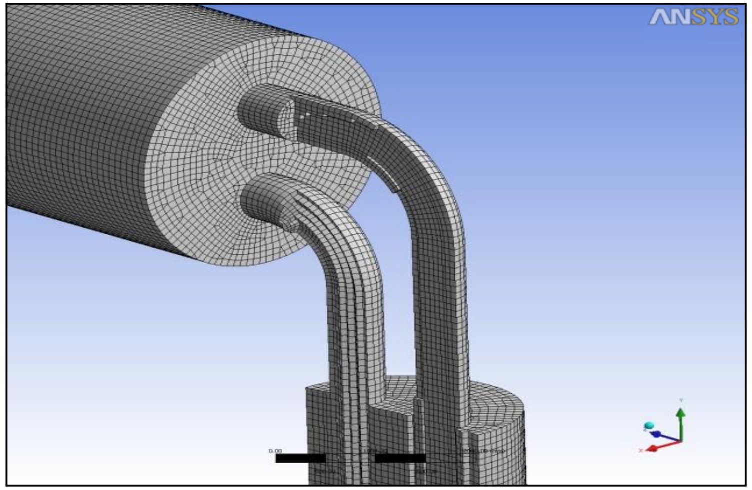


Geometry for MultiZone Meshing

Details of "MultiZone" - Method

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	MultiZone
Mapped Mesh Type	Hexa
Surface Mesh Method	Program Controlled
Free Mesh Type	Not Allowed
Element Midside Nodes	Use Global Setting
Src/Trg Selection	Manual Source
Source Scoping Method	Geometry Selection
Source	4 Faces
Sweep Size Behavior	Sweep Element Size
Sweep Element Size	Default

Details View of MultiZone Method

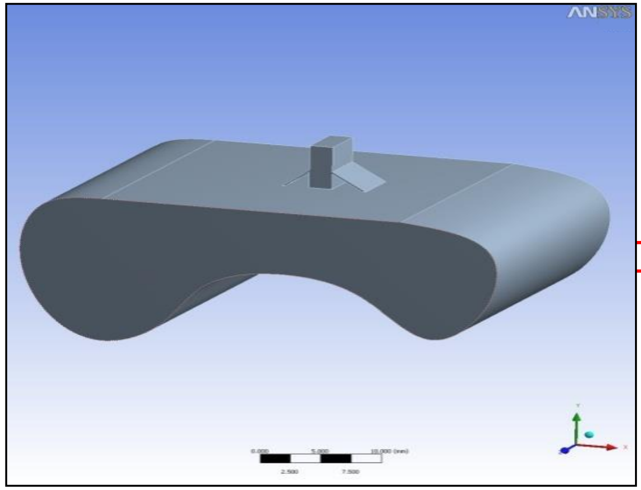


Cut section of MultiZone Mesh

05.03 Local Mesh Controls

Method = MultiZone—Controls:

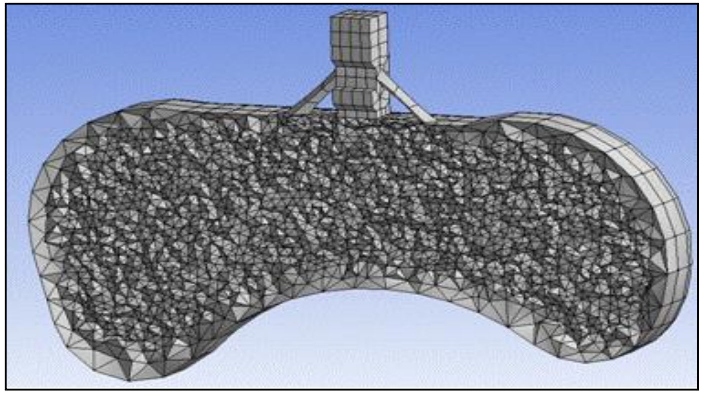
- Free Mesh Type = Tetra, Tetra/Pyramid, Hexa Dominant, or Hexa Core



Geometry

Method	MultiZone
Mapped Mesh Type	Hexa
Surface Mesh Method	Program Controlled
Free Mesh Type	Tetra

Type: Tetra

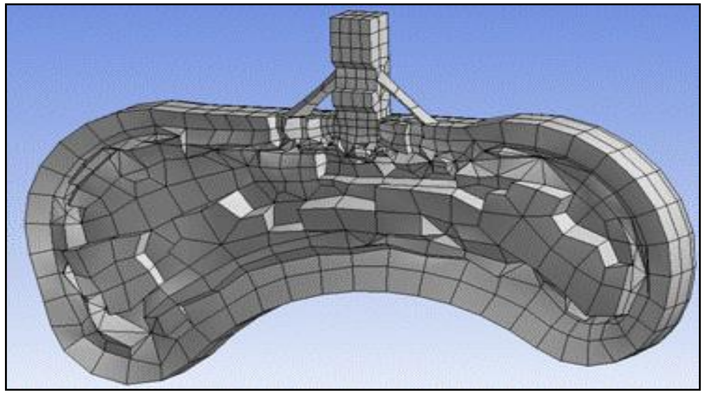


Method	MultiZone
Mapped Mesh Type	Hexa
Surface Mesh Method	Program Controlled
Free Mesh Type	Tetra/Pyramid

Type: Tetra/Pyramid

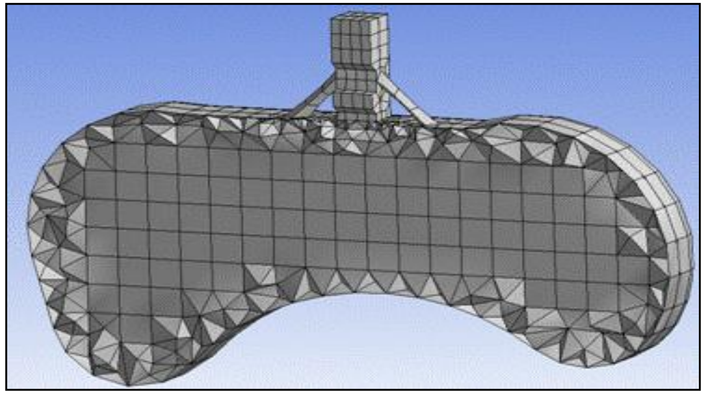
Method	MultiZone
Mapped Mesh Type	Hexa
Surface Mesh Method	Program Controlled
Free Mesh Type	Hexa Dominant

Type: Hexa Dominant



Method	MultiZone
Mapped Mesh Type	Hexa
Surface Mesh Method	Program Controlled
Free Mesh Type	Hexa Core

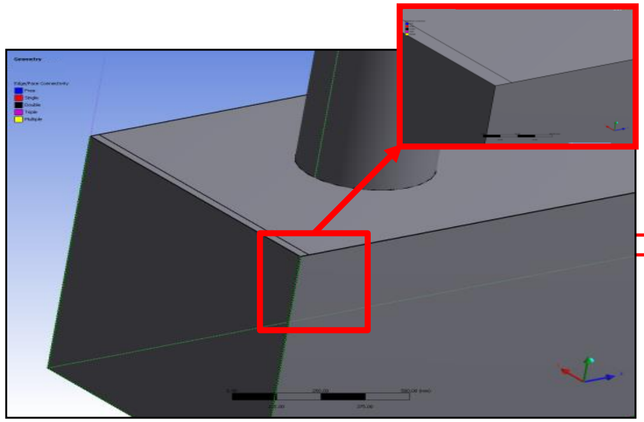
Type: Hexa Core



05.03 Local Mesh Controls

Method = MultiZone—Controls:

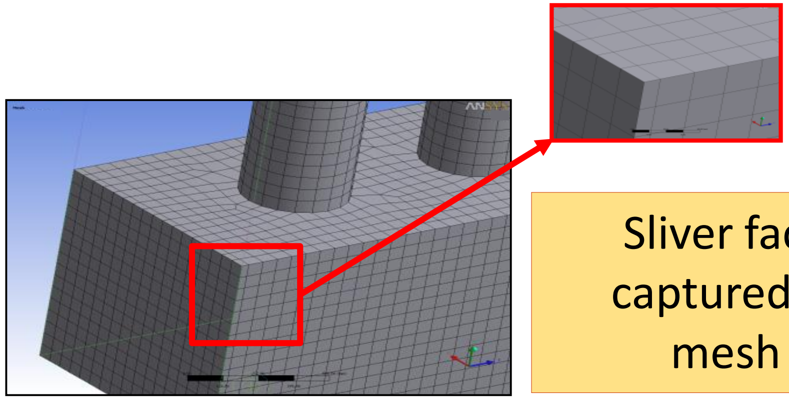
– **Local Defeaturing Tolerance**



Geometry with a sliver face

Advanced	
Preserve Boundaries	Protected
Mesh Based Defeaturing	Off
Minimum Edge Length	304.8 mm

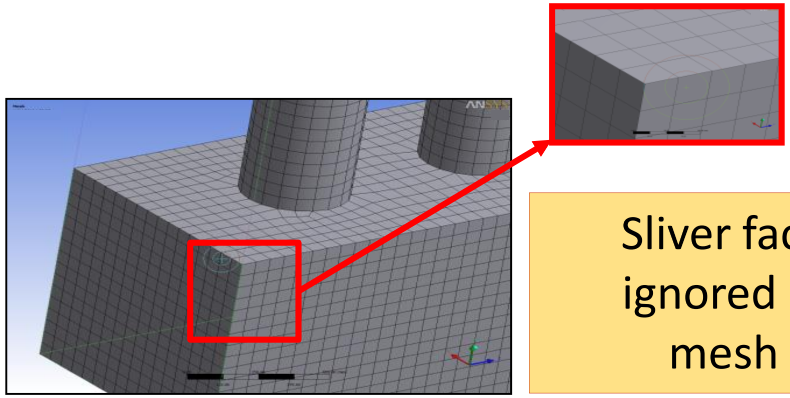
No Defeaturing



Sliver face captured in mesh

Advanced	
Preserve Boundaries	Protected
Mesh Based Defeaturing	On
Defeaturing Tolerance	Default
Minimum Edge Length	304.8 mm

Using Defeaturing



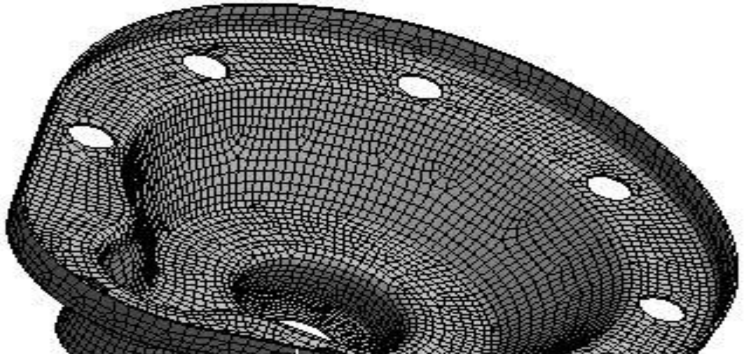
Sliver face ignored in mesh

– **Can be also controlled with global defeaturing tolerance**

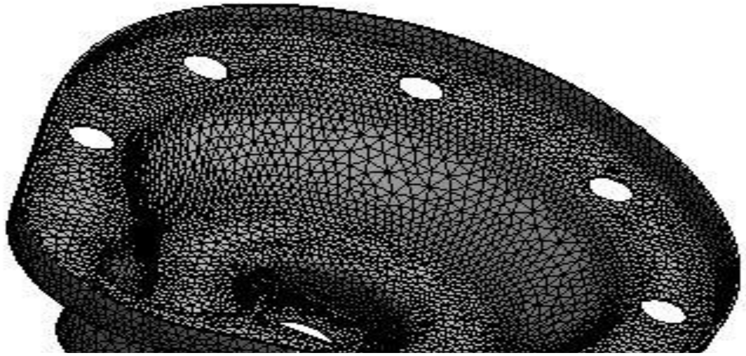
05.03 Local Mesh Controls

Surface Body Methods:

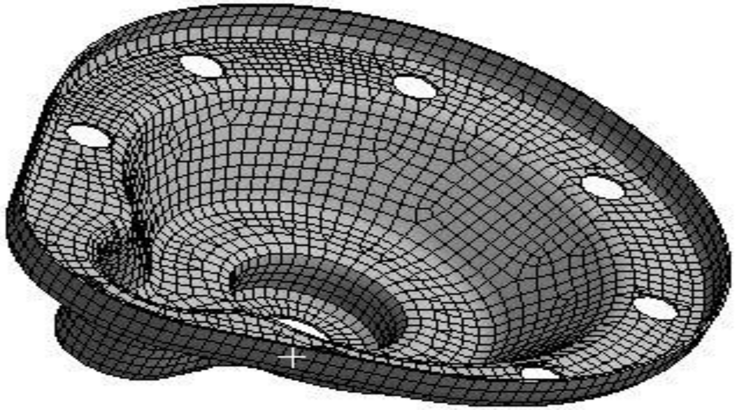
- Quadrilateral Dominant (default): attempts to mesh with as many quadrilateral elements as possible, fills in with triangles.



- Triangles: all triangular shapes are used.



- MultiZone Quad/Tri: Depending on settings, quad or tri shapes are created using a patch independent algorithm.



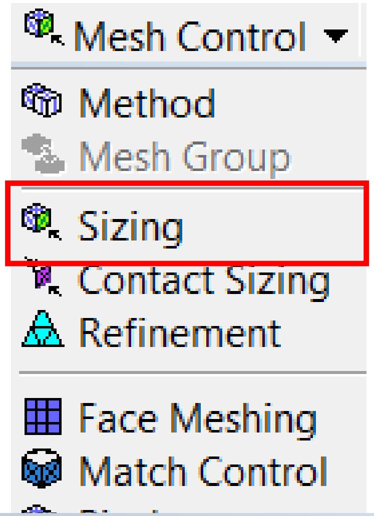
Note: Each method contains a unique set of options in the details allowing additional configuration.

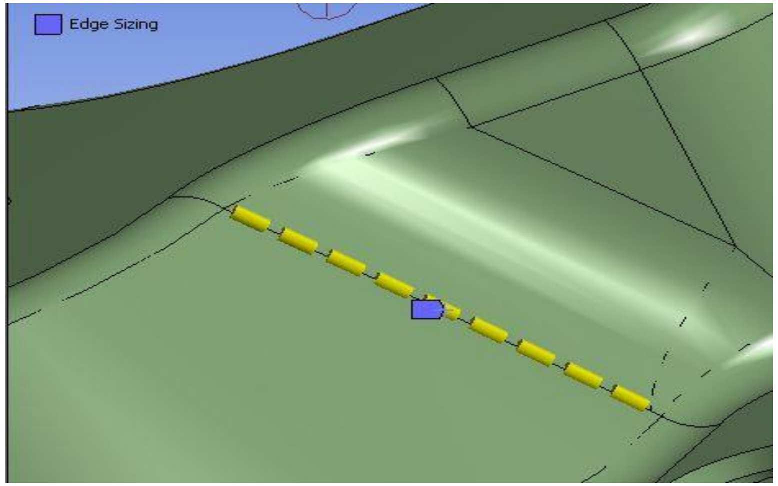
05.03 Local Mesh Controls

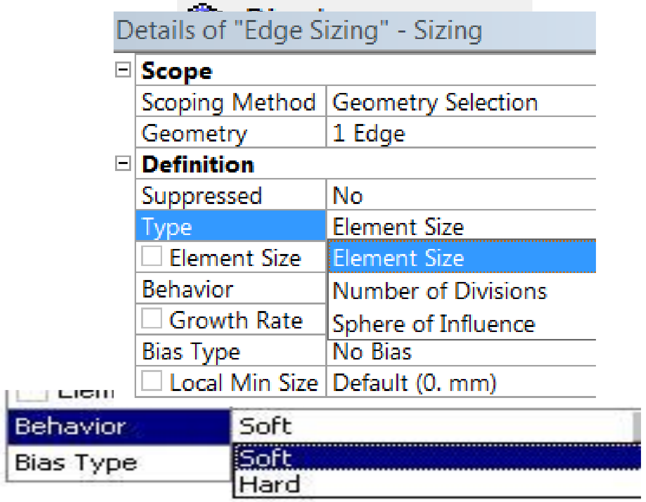
Sizing:

- Element Size (element edge length)
- Number of Divisions
- Sphere of Influence (see next slide)

"Soft" control may be overridden by other mesh controls, "Hard" may not.







Entity	Element Size	# of Elem. Division	Sphere of Influence
Bodies	x		x
Faces	x		x
Edges	x	x	x
Vertices			x

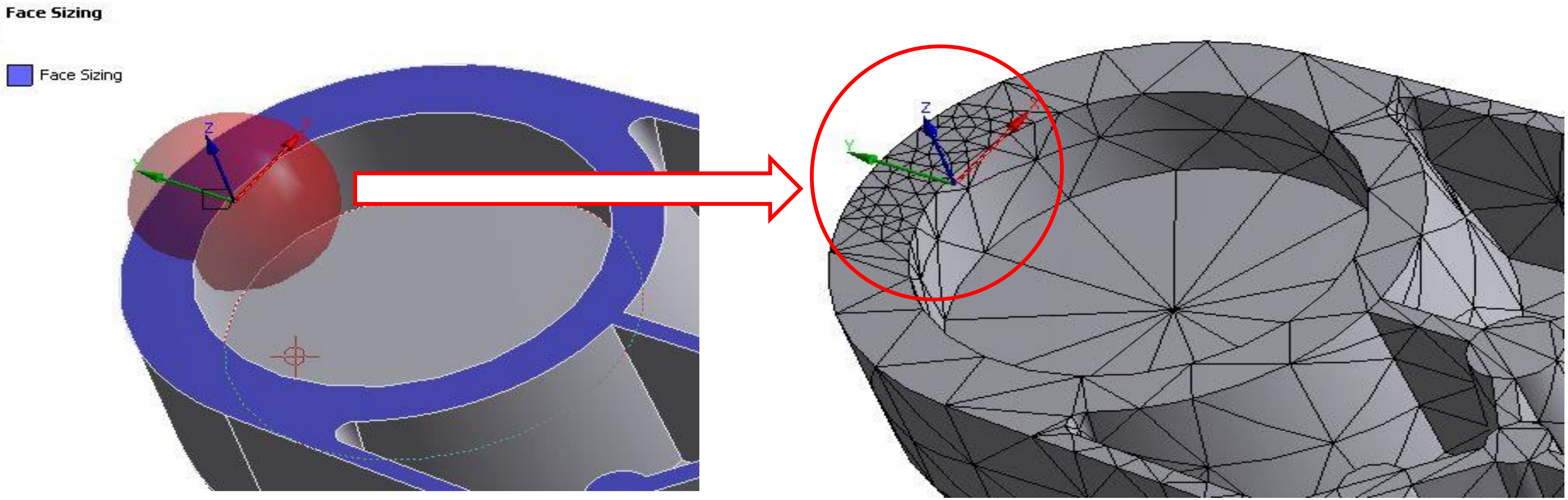
05.03 Local Mesh Controls

Sizing—Sphere of Influence:

- Center is located using a coordinate system.
- All scoped entities within the sphere are affected by size settings.
- Only the portion of the scoped face or body within the sphere is included in the scope of the mesh control.

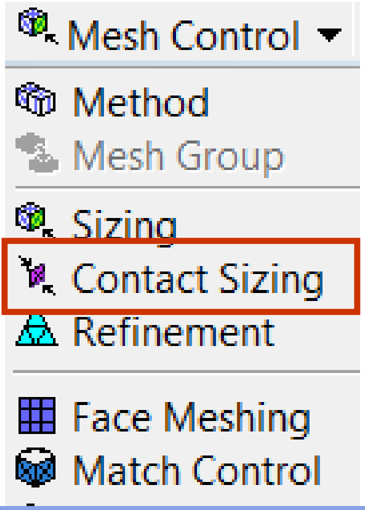
Details of "Face Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Suppressed	No
Type	Sphere of Influence
Sphere Center	Coordinate System
Sphere Radius	15. mm
Element Size	2. mm



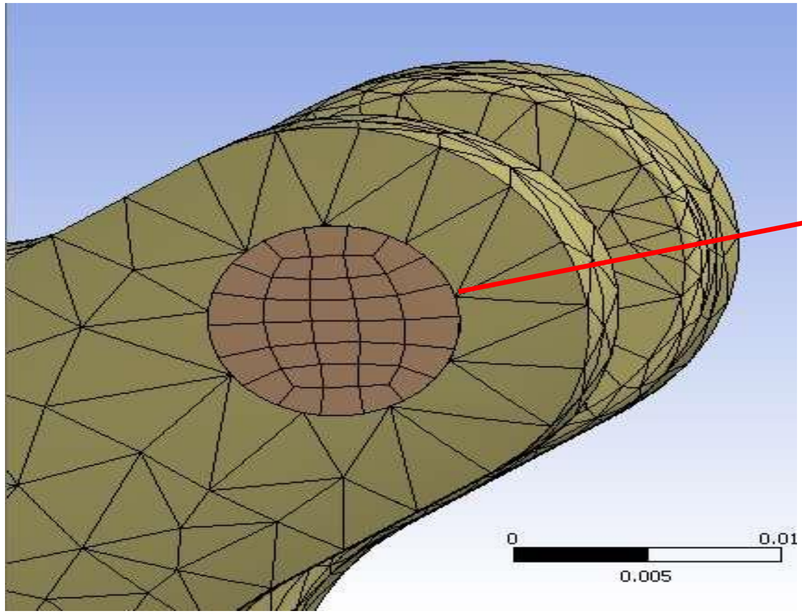
ANSYS

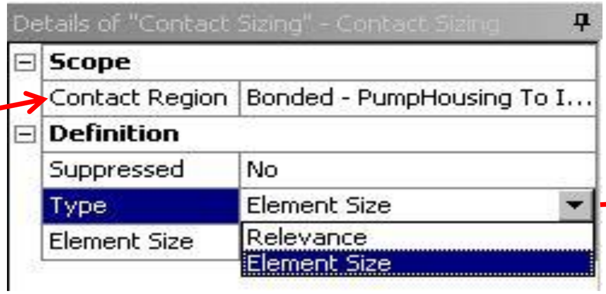
05.03 Local Mesh Controls

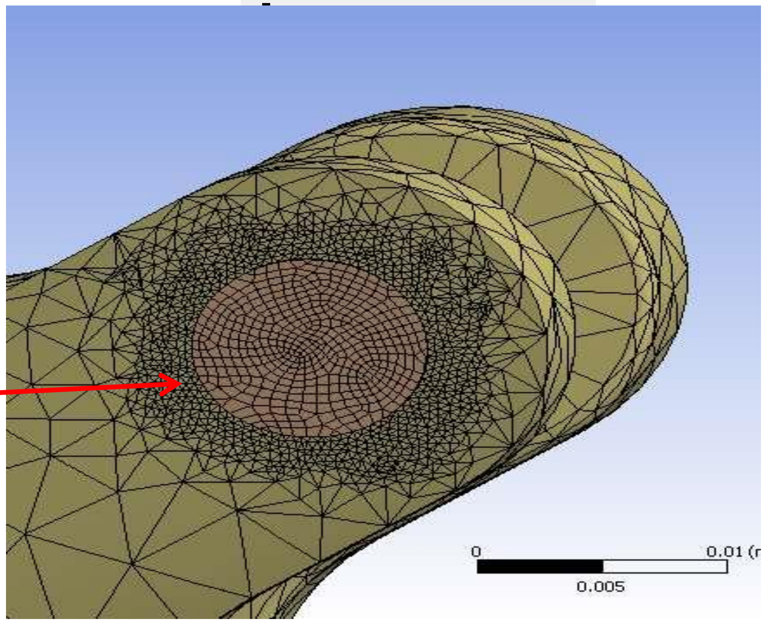


Contact Sizing: generates similarly-sized elements on contact faces for face/face or face/edge contact regions.

- **"Element Size" or "Relevance" can be specified.**
- **Can drag and drop a Contact Region object onto the "Mesh" branch as a shortcut.**



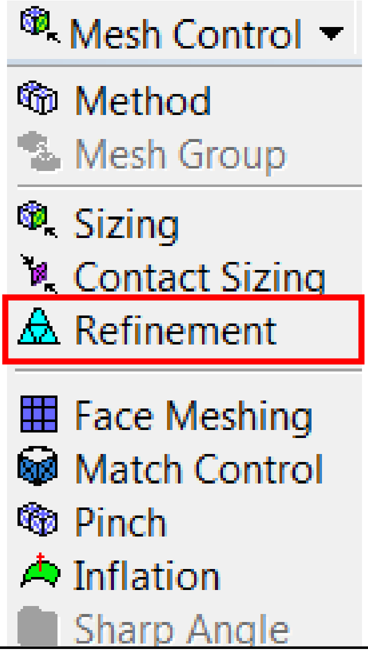


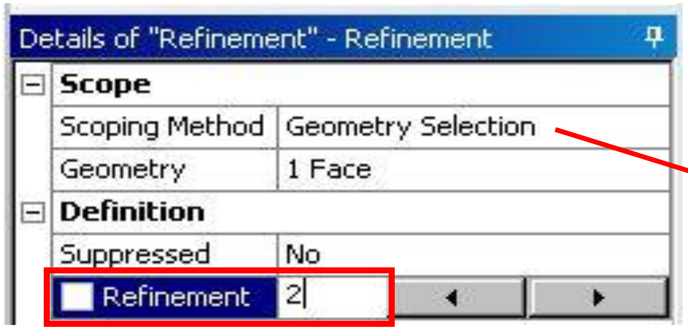


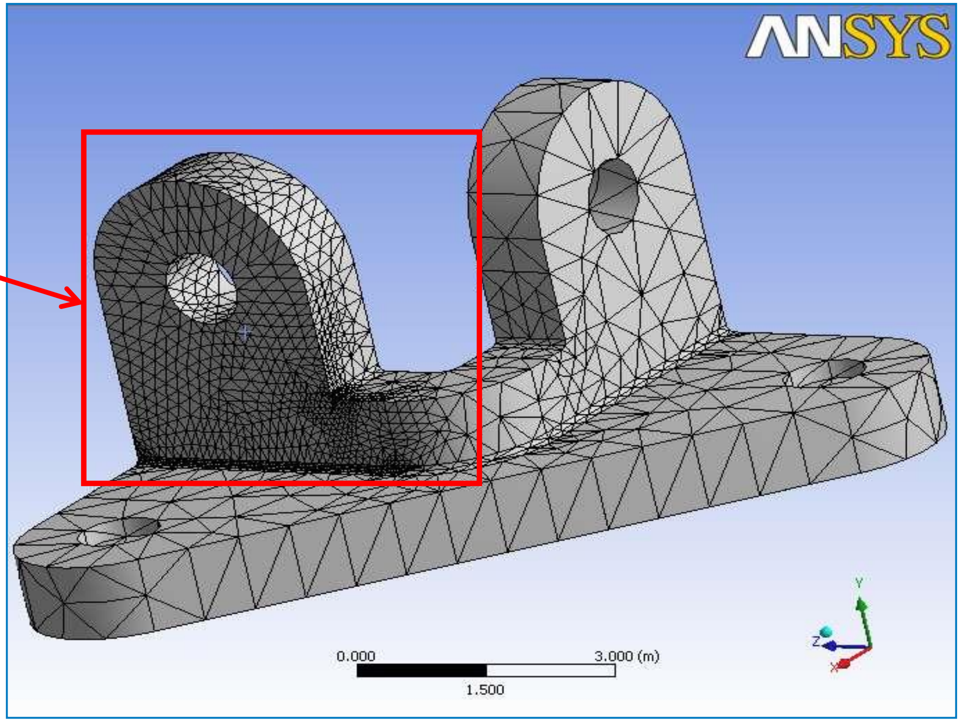
05.03 Local Mesh Controls

Refinement:

– An initial mesh is created using the global and local size settings, then elements are divided at the scoped locations (up to 3 times).



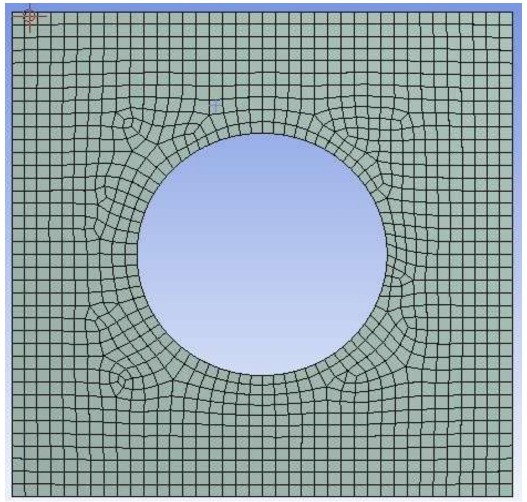


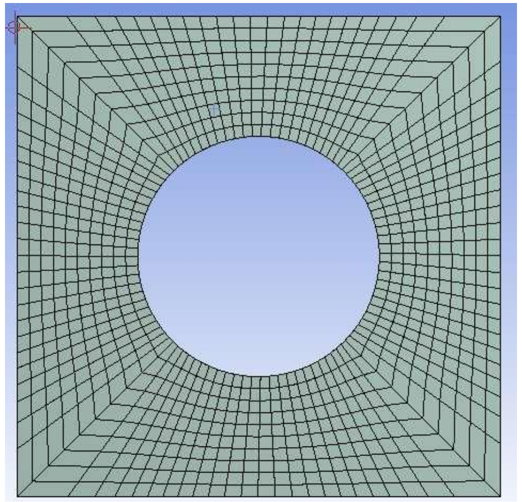


Note: The refinement method generally offers less control and/or predictability over the final mesh, since the initial mesh is simply split. This splitting process may adversely affect other meshing controls.

05.03 Local Mesh Controls

Mapped Face Meshing: generates structured meshes on surfaces



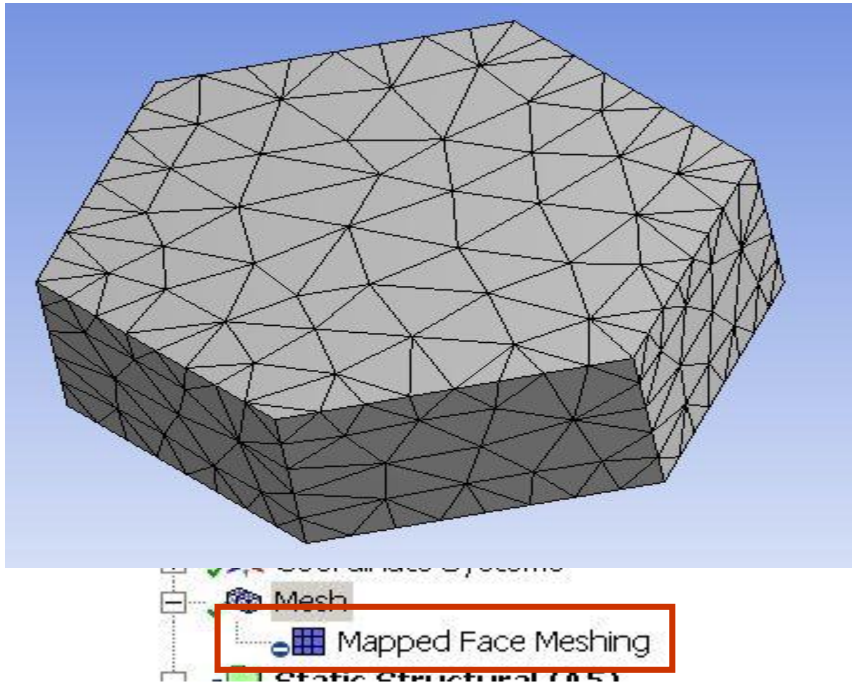


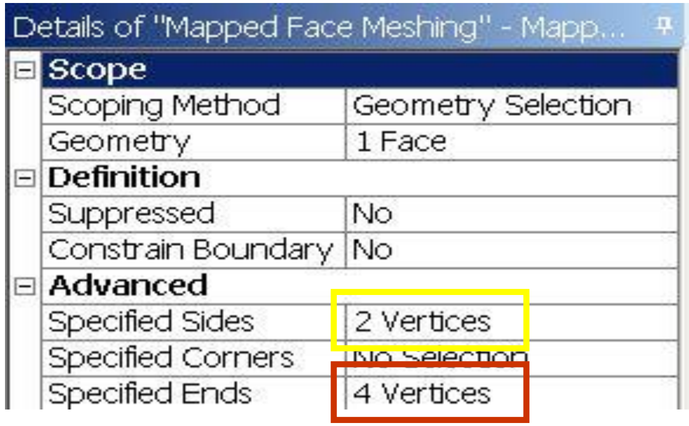
Mapped quad or tri mesh also available for surface bodies.

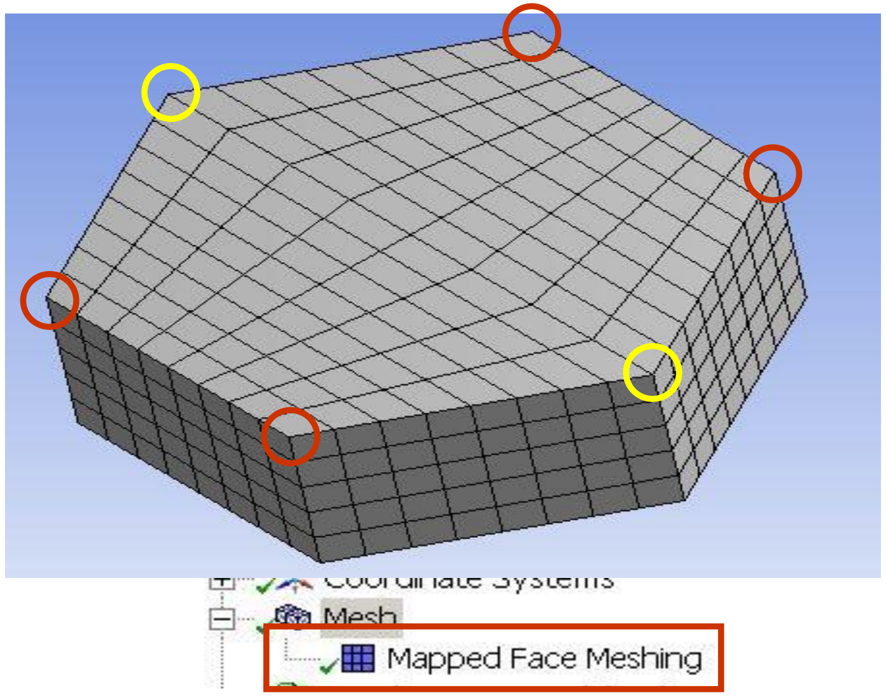
05.03 Local Mesh Controls

Mapped Face Meshing:

- For some geometry mapping will fail if an obvious pattern is not recognized.
- By specifying side, corner or end vertices a mapped face can be achieved.





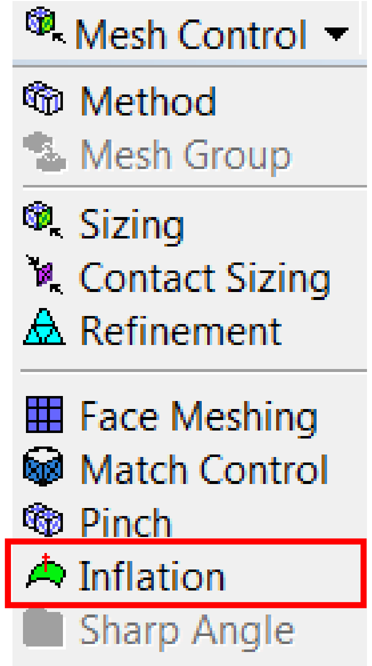


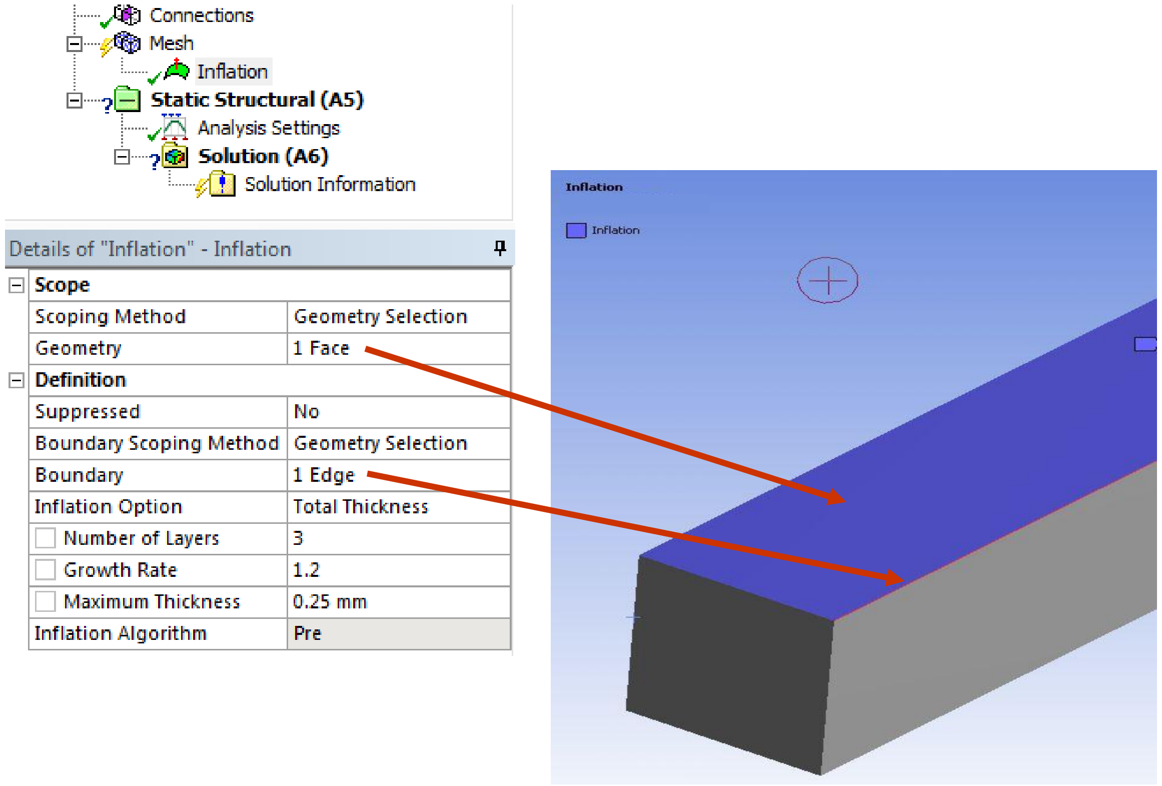
Original mapping failed as indicated by the status icon.

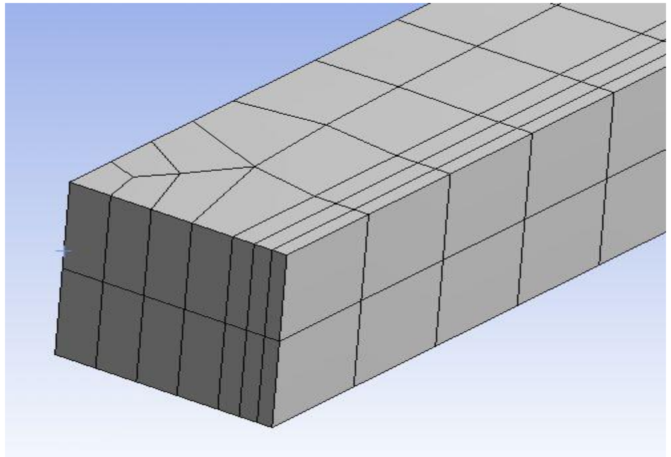
By specifying side and end vertices, the mapped face mesh succeeds.

05.03 Local Mesh Controls

Inflation: useful for adding layers of prismatic elements along specific boundaries.

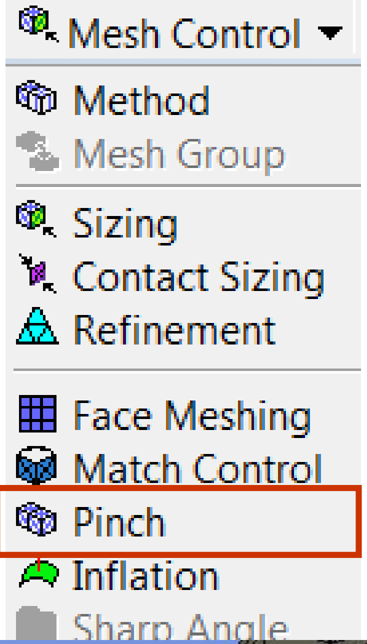






Note: Inflation is used more often in CFD and EMAG applications, but may occasionally be useful for capturing stress concentrations etc. in structural applications.

05.03 Local Mesh Controls

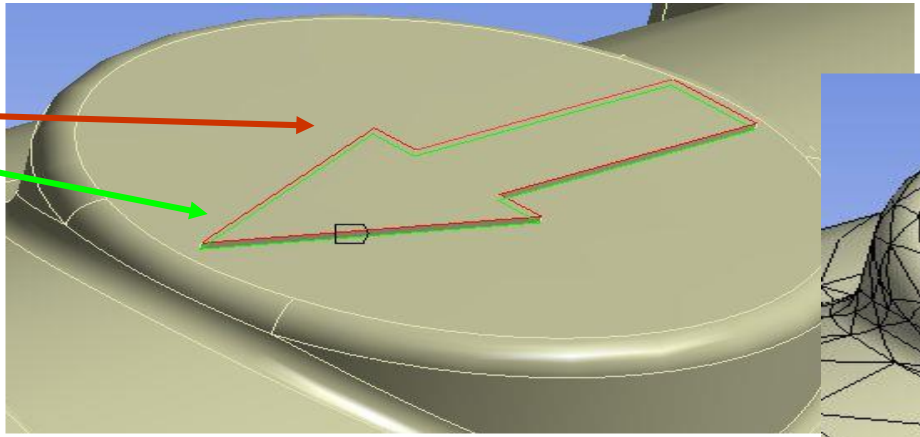


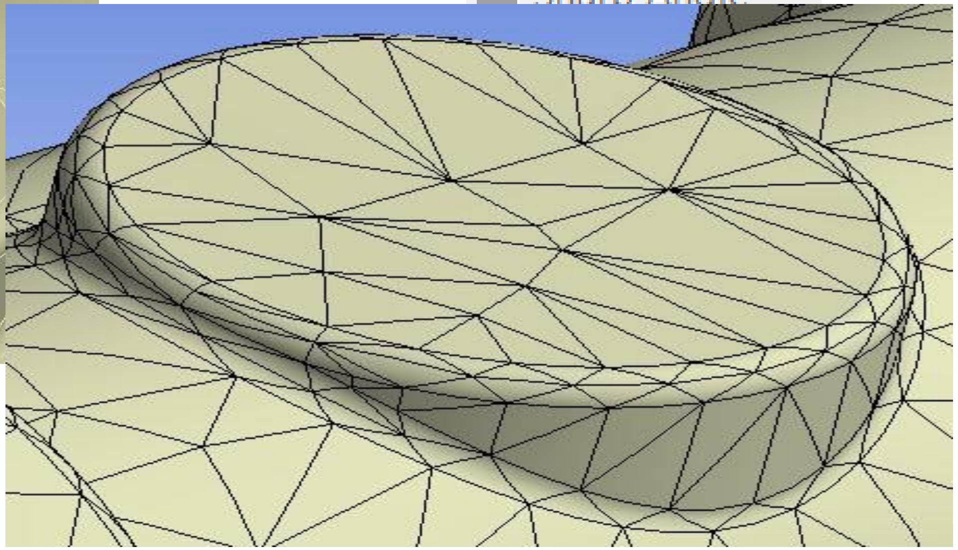
Pinch: allows the removal of small features by "pinching" out small edges and vertices.

- **Master: geometry that retains the original geometry profile.**
- **Slave: geometry that changes to move toward the master.**
- **Can be automatic (mesh branch details) or local (add Pinch branch).**

Details of "Pinch" - Pinch

Scope	
Master Geometry	7 Edges
Slave Geometry	7 Edges
Definition	
Suppressed	No
Tolerance	0.1 mm
Scope Method	Manual

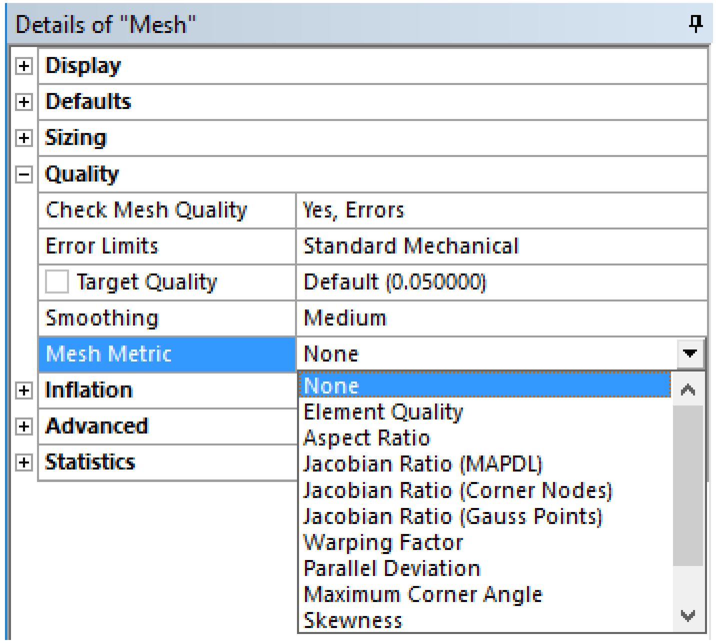


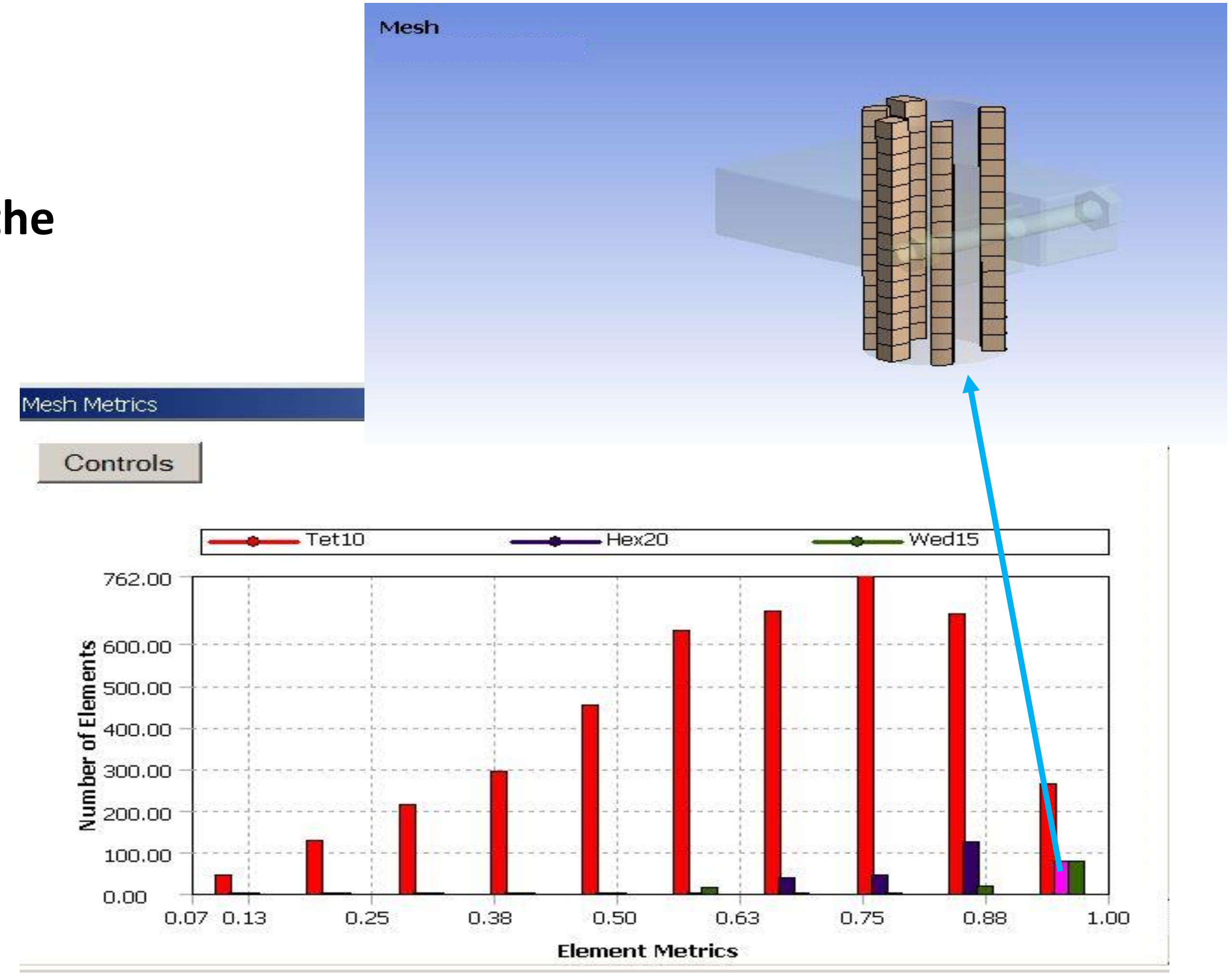


05.04 Troubleshooting

Mesh Metrics

- **Requested in the “Quality” section**
- **Select individual bars in the graph to view the elements graphically.**



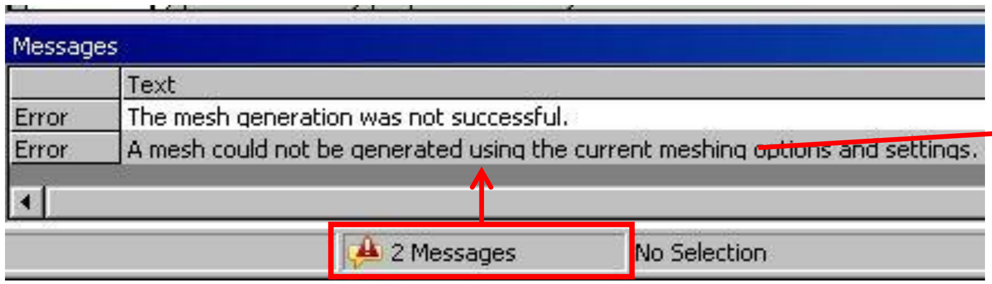


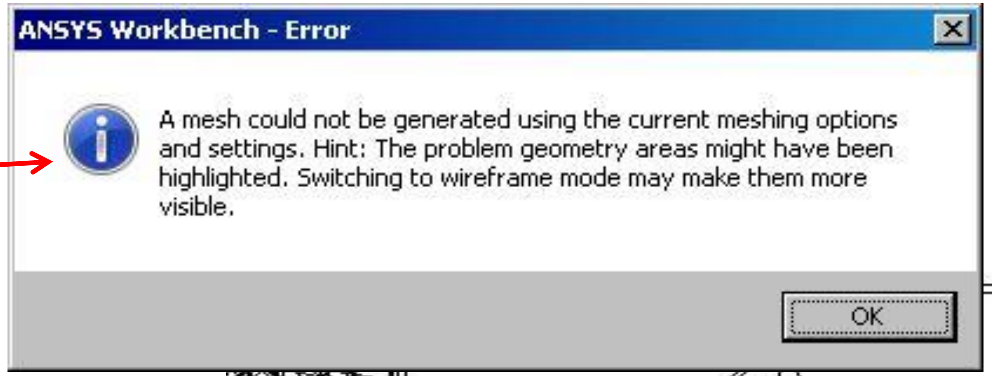
Mesh metrics are described further Section 05.07 “Mesh Quality Criteria” and in Appendix 05.1 “Mesh Quality Criteria.”

05.04 Troubleshooting

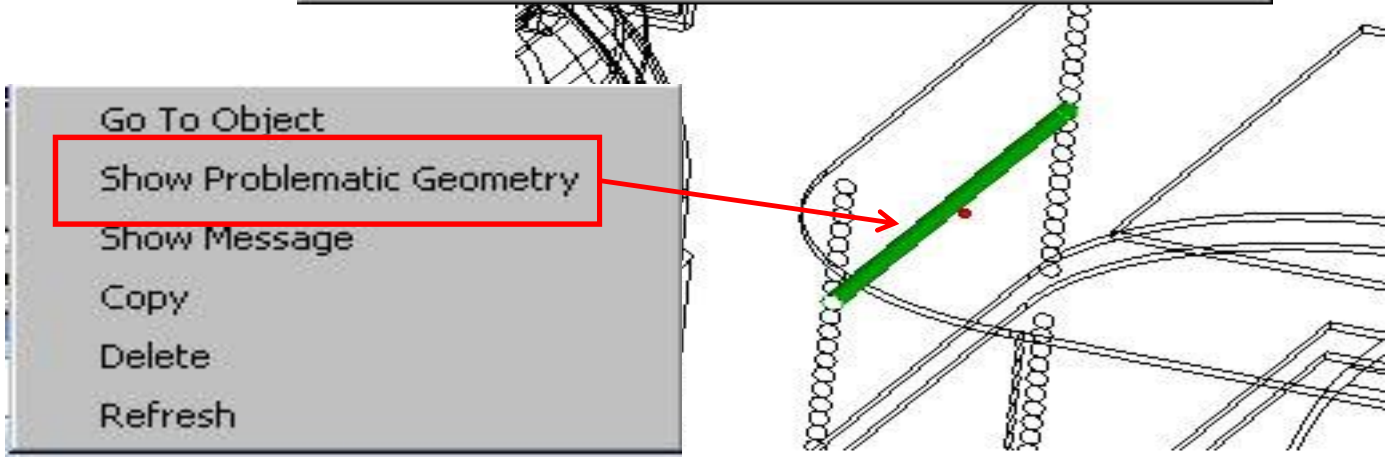
If the mesher is not able to generate a mesh, an error message will be returned.

- **Double-click the message field in the status bar to open the messages window.**
- **Double-click individual messages to show the complete error text in a separate window.**



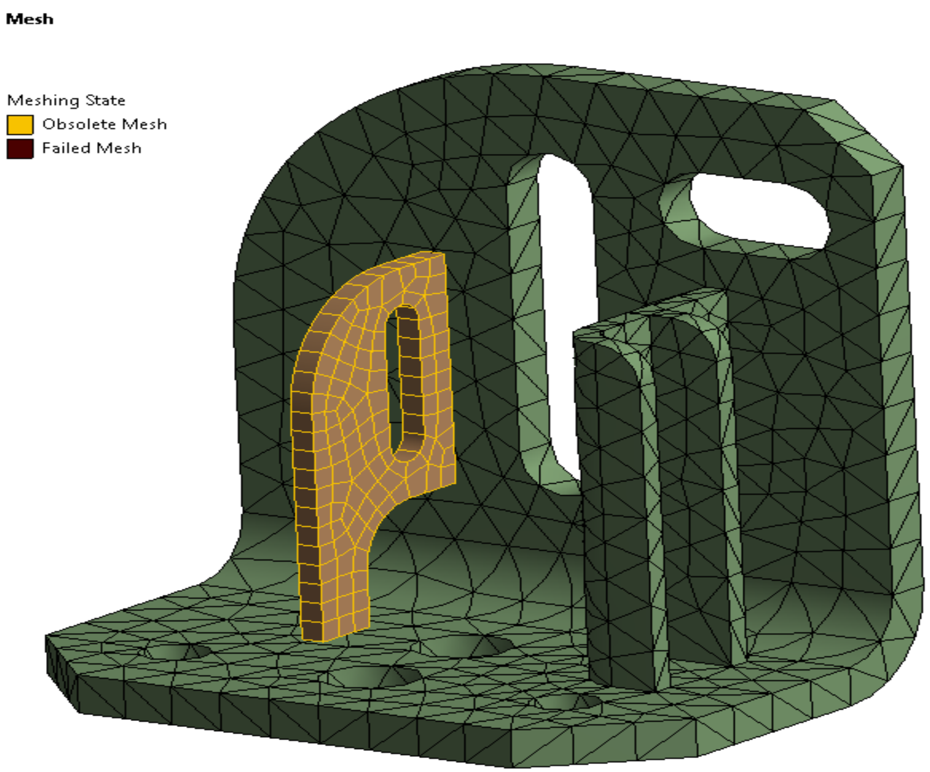


When possible, Mechanical will graphically display the problem region(s)—select the message in the message window, then RMB > Show Problematic Geometry. Using the wireframe view will make finding these areas easier.



05.04 Troubleshooting

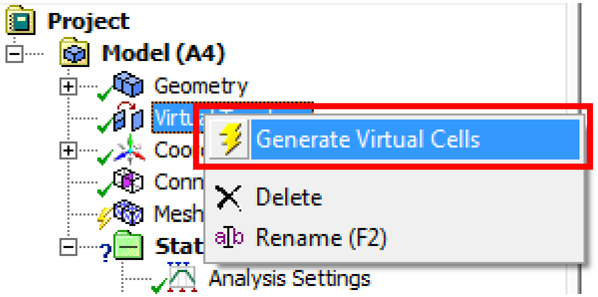
The mesher also provides visual cues to identify obsolete and/or failed meshes. As shown in the figures below, failed meshes are shaded in maroon and obsolete meshes are shaded in yellow.

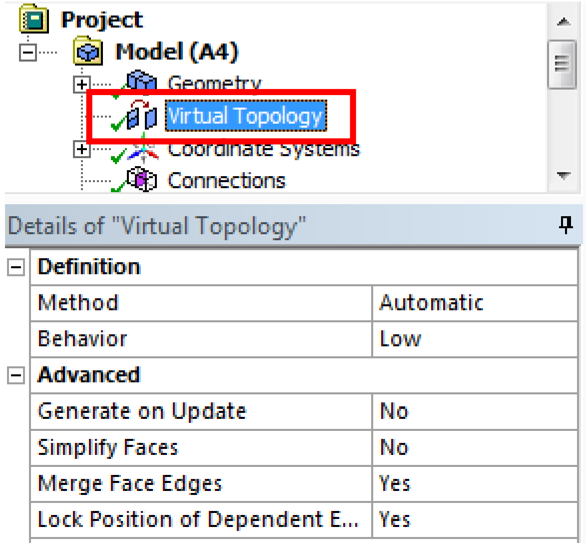


05.05 Virtual Topology

Virtual topology is a feature that can aid you in reducing the number of elements in the model, simplifying small features out of the model, and simplifying load abstraction. The "Virtual Topology" branch is added below the "Model" branch.

- **For meshing certain CAD models you may want to group faces and/or edges together allowing you to form virtual cells in order to reduce or improve the elements.**
- **You can split a face to create two virtual faces, or split an edge to create two virtual edges for improved meshing.**
- **Virtual Cells can be created automatically.**

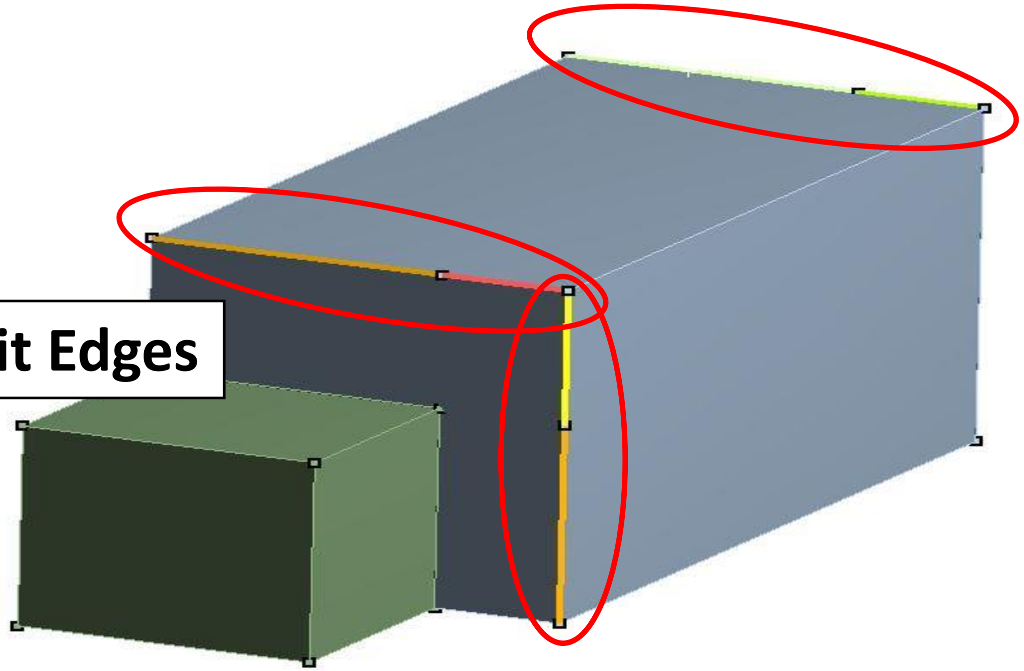


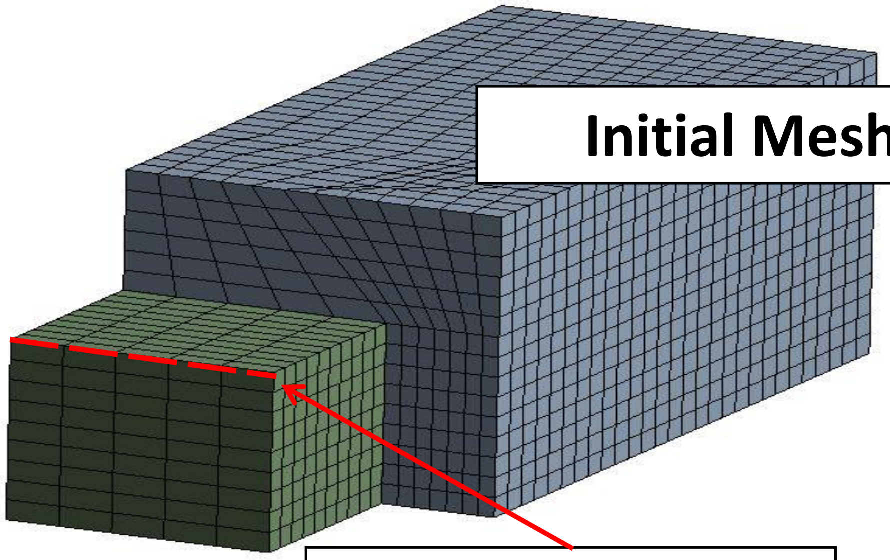


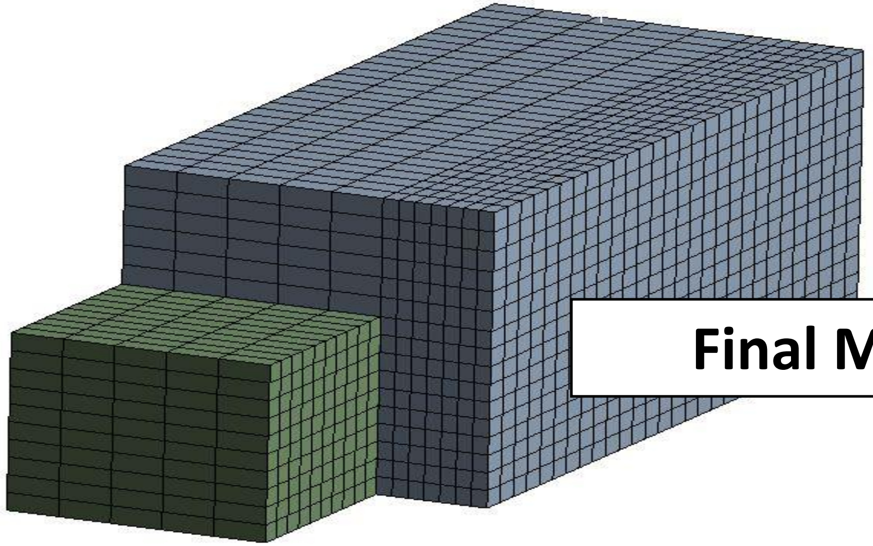
ANSYS

05.05 Virtual Topology

In this example, one edge of this multibody part has a size control assigned which causes irregularities in the overall mesh.



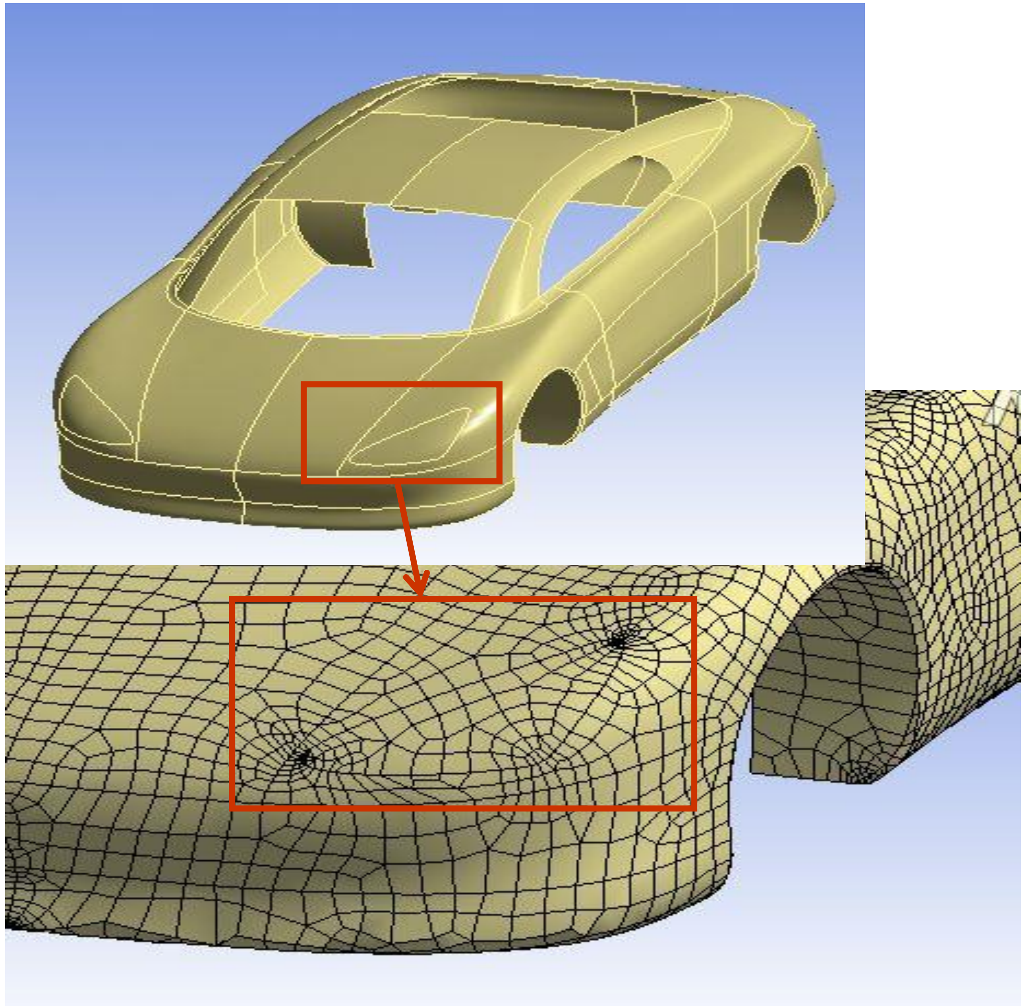


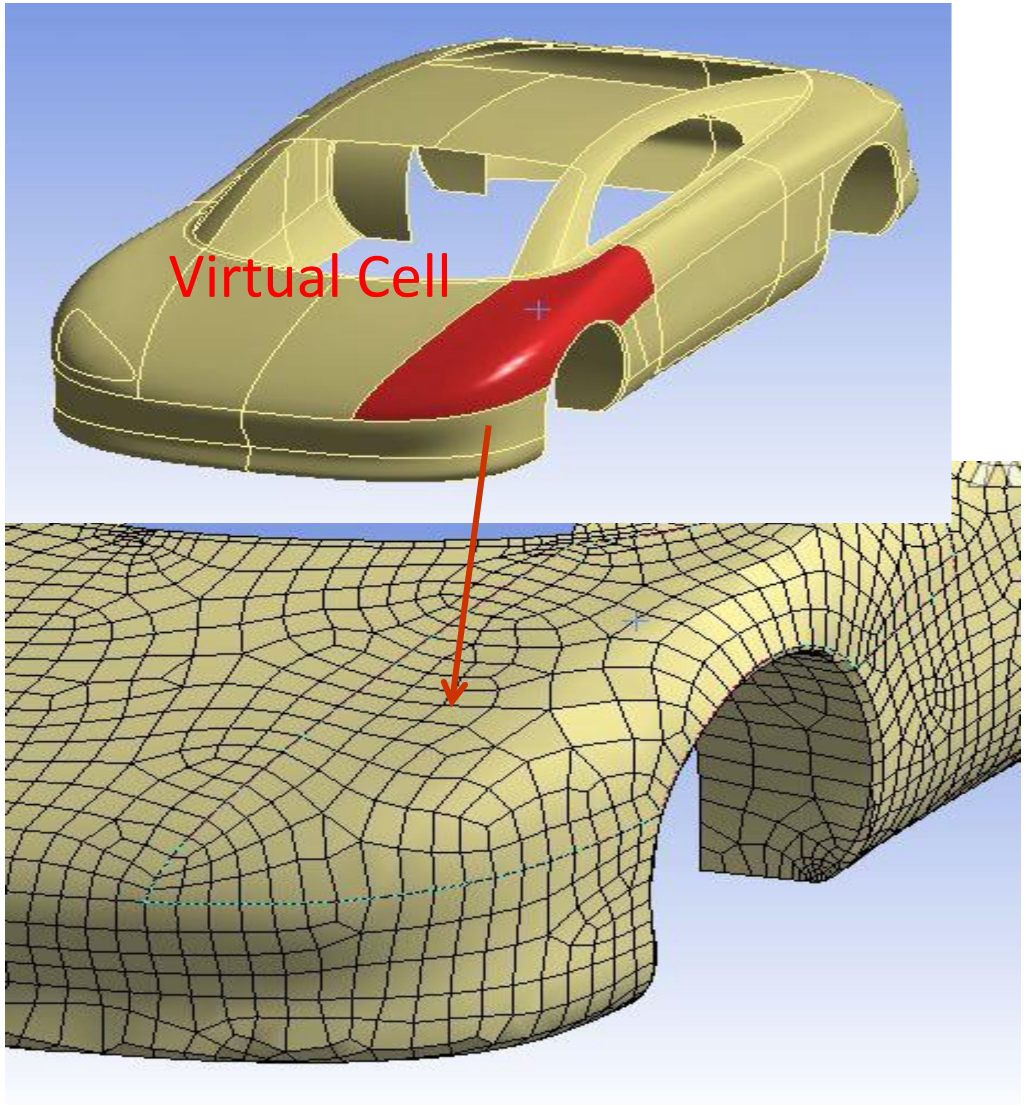


In the figure at upper right, 3 edges are "virtually" split to produce improved elements shapes.

05.05 Virtual Topology

Surface Model Example:



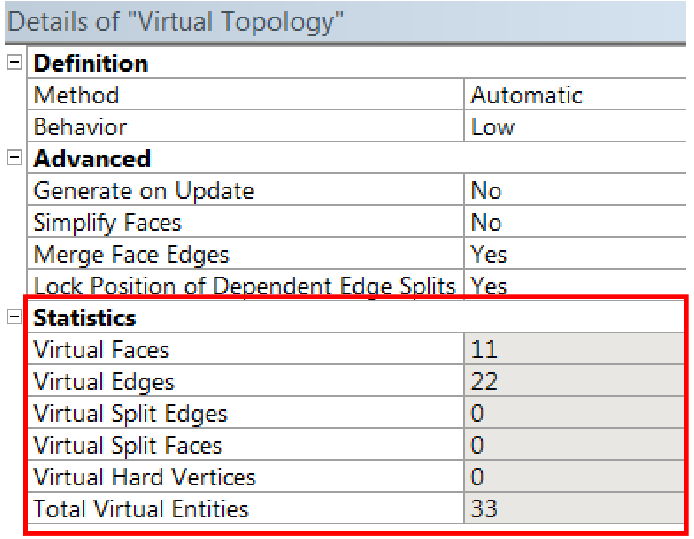


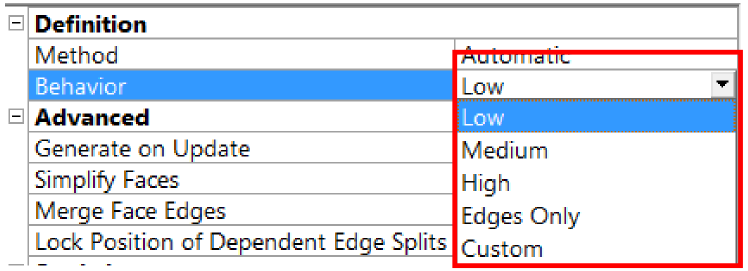
ANSYS

05.05 Virtual Topology

A "Virtual Topology" branch is added below the "Model" branch:

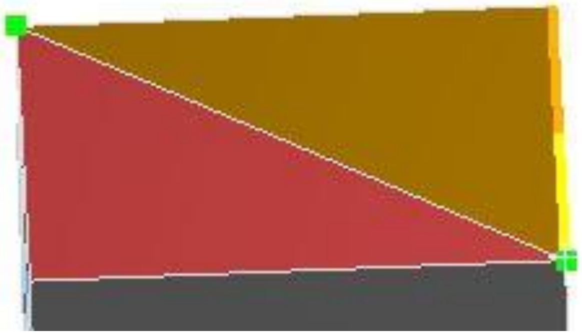
- Individual virtual entities do not appear in the tree. Instead, a statistics section in the details lists virtual entities.
- Virtual Cells can be created manually:
  - Select the entities to be included in the virtual cell.
  - Choose "Merge Cells" in the context menu (or RMB > Insert > Virtual Cell)
- Virtual Cells can be created automatically:
  - Low, Medium, High: Indicates how aggressively virtual topology will be searched for.
  - Edges Only: Searches for adjacent edges to be combined.
  - Custom: users have control on specific options





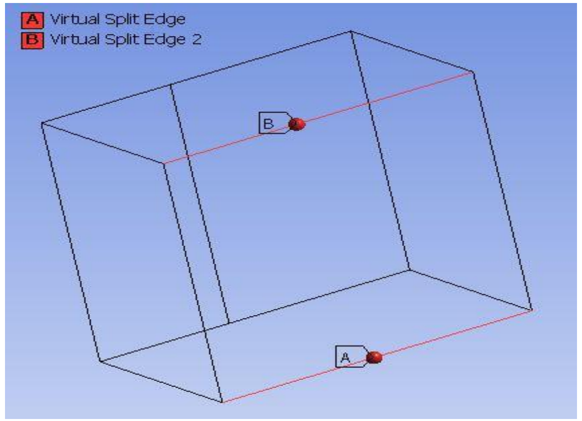
05.05 Virtual Topology

In some instances it may be desirable to modify topology to allow application of some desired effect (e.g., mesh control, load, support, …):



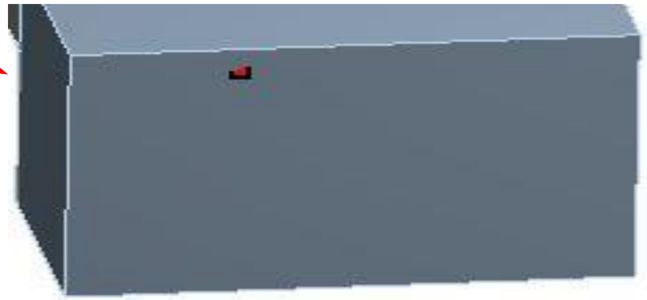
- Split face at vertices (preselect the desired vertices)

  Split Face at Vertices



- Split Edge (preselect the desired edge)

  Split Edge at + Split Edge



- Add a hard vertex (preselect the face at the desired vertex location)

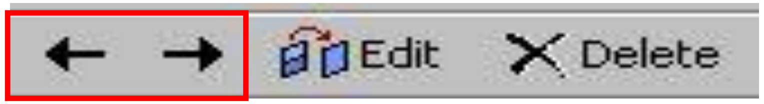
  Hard Vertex at +

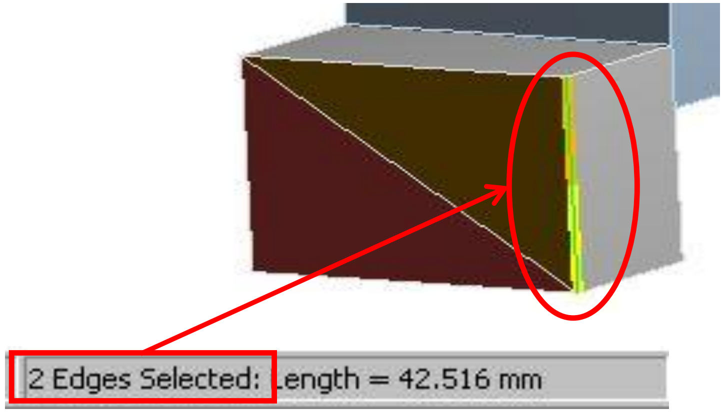
ANSYS

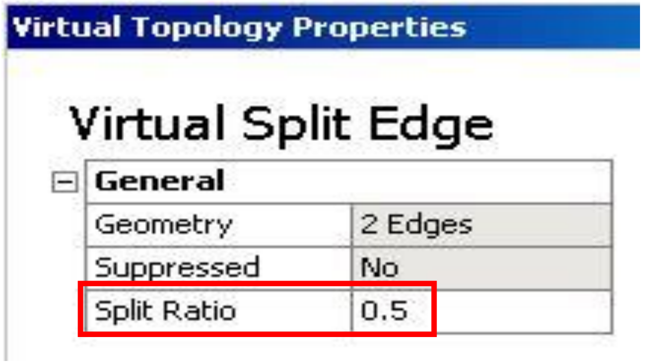
05.05 Virtual Topology

Virtual entities can be reviewed, edited or deleted from the context toolbar (highlight Virtual Topology branch):

- **Use the arrow keys to cycle through next/previous virtual entities.**
- **The virtual entity is highlighted graphically and the status bar (bottom of graphics window) indicates the current selection.**
- **The Edit icon allows access to an editor window where modifications to the virtual entity definition can be made.**
- **Use "Delete" to remove unwanted virtual entities.**



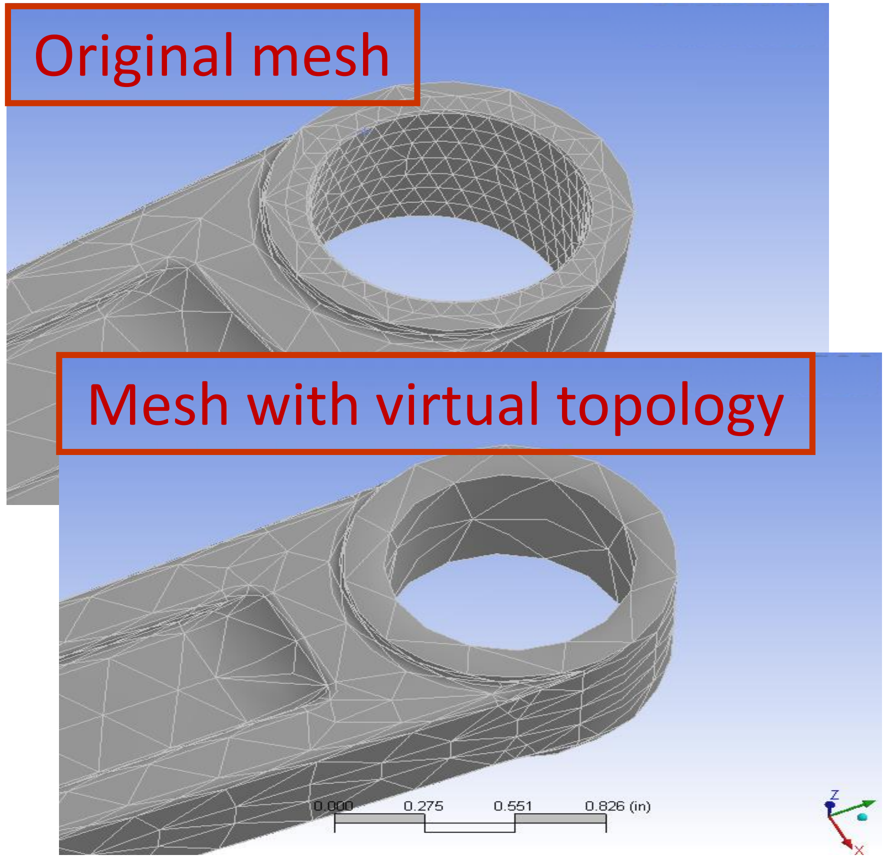


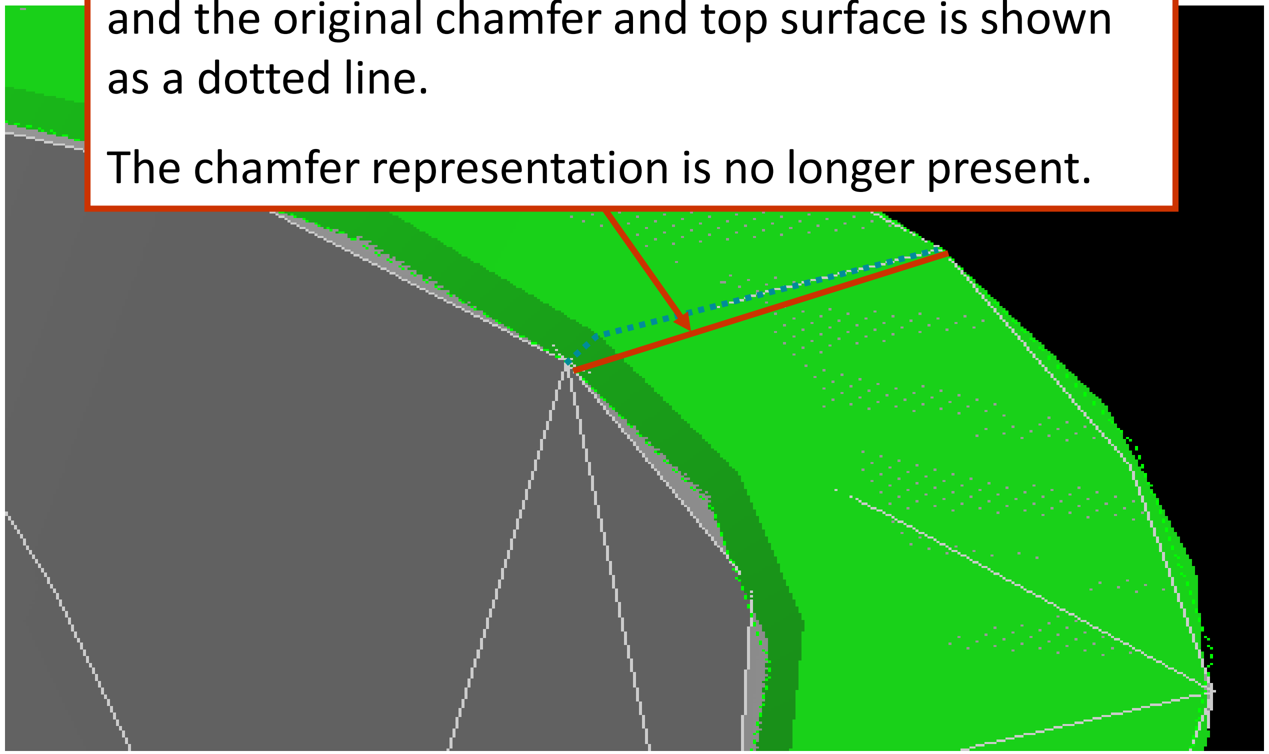


05.05 Virtual Topology

Keep in mind that the underlying "real" topology can change!

Example: a chamfer is included along with the top surface in this virtual cell.
The interior lines are not recognized anymore.





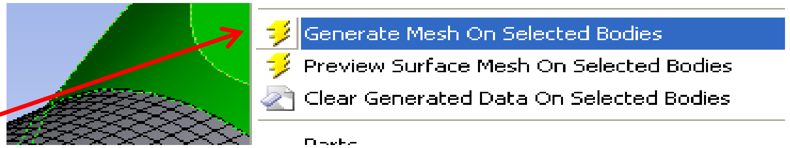
The resulting element's edge is shown as a solid line and the original chamfer and top surface is shown as a dotted line.

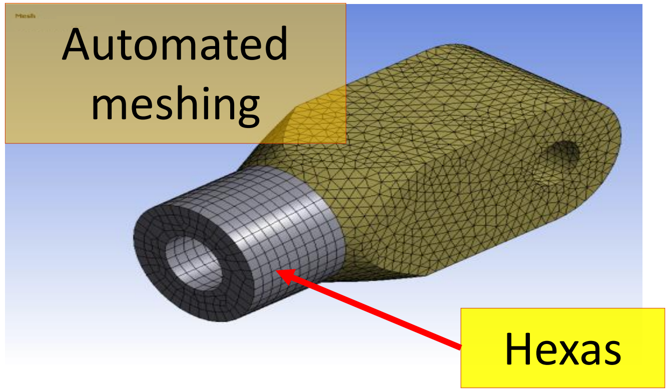
The chamfer representation is no longer present.

05.06 Direct Meshing

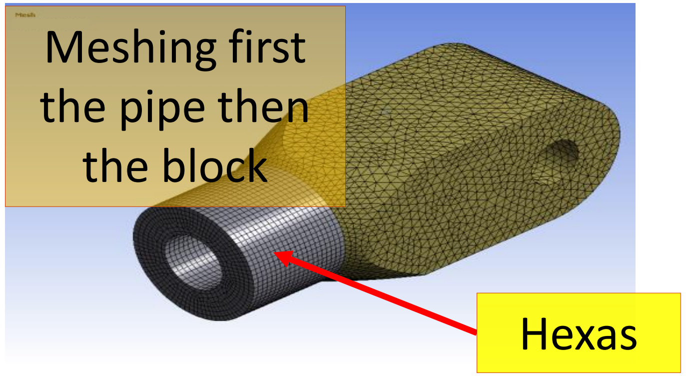
Bodies can be meshed/remeshed individually in any desired order

- **Subsequent bodies will use the attached face mesh**
- **The meshing results will depend on the meshing order**
- **RMB on the body/bodies to generate the mesh locally**

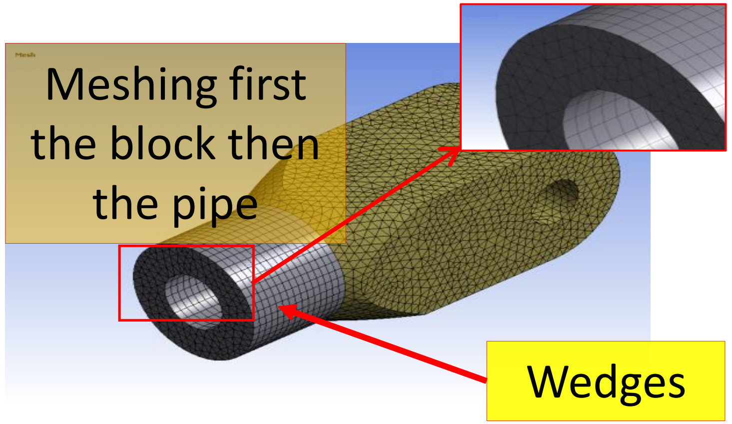




Statistics	
Nodes	8282
Elements	34527
Mesh Metric	Orthogonal Quality
Min	0.280416425480607
Max	0.998788494222392
Average	0.85833002242294
Standard Deviation	8.53178108109779E-02



Statistics	
Nodes	24810
Elements	80178
Mesh Metric	Orthogonal Quality
Min	0.172748547853107
Max	0.999836842472065
Average	0.869178740711722
Standard Deviation	0.093626215921133



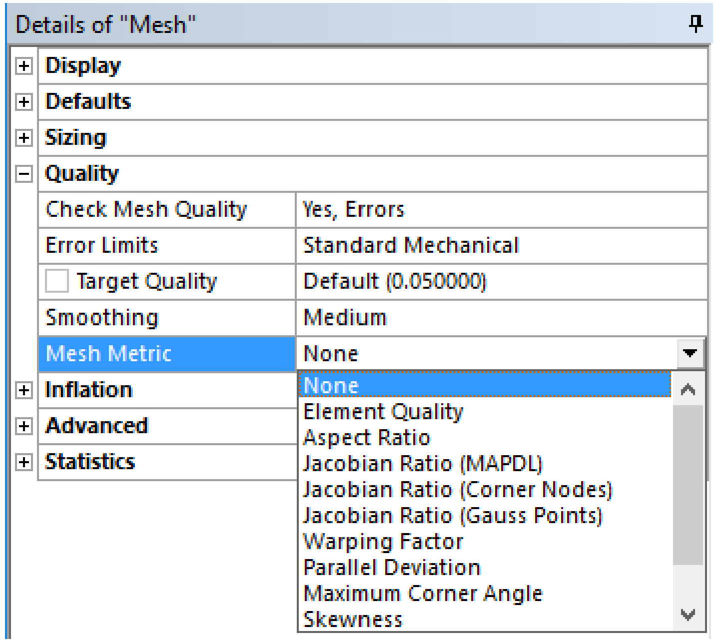
Statistics	
Nodes	13897
Elements	60932
Mesh Metric	Orthogonal Quality
Min	0.336740641439225
Max	0.999369993571416
Average	0.866831224319874
Standard Deviation	8.45868210177912E-02

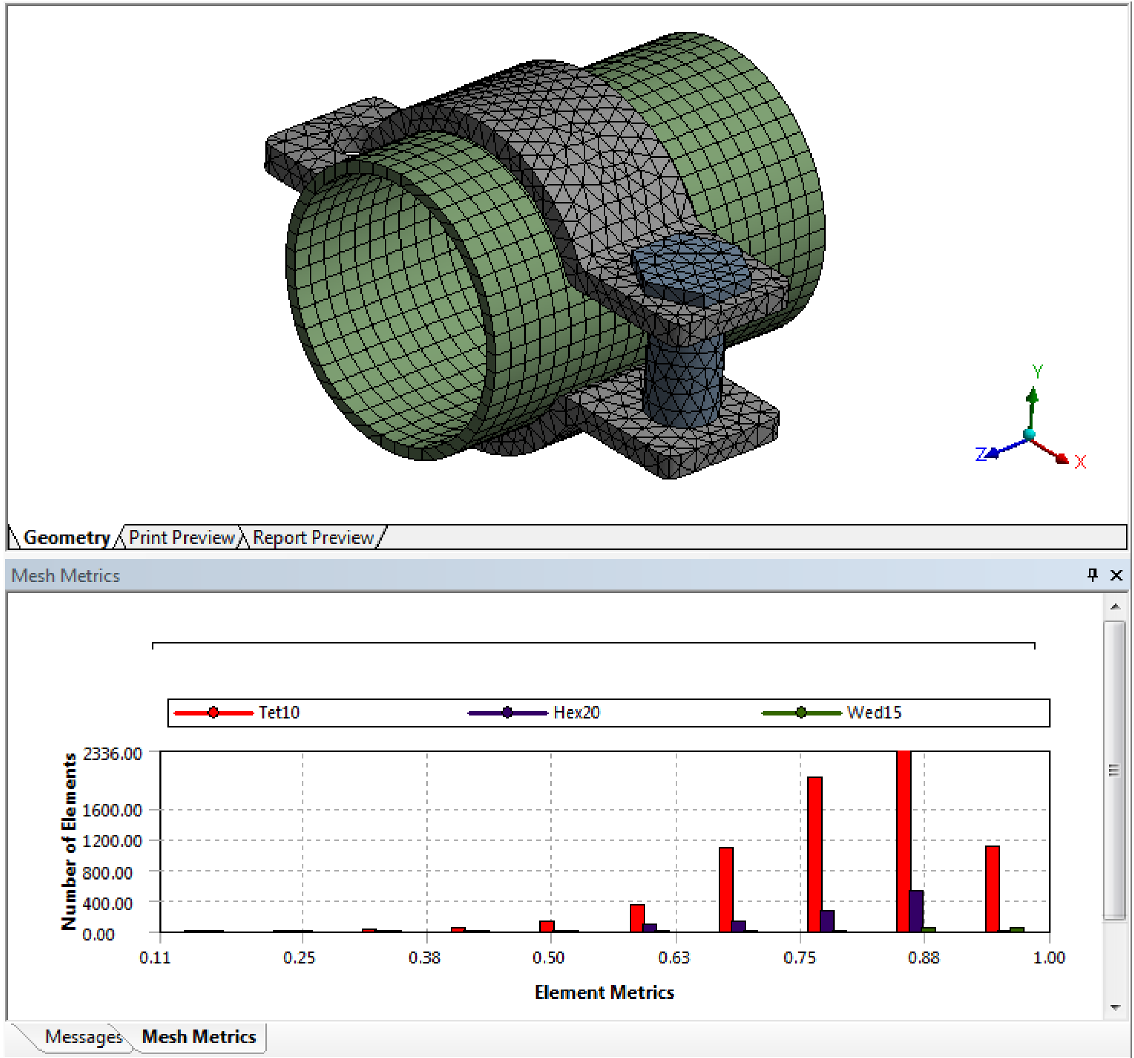
ANSYS

05.07 Mesh Quality Criteria

You can check mesh quality using Mesh Metrics.

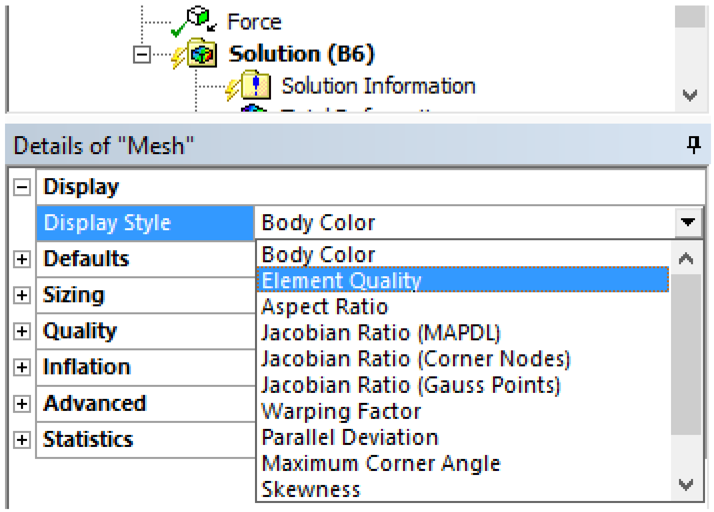
Remember: Each physics type has its own quality criteria.

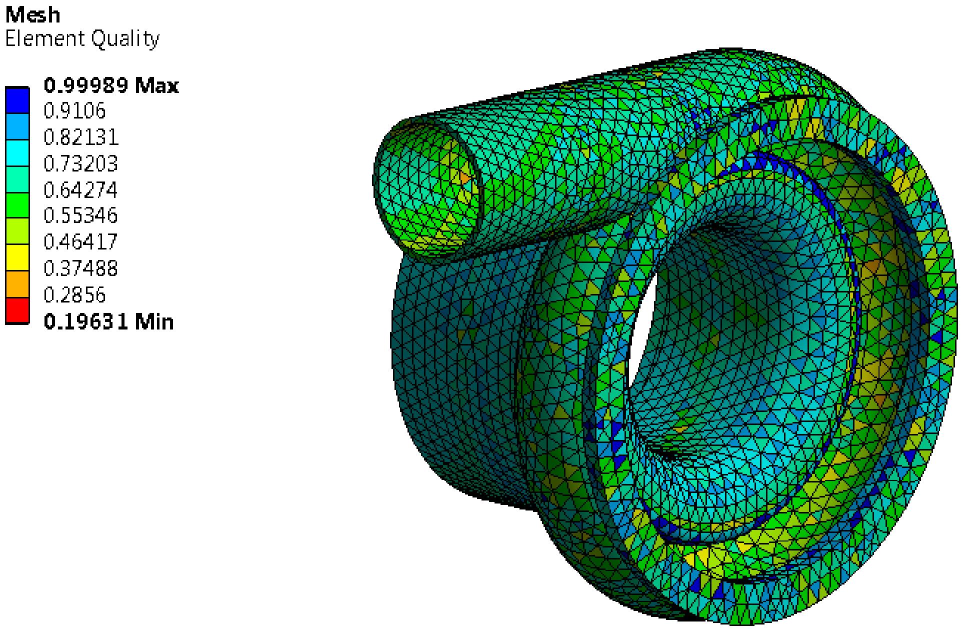




05.07 Mesh Quality Criteria

The Mesh Details *Display Style* option can be used to display the mesh color-coded by the various mesh quality measures:

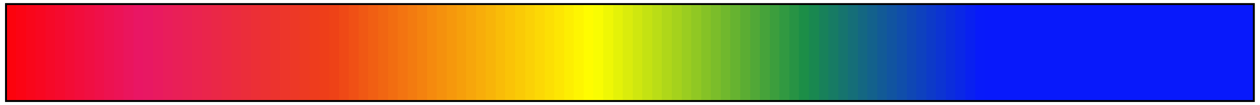




05.07 Mesh Quality Criteria

Example mesh metric : Element Quality

This metric is based on the ratio of the volume to the edge length for a given element.



0	**1**
Bad	**Perfect**

05.07 Mesh Quality Criteria

For additional information on mesh metrics, see
Appendix 05.1: Mesh Quality Criteria

05.08 Workshop 05.1: Mesh Creation

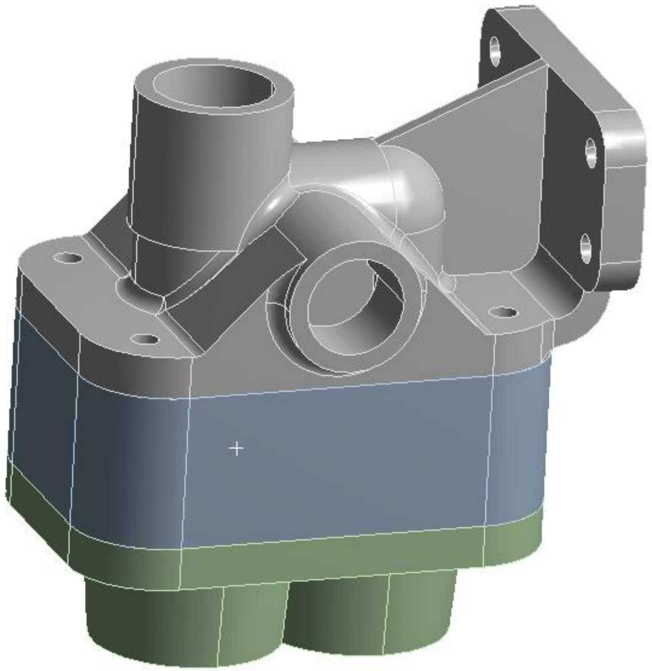
Goal:

Use the various mesh controls to mesh a small assembly.

05.09 Workshop 05.2: Mesh Control

Goal:

Use the various mesh controls to enhance the mesh for a small assembly.

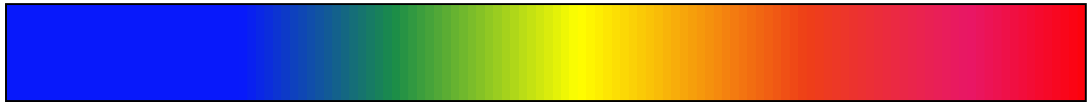


05.10 Appendix 05.1

Mesh Quality Criteria

05.10 Appendix 05.1: Mesh Quality Criteria

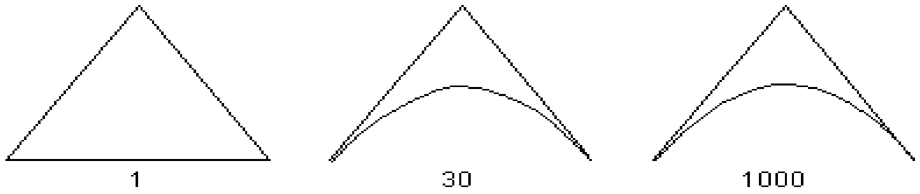
Aspect Ratio:

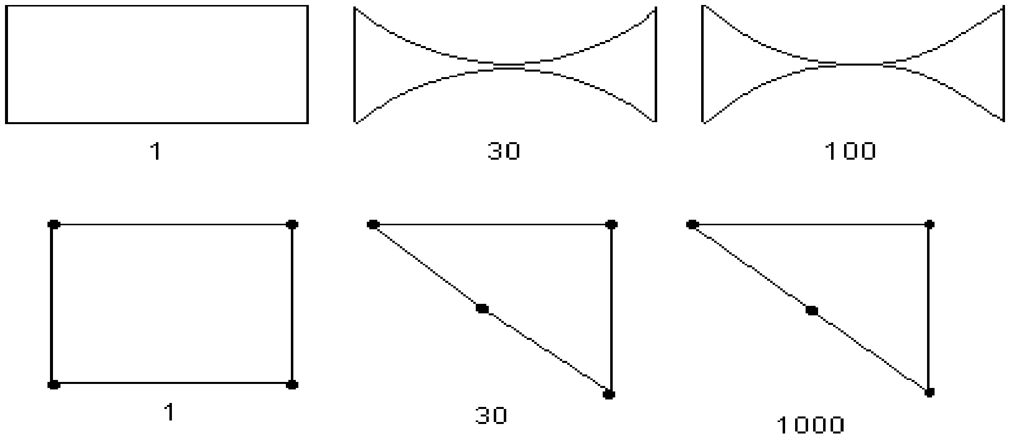


1	**5-10**	**20**	**∞**
Perfect			**Bad**

05.10 Appendix 05.1: Mesh Quality Criteria

Jacobian Ratio:

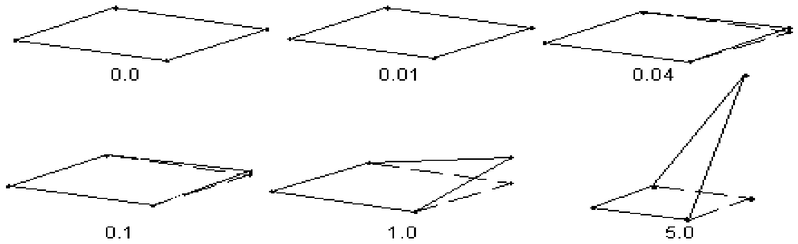


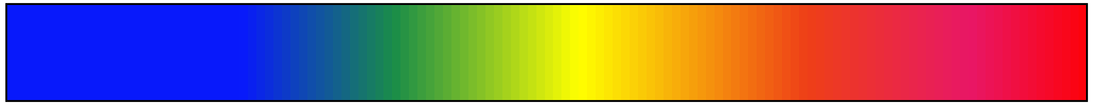


1 Perfect	10	30	∞ Bad

05.10 Appendix 05.1: Mesh Quality Criteria

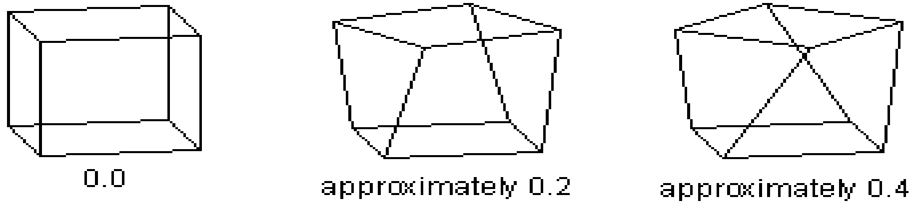
Warping Ratio:





0 — 0.1 — 1 — ∞

Perfect — Bad

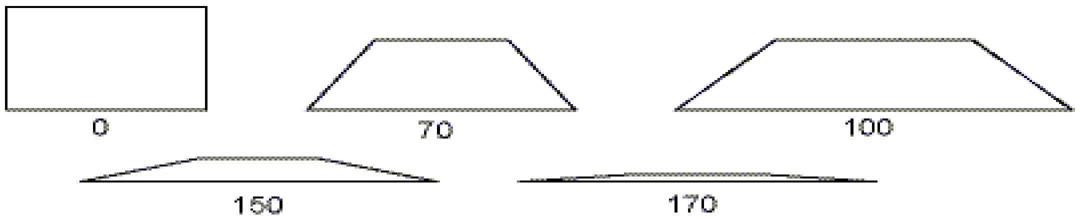


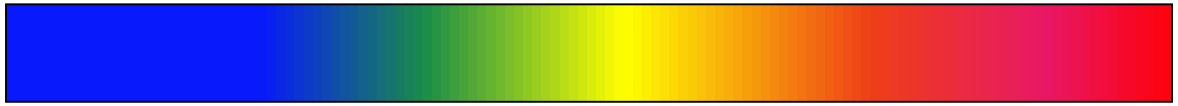
0 — 0.2 — 0.4 — ∞

Perfect — Bad

05.10 Appendix 05.1: Mesh Quality Criteria

Parallel Deviation:

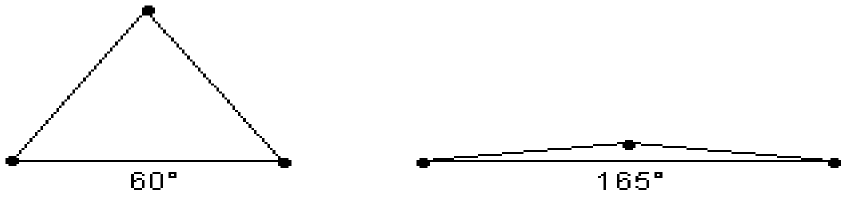


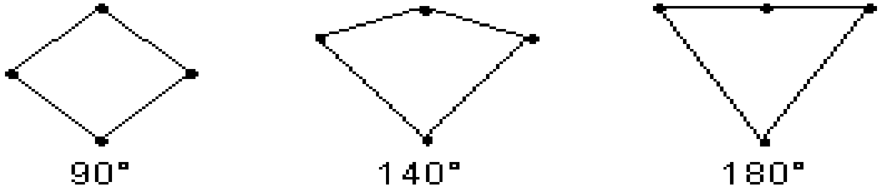


0 Perfect	170 Bad

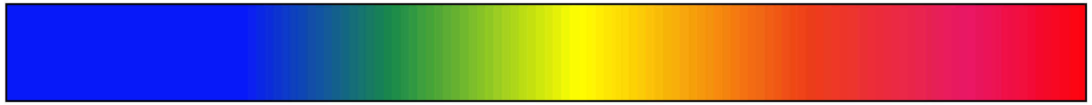
05.10 Appendix 05.1: Mesh Quality Criteria

Maximum Corner Deviation:





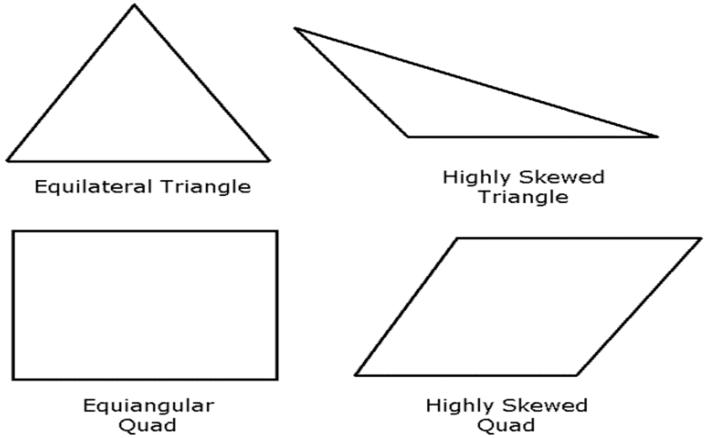
60	**165**
Perfect	**Bad**



90	**180**
Perfect	**Bad**

05.10 Appendix 05.1: Mesh Quality Criteria

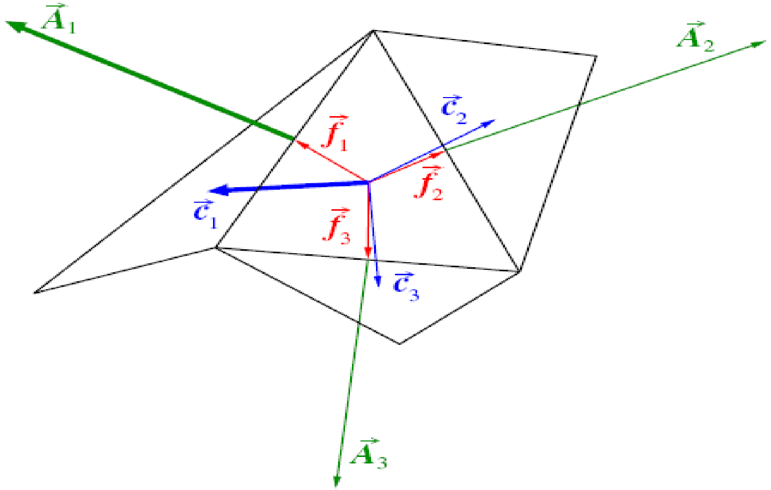
Skewness:



0 Perfect	0.75	1 Bad

05.10 Appendix 05.1: Mesh Quality Criteria

Orthogonal Quality:

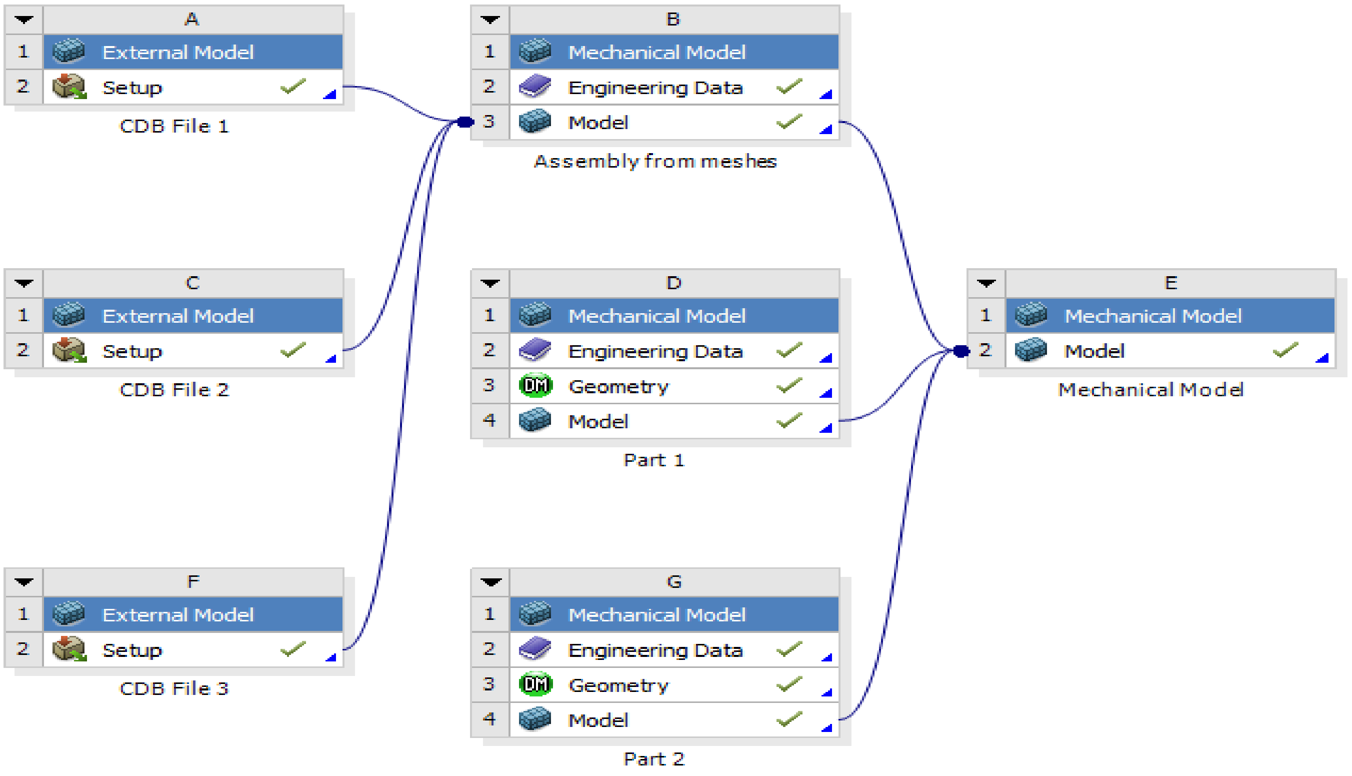


0	1
Bad	Perfect

05.11 Appendix 05.2

Model Assembly

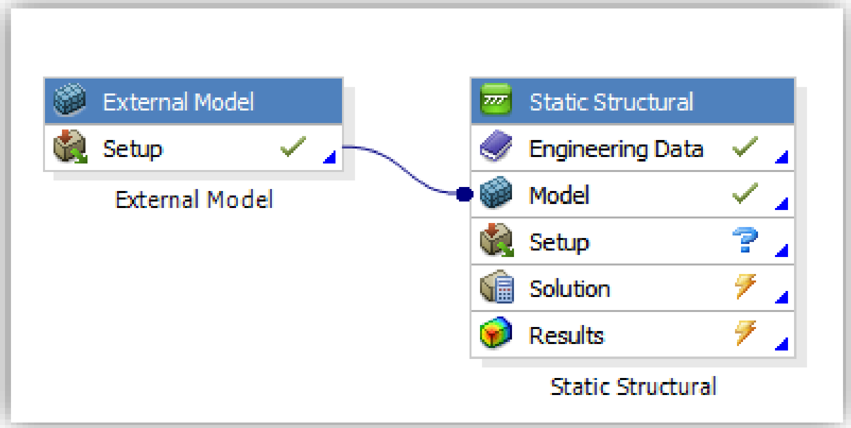
05.11 Appendix 05.2: Model Assembly

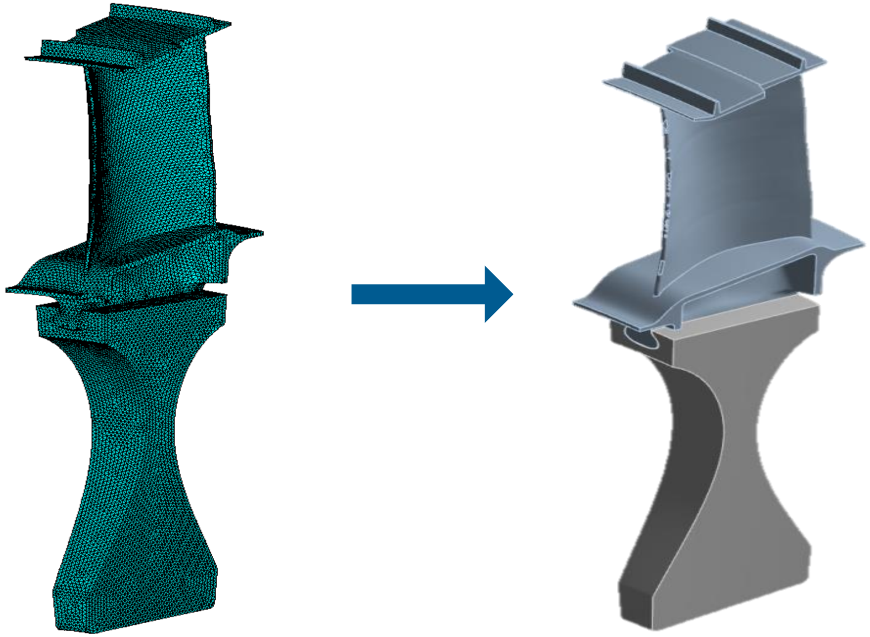


Geometry is not only the starting point for a Workbench-based structural simulation.

Multiple finite element models can be assembled to leverage all Mechanical functionalities, including contact detection.

05.11 Appendix 05.2: Model Assembly





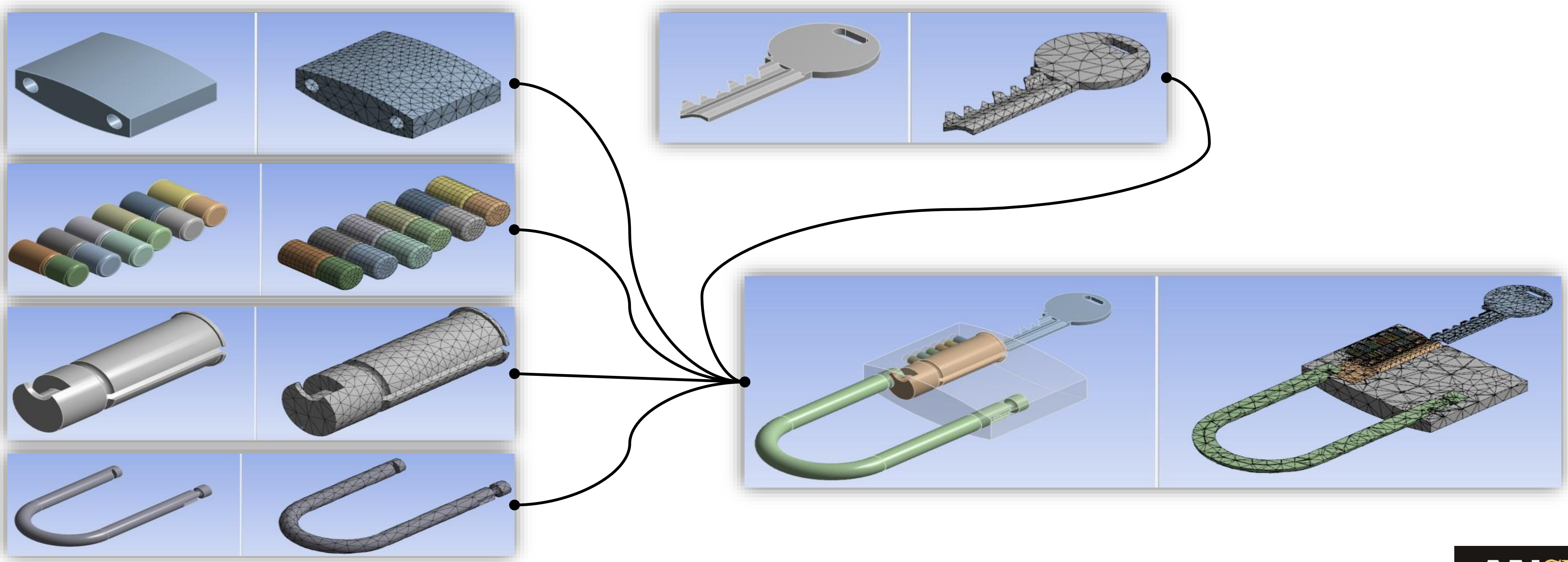
You can import mesh data (solids, shells, and beams) from *.cdb† files into Workbench using the External Model component system and scale, rotate, and/or translate parts as needed.

Contact detection will be performed as if you were working with geometry data.

†A *.cdb file is an ANSYS format that contains model information in terms of ANSYS Mechanical APDL commands.

05.11 Appendix 05.2: Model Assembly

Multiple Workbench systems can also be combined. Geometry, Mesh, and Named Selections are retrieved.



Realize Your Product Promise®

Module 05: Mesh Control

END