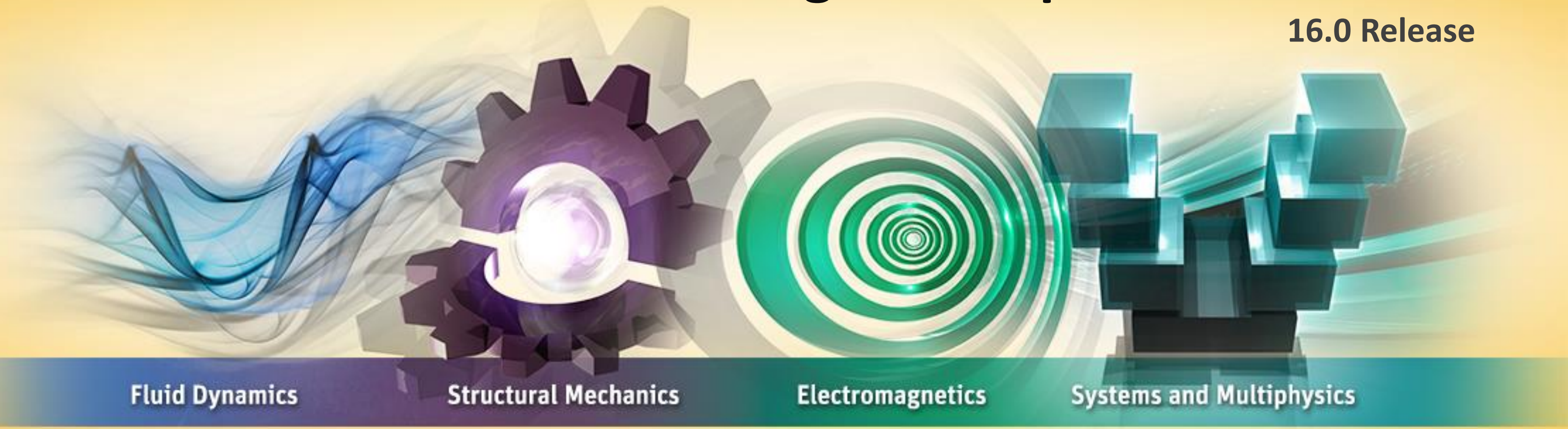
Lecture 4
Meshing Techniques

16.0 Release



Introduction to ANSYS Mechanical

Chapter Overview

In this chapter controlling meshing operations is described.

Topics:

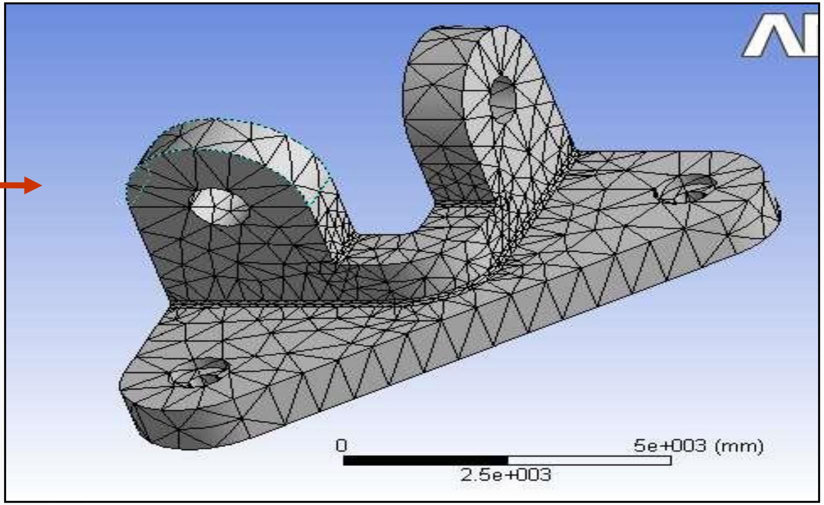
A. Global Meshing Controls
B. Local Meshing Controls
C. Meshing Troubleshooting
D. Virtual Topology
E. Direct Modeling
F. Mesh quality Criteria
G. Workshop 4.1 – Mesh Control
H. Appendix

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).



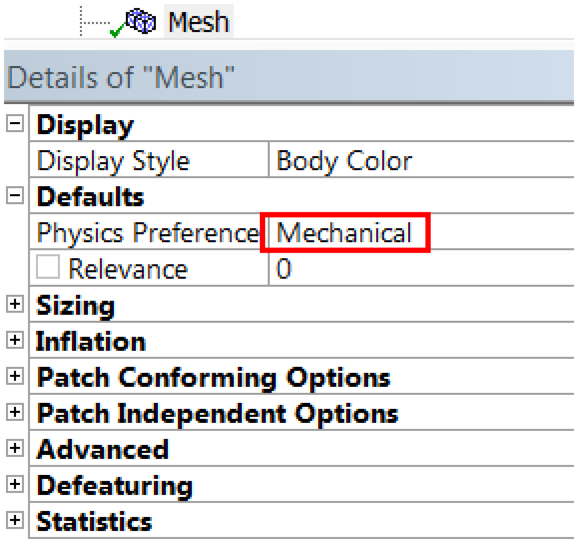
A. Global Meshing Controls

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:

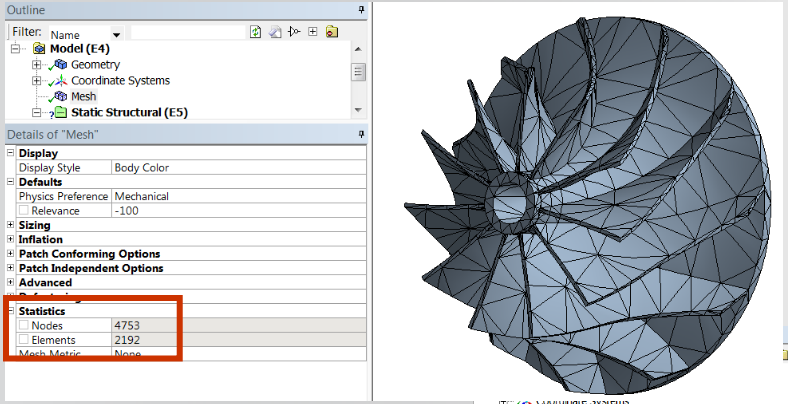
- **Mechanical**
- **Electromagnetics**
- **CFD**
- **Explicit**

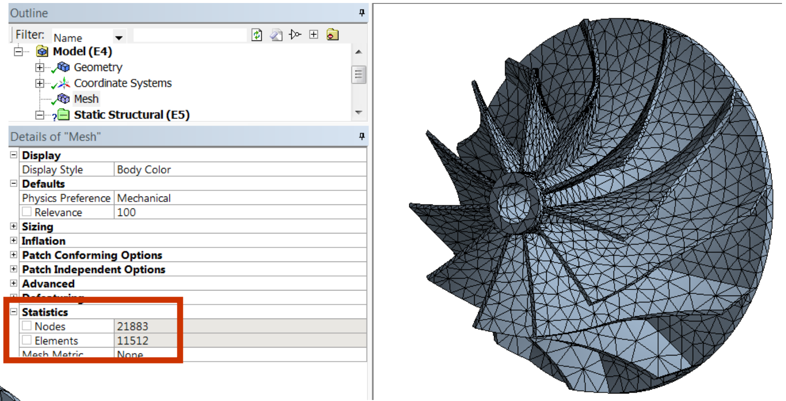
Different analysis types define acceptable or favorable element shapes differently. For this course we limit the discussion to Mechanical.

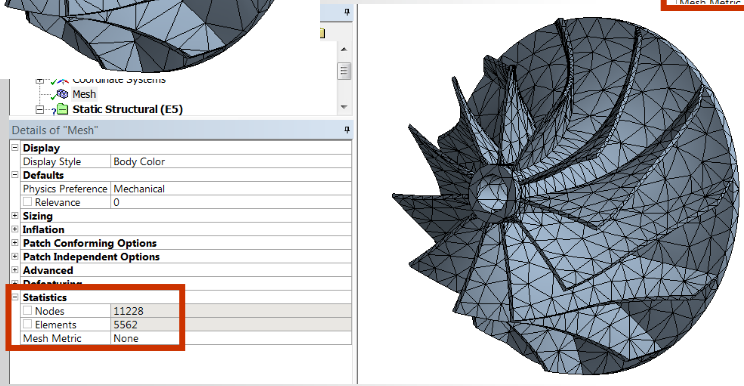


… Global Meshing Controls

- 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).



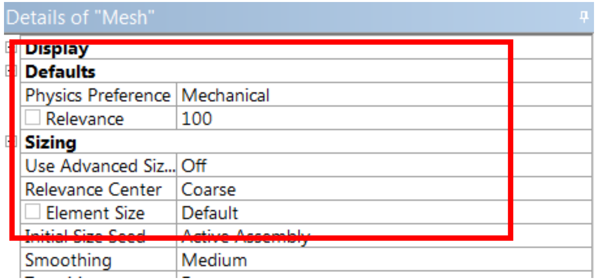




- Relevance = coarse mesh

+ Relevance = fine mesh

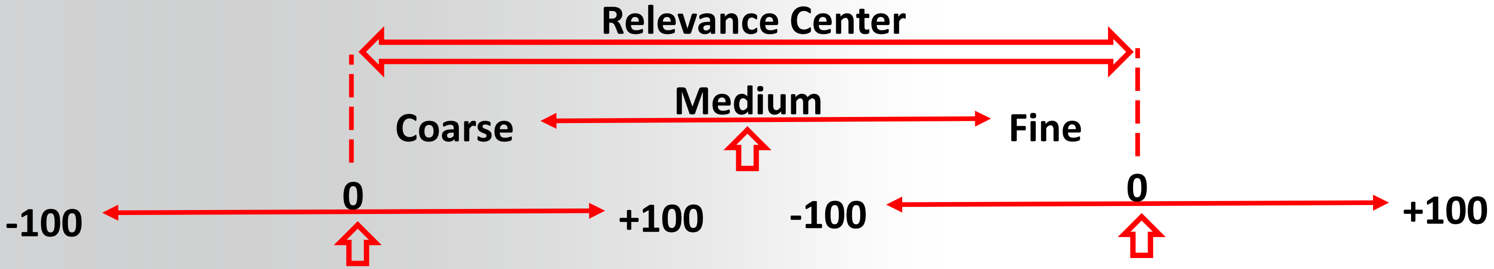
... Global Meshing Controls



Sizing Control:

- **These settings assume the “Use Advanced Size Function” is set to “Off”.**

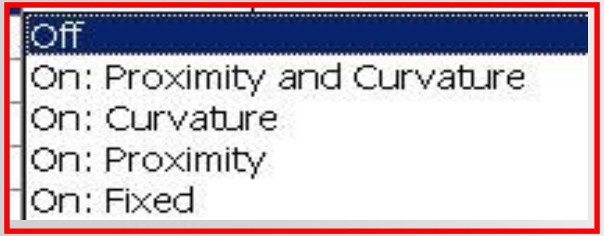
- **Relevance Center:** sets the mid point of the “Relevance” slider control.
- **Element Size:** defines the maximum element size used for the entire model.
- **For most static structural applications the default values for the remaining global controls are usually adequate.**

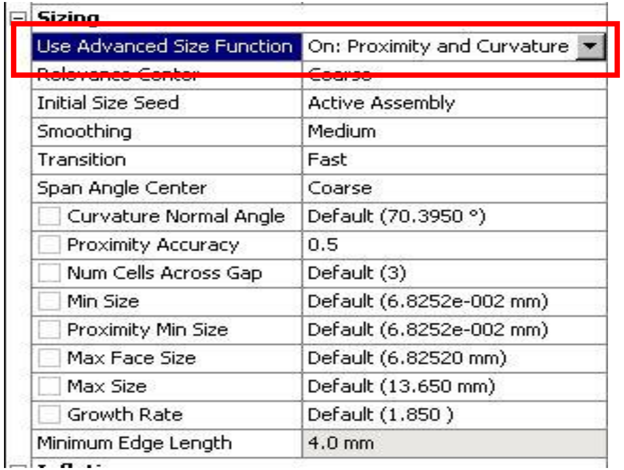


… Global Meshing Controls

Advanced Size Functions provide additional control over the global mesh sizing and are activated in the mesh details.

- **While many of these controls are beyond the scope of an introductory course we'll explain 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 (e.g. nonlinear, transient thermal, etc.).**
- **Three advanced size functions can be employed: proximity, curvature and fixed (proximity and curvature can be combined).**





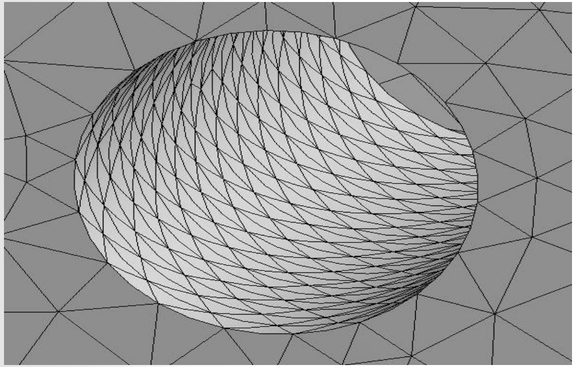
… Global Meshing Controls

The "Fixed" size function provides minimum and maximum element size controls.

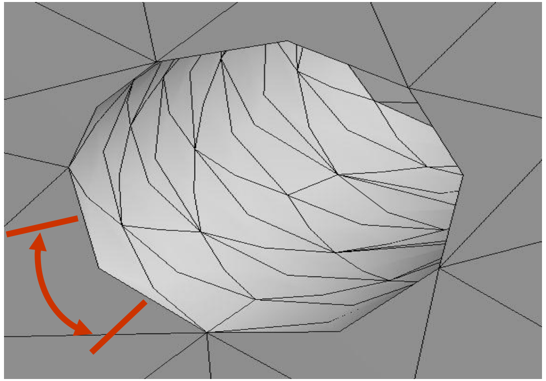
"Curvature" as the name implies, is driven by the curvature encountered in the geometry. For models dominated by lots of curved features this control provides a way to refine the mesh over much of the model without using local controls.

For models composed of mostly straight features the control will have a lesser impact.

Sizing	
Use Advanced Size Function	On: Curvature
Relevance Center	Coarse
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Curvature Normal Angle	Default (70.3950 °)
Min Size	Default (5.8756e-002 mm)
Max Face Size	5.0 mm
Max Size	Default (11.7510 mm)
Growth Rate	Default (1.850)
Minimum Edge Length	1.250 mm



Curvature = 20 deg.



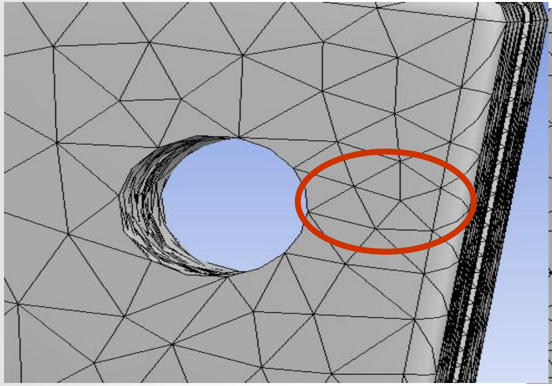
Curvature = 75 deg.

… Global Meshing Controls

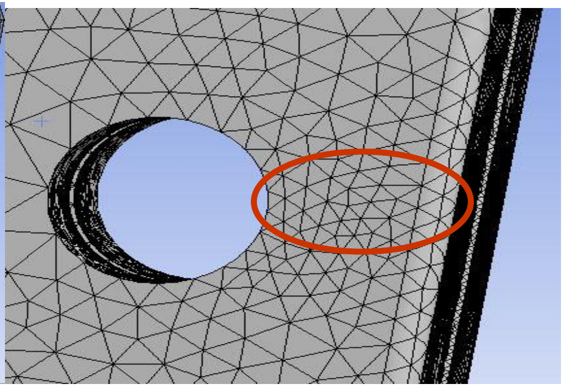
Proximity provides a means to control the mesh density in regions of the model where features are located closely together. In cases where the geometry contains lots of detail this can be a quick way to refine the mesh in all areas without applying numerous local controls.

As mentioned earlier proximity and curvature can be combined. The choice of control is dictated by the geometry being meshed.

Sizing	
Use Advanced Size Function	On: Proximity
Relevance Center	Coarse
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Num Cells Across Gap	Default (5)
Proximity Size Function Sources	Faces and Edges
Proximity Min Size	Default (8.4169e-002 mm)
Max Face Size	Default (8.41690 mm)
Max Size	Default (16.8340 mm)
Growth Rate	Default (1.50)
Minimum Edge Length	1.06650 mm



Num Cells = 4



Num Cells = 12

... Global Meshing Controls

Shape Checking:

- Standard Mechanical – linear stress, modal and thermal analyses.
- Aggressive Mechanical – large deformations and material nonlinearities.

Element Midside Nodes:

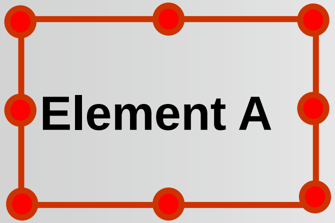
- Program Controlled (default), Dropped or Kept (see below).

Number of Retries: if poor quality elements are detected the mesher will retry using a finer mesh.

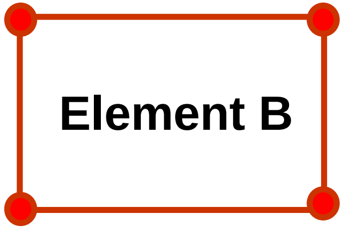
Details of "Mesh"

⊞ Display	
⊞ Defaults	
⊞ Sizing	
⊞ Inflation	
⊞ Patch Conforming Options	
⊞ Patch Independent Options	
⊟ Advanced	
Number of CPUs for Parallel P...	Program Controlled
Shape Checking	Standard Mechanical
Element Midside Nodes	Program Controlled
Straight Sided Elements	No
Number of Retries	0
Extra Retries For Assembly	Yes
Rigid Body Behavior	Dimensionally Reduced
Mesh Morphing	Disabled
⊞ Defeaturing	

Mesh Morphing: when enabled allows updated geometry to use a morphed mesh rather than remeshing (saves time). Topology must remain the same and large geometry changes cannot be morphed.



Kept

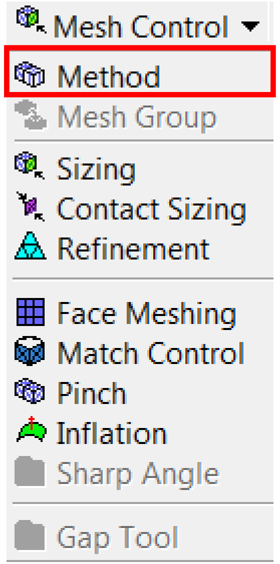


Dropped

B. Local Meshing Controls

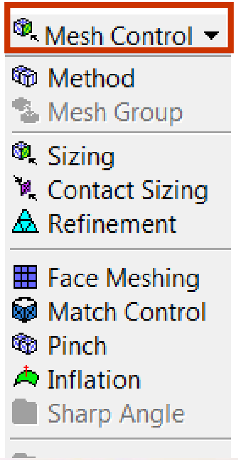
Local Mesh Controls can be applied to either a Geometry Selection or a Named Selection. These are available only when the mesh branch is highlighted. Available controls include :

- **Method Control**
- **Sizing Control**
- **Contact Sizing Control**
- **Refinement Control**
- **Mapped Face Meshing**
- **Match Control**
- **Pinch Control**
- **Inflation Control**



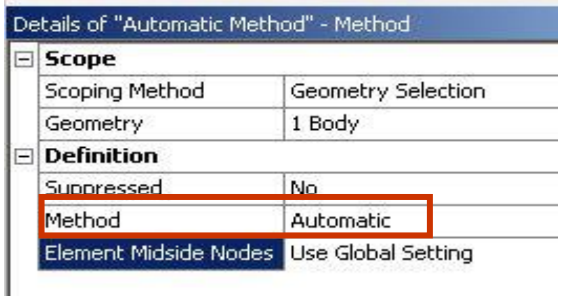
… Local Meshing Controls

Method Control : Provides the user with options as to how bodies are meshed:



Automatic (default):

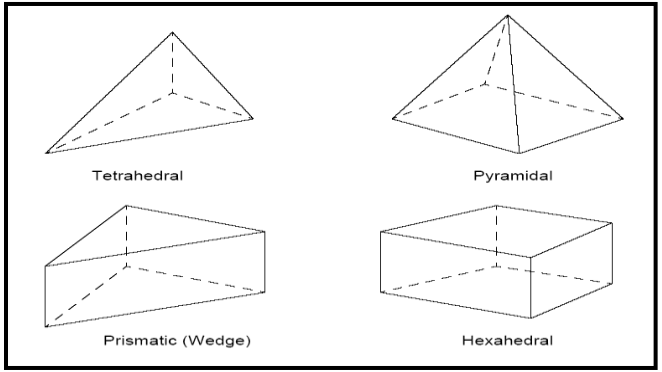
- **Body will be swept if possible. Otherwise, the “Patch Conforming” mesher under “Tetrahedrons” is used.**



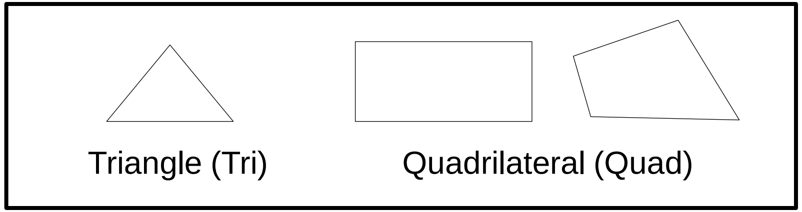
Continued . . .

Meshing Methods

- Meshing Methods available for 3D bodies
  - Automatic
  - Tetrahedrons
    - Patch Conforming
    - Patch Independent
  - MultiZone
    - Mainly hexahedral elements
  - Hex dominant
  - Sweep



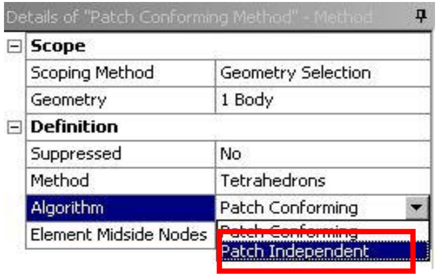
- Meshing Methods available for 2D bodies
  - Automatic Method (Quad Dominant)
  - Triangles
  - Uniform Quad/Tri
  - Uniform Quad



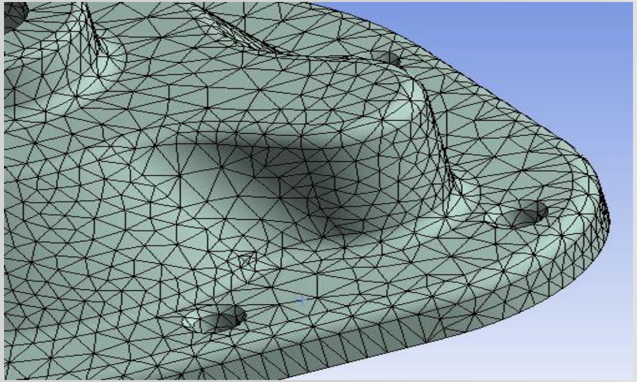
… Local Meshing Controls

Tetrahedrons Method:

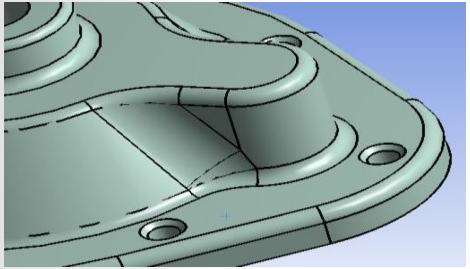
- All Tetrahedra mesh is generated (not usually requested for mechanical applications).
- Can use Patch Conforming or Patch Independent Meshing algorithms.



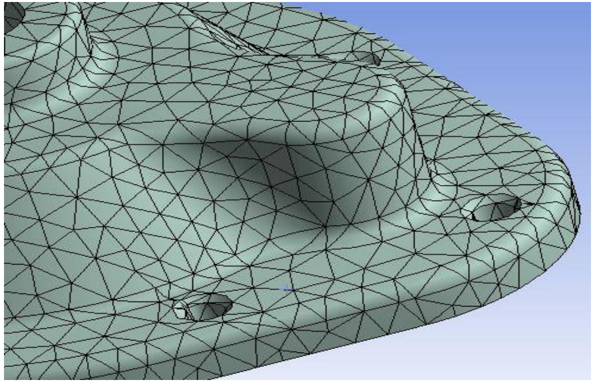
Patch Conforming



Underlying Geometry



Patch Independent

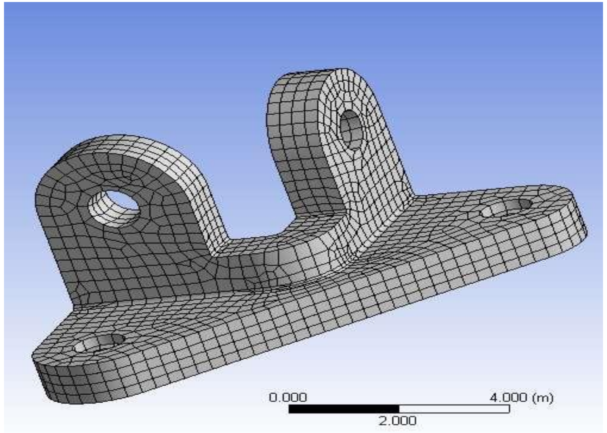


ANSYS License	Availability
DesignSpace	x
Professional	x
Structural	x
Mechanical/Multiphysics	x

… Local Meshing Controls

Hex Dominant Method : 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%)

ANSYS License	Availability
DesignSpace	
Professional	x
Structural	x
Mechanical/Multiphysics	x

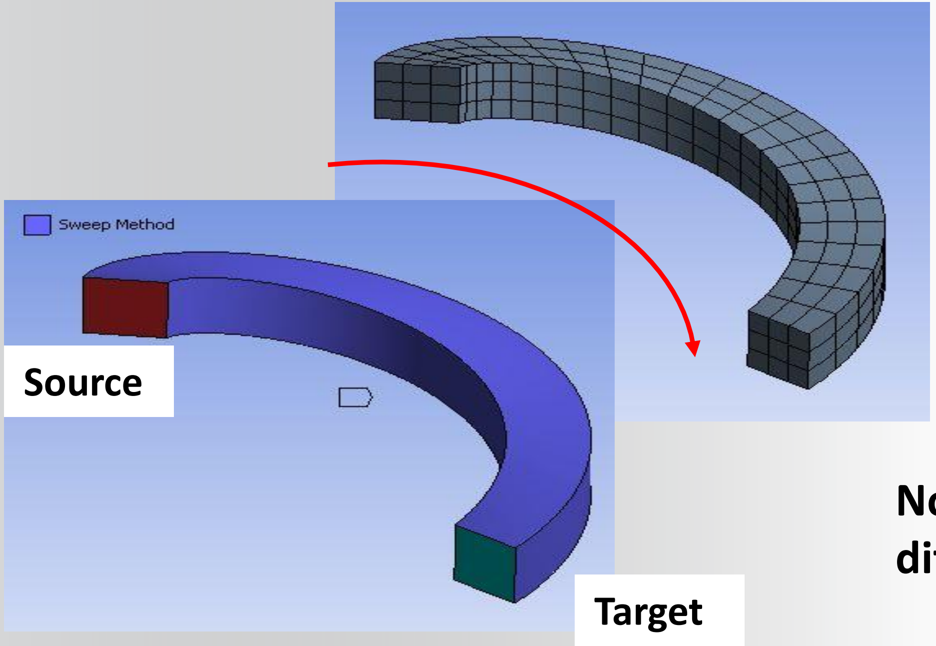
… Local Meshing Controls

Sweep Method (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.**

Details of "Sweep Method" - Method

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Active	Yes
Method	Sweep
Element Midside Nodes	Use Global Setting
Src/Trg Selection	Manual Source and Target
Source	1 Face
Target	1 Face
Free Face Mesh Type	All Quad
Type	Number of Divisions
Sweep Num Divs	Default
Sweep Bias Type	No Bias
Element Option	Solid



Note: the geometry shown here has 6 different possible sweep directions.

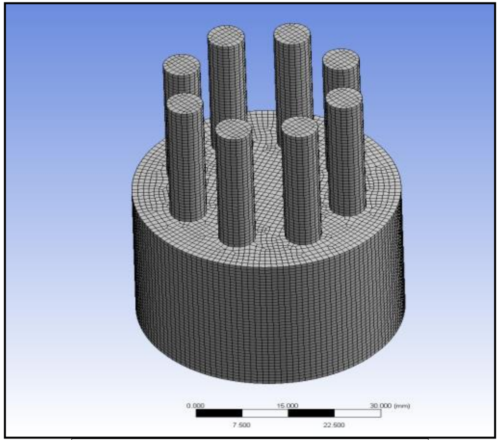
MultiZone Meshing

- 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, Hexa Dominant or Hex Core based on the selected method

Program Controlled inflation

Access

- RMB on Mesh
- Insert → Method
- Definition → Method → MultiZone

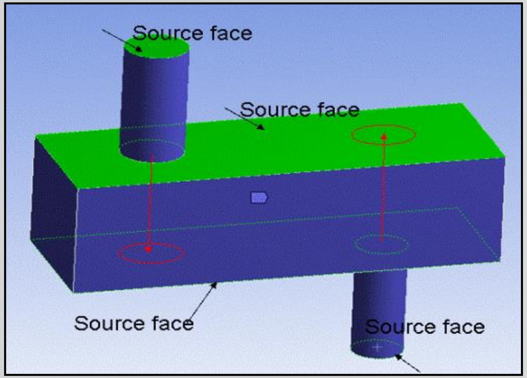


MultiZone Mesh

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
Minimum Edge Length	1.0665 mm
Write ICEM CFD Files	No

MultiZone Meshing

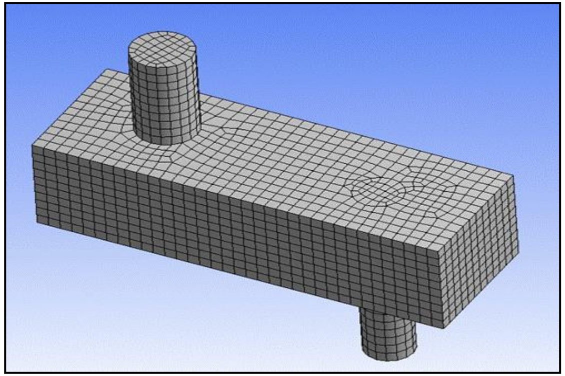
- **Control:**
  - **Src/Trg Selection - Automatic**



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/Prism
Free Mesh Type	Not Allowed
Element Midside Nodes	Use Global Setting
Src/Trg Selection	Automatic
Source	Program Controlled

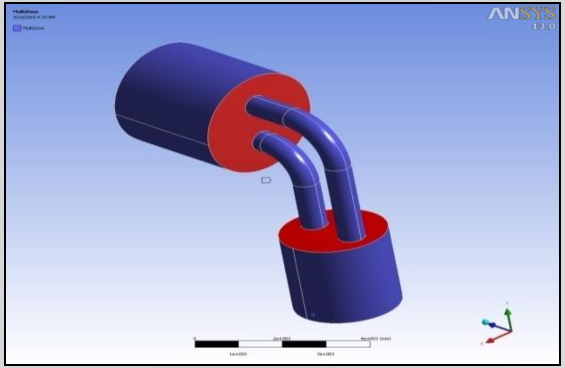
Details View of MultiZone Method



MultiZone Mesh

MultiZone Meshing

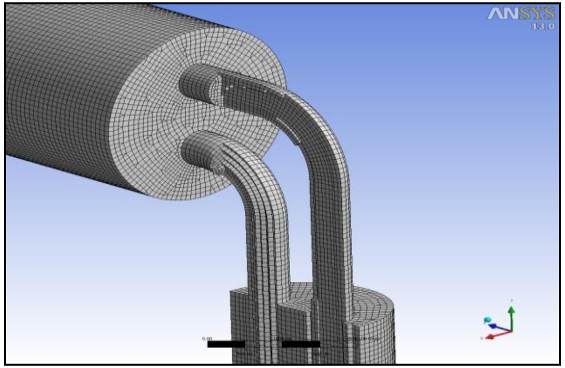
- **Control:**
  - **Src/Trg Selection - Manual**



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/Prism
Free Mesh Type	Not Allowed
Element Midside Nodes	Use Global Setting
Src/Trg Selection	Manual Source
Source	4 Faces

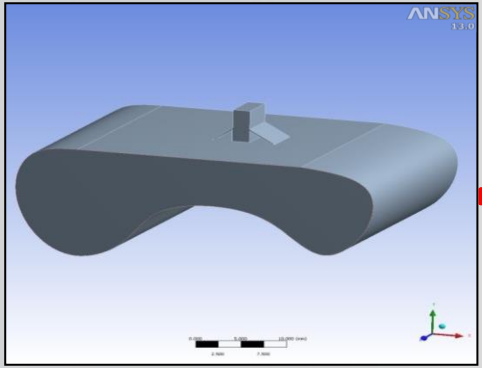
Details View of MultiZone Method



Cut section of MultiZone Mesh

MultiZone Meshing

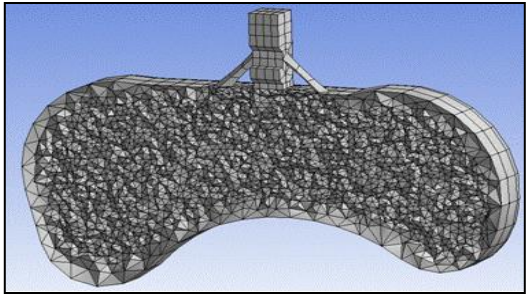
- **Control:**
  - **Free (unstructured) Mesh Type**



Geometry

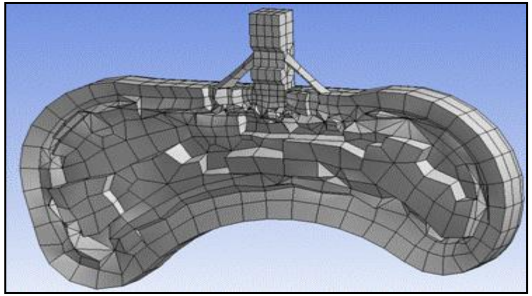
Method	MultiZone
Mapped Mesh Type	Hexa/Prism
Free Mesh Type	Tetra

Type: Tetra



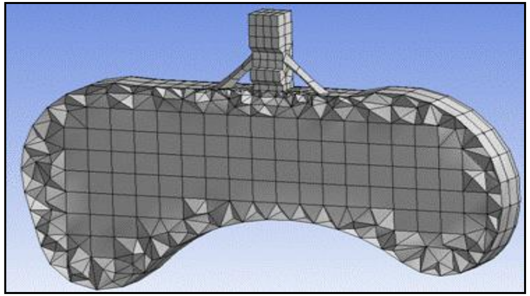
Method	MultiZone
Mapped Mesh Type	Hexa/Prism
Free Mesh Type	Hexa Dominant

Type: Hexa Dominant



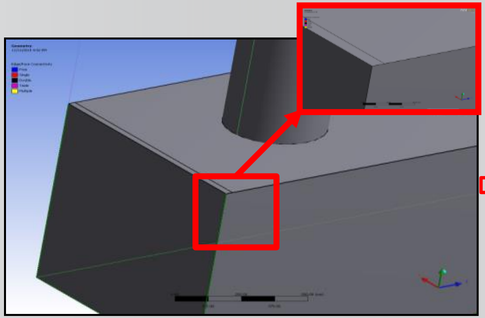
Method	MultiZone
Mapped Mesh Type	Hexa/Prism
Free Mesh Type	Hexa Core

Type: Hexa Core



MultiZone Meshing

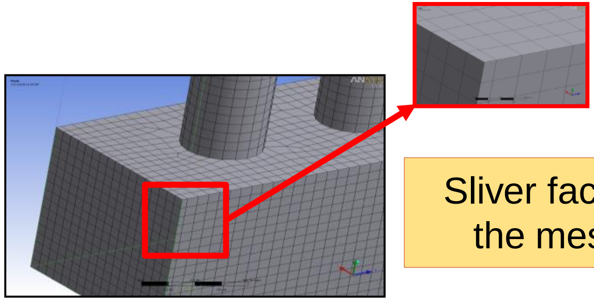
- **Control:**
  - **Local Defeaturing Tolerance**



Geometry with a sliver face

Advanced	
Mesh Based Defeaturing	Off
Minimum Edge Length	30. mm
Write ICEM CFD Files	No

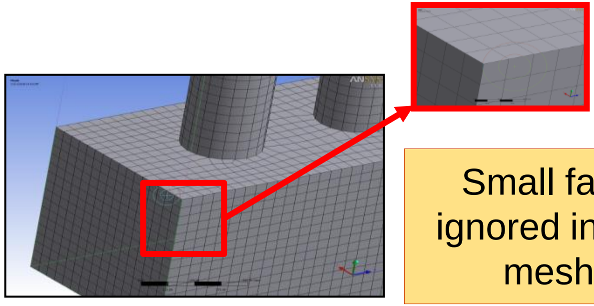
No Defeaturing



Sliver face in the mesh

Advanced	
Mesh Based Defeaturing	On
Defeaturing Tolerance	35. mm
Minimum Edge Length	30. mm
Write ICEM CFD Files	No

Using Defeaturing



Small face ignored in the mesh

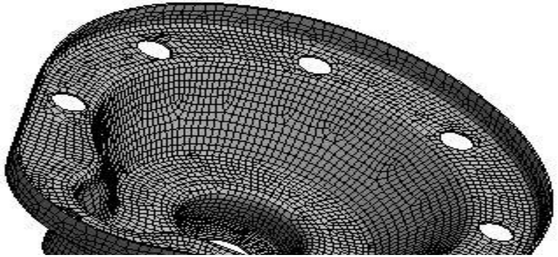
  - **Can be also controlled with global defeaturing tolerance**

Defeaturing	
Pinch Tolerance	Default (31.50 mm)
Generate Pinch on Refresh	Yes
Automatic Mesh Based Defeaturing	On
Defeaturing Tolerance	35. mm

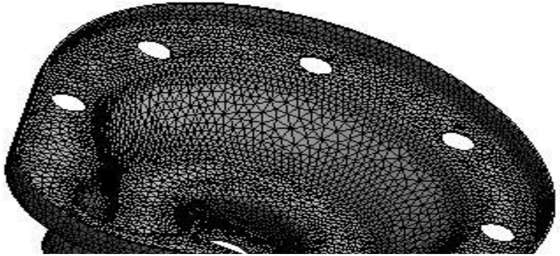
… Local Meshing Controls

Surface Body Methods:

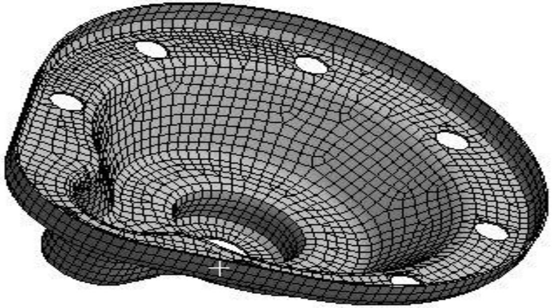
- **Quadrilateral Dominant (default): attempts to mesh with as many quad elements as possible, fills in with triangles.**



- **Triangles: all triangular shapes are used.**



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

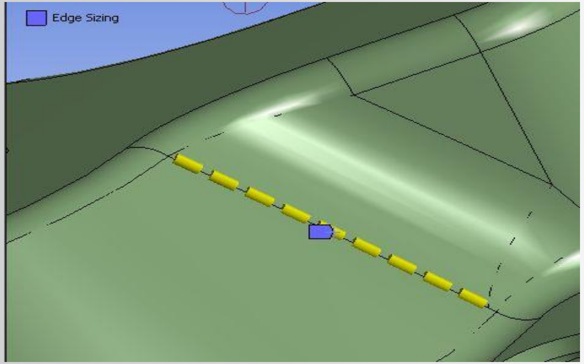


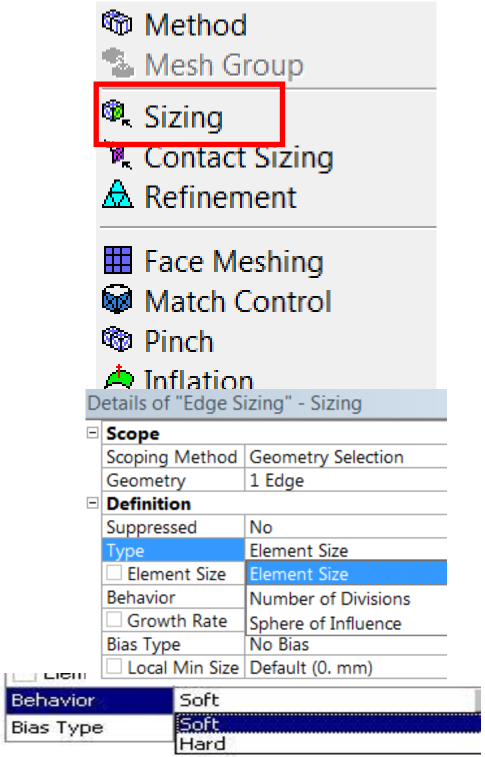
Note, each method contains a unique set of options in the details allowing additional configuration.

… Local Meshing Controls

Sizing (3 configurations):

- **Element Size (element edge length) OR Number of Divisions.**
- **Sphere of Influence (see next page)**
  - **"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

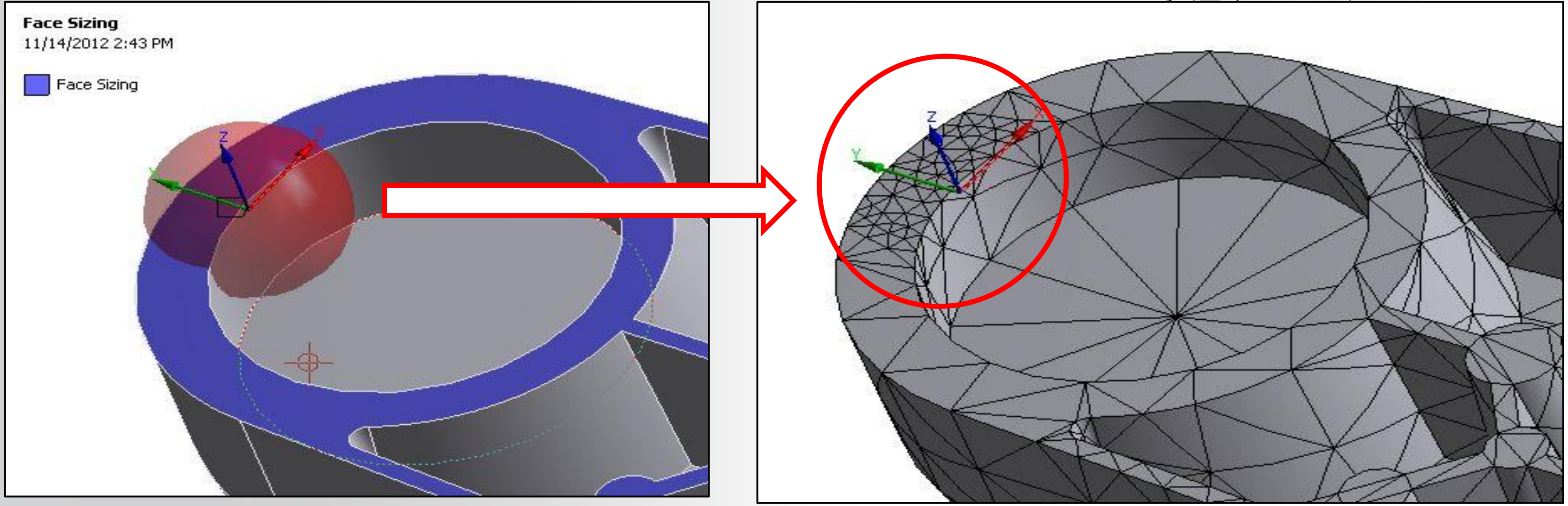
… Local Meshing Controls

Sphere of Influence:

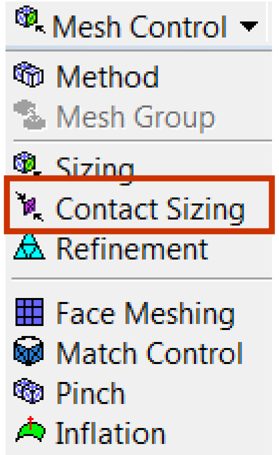
- **Center is located using a coordinate system.**
- **All scoped entities within the sphere are affected by size settings.**

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

Only the portion of the scoped face or body within the sphere is included in the scope of the mesh control.

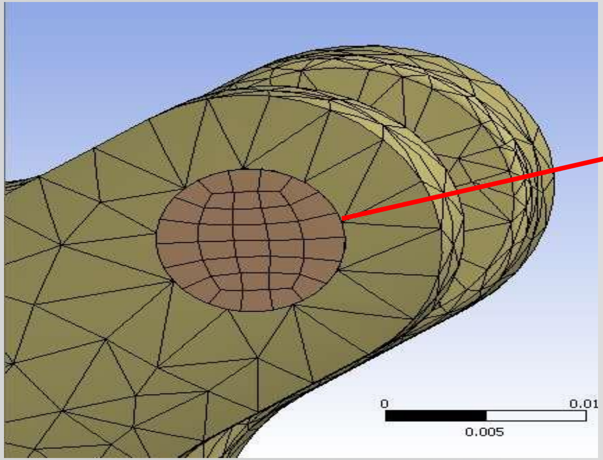


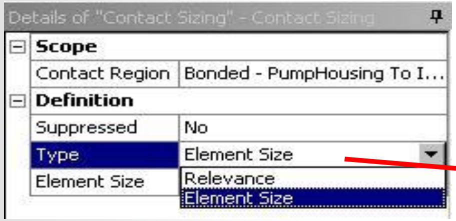
… Local Meshing Controls

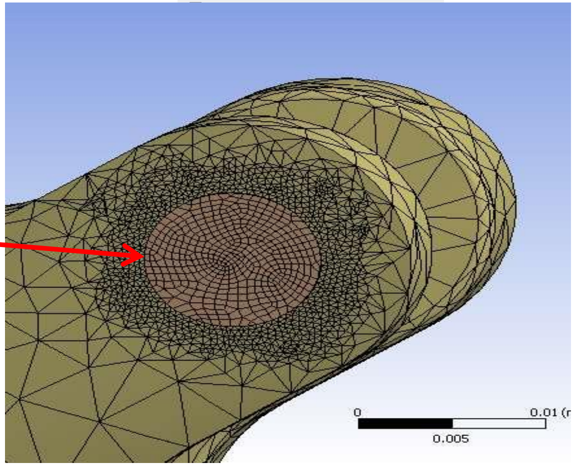


Contact Sizing: generates similar-sized elements on contact faces for face/face or face/edge contact regions.

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



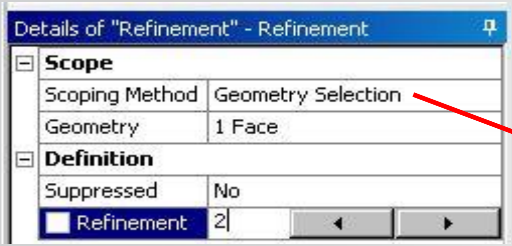




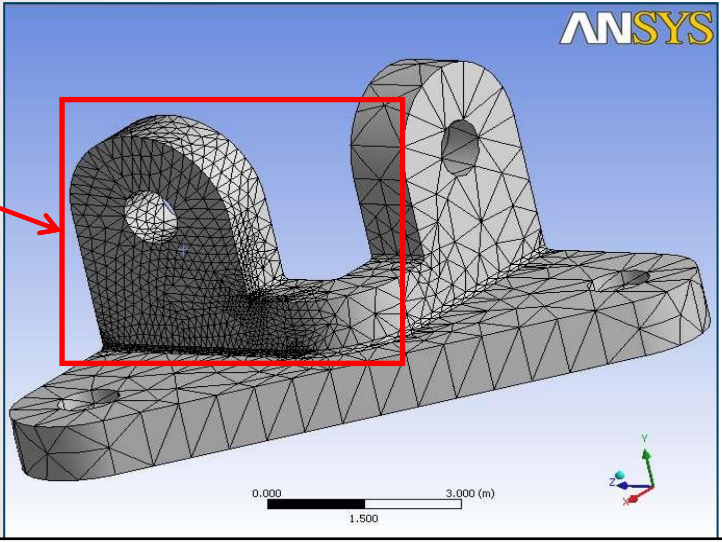
… Local Meshing Controls

Element 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).**



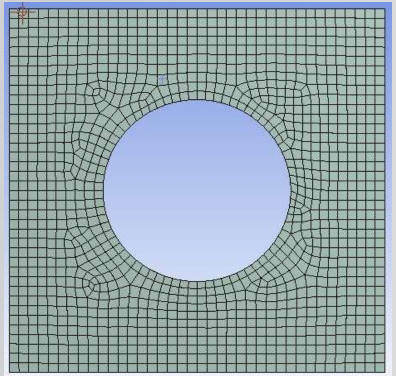
For example shown the scoped face has a refinement level of two.

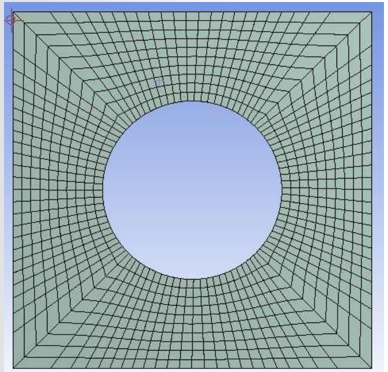


Note: the refinement method generally offers less control or predictability over the final mesh since an initial mesh is split. This splitting process may adversely affect other meshing controls as well.

… Local Meshing Controls

Mapped Face Meshing: generates structured meshes on surfaces:





Mapped quad or tri mesh also available for surface bodies.

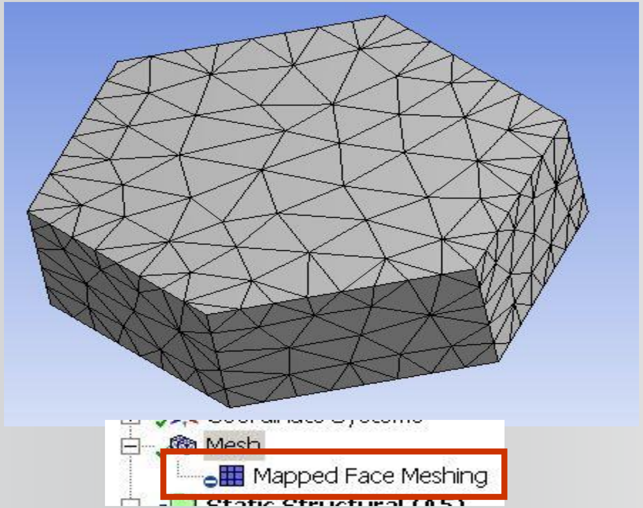
See next slide for advanced options . . . .

… Local Meshing Controls

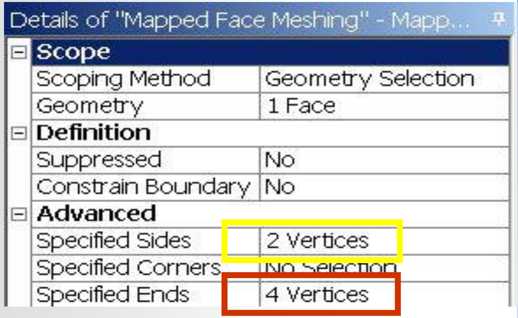
For some geometry mapping will fail if an obvious pattern is not recognized.

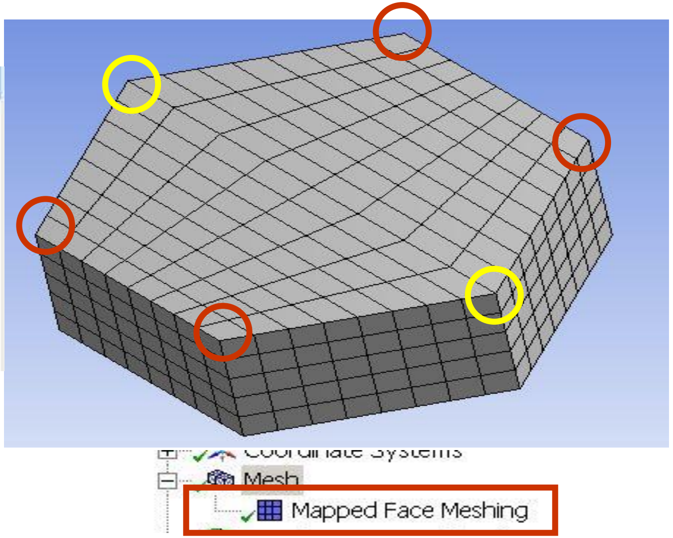
By specifying side, corner or end vertices a mapped face can be achieved.

See next page for additional examples.



Original mapping failed as indicated next to the mesh control.

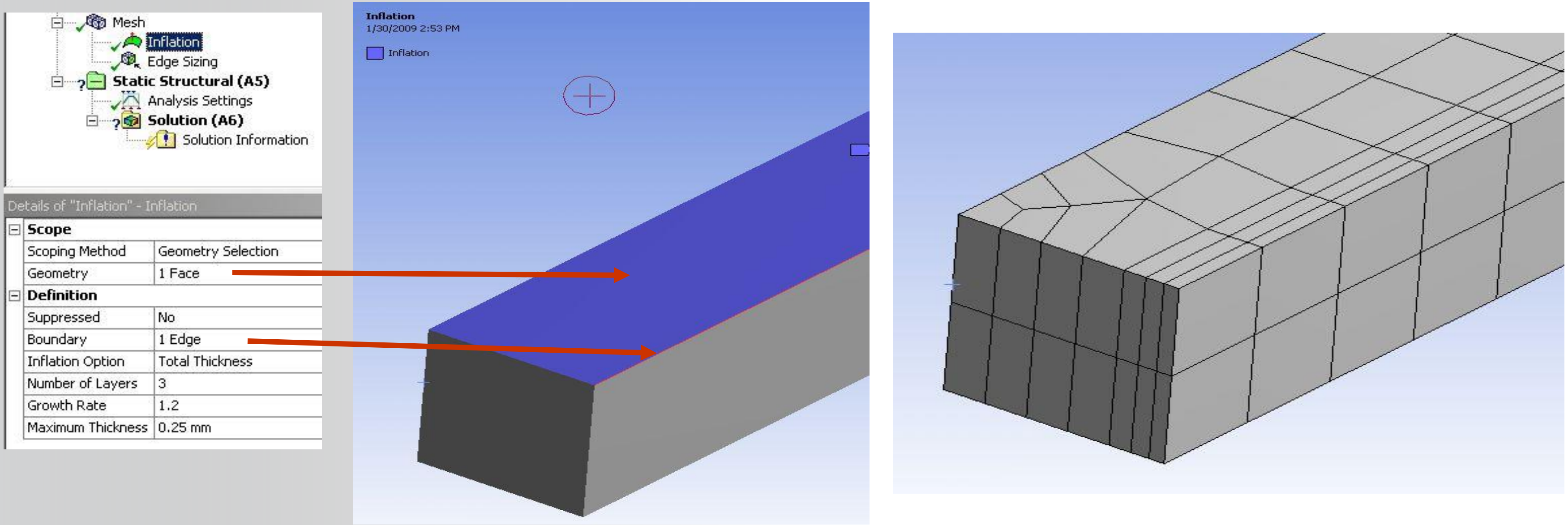




By setting side and end vertices the mapped mesh succeeds resulting in a uniform sweep.

… Local Meshing Controls

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

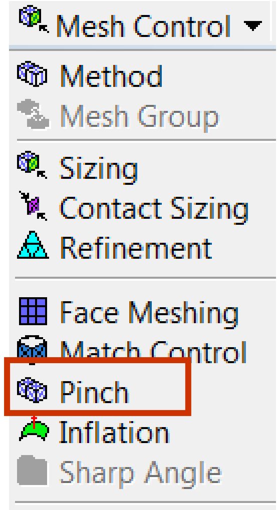


Note: Inflation is more often used in CFD and EMAG applications but may be useful for capturing stress concentrations etc. in structural applications.

… Local Meshing Controls

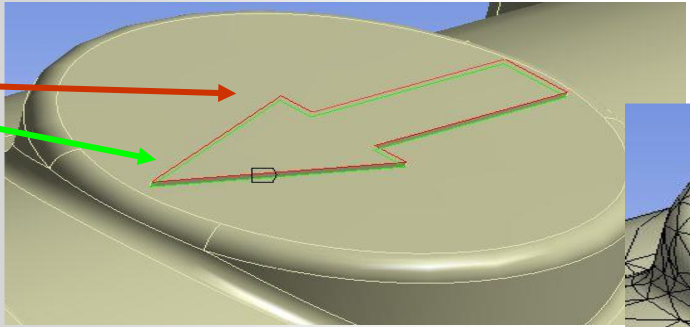
Pinch: allows the removal of small features by "pinching" out small edges and vertices.

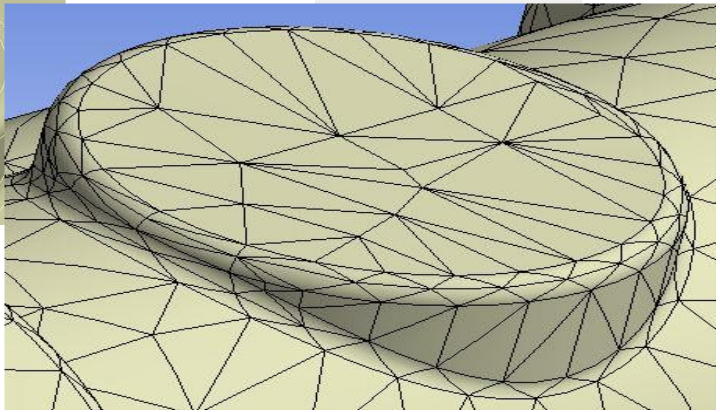
- **Master: geometry that retains the original geometry profile.**
- **Slave: geometry that changes to move toward the master.**
- **Can be automatic (mesh branch details) or local (add Pinch branch).**



Details of "Pinch" - Pinch

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



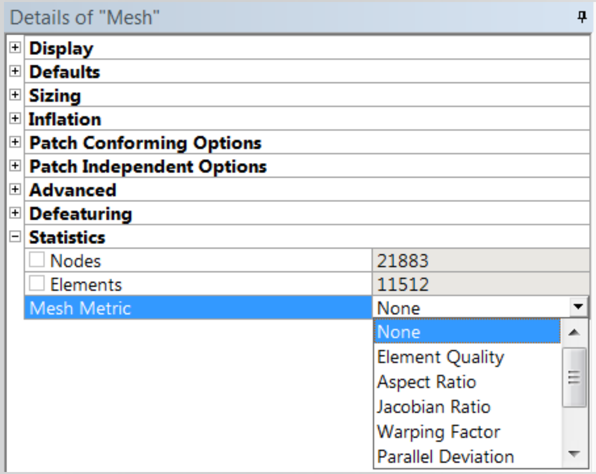


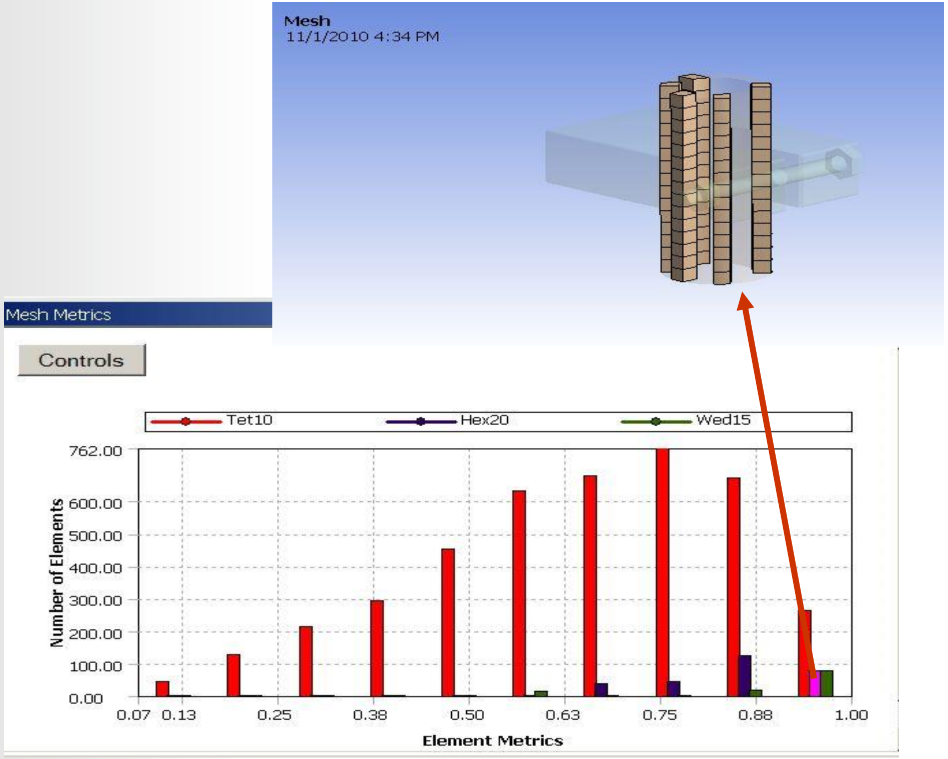
  February 27, 2015

C. Meshing Troubleshooting

Mesh Metric (requested in the “statistics” section):

- **Select individual bars in the graph to view the elements graphically.**



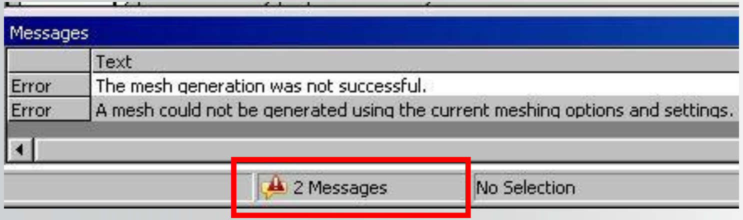


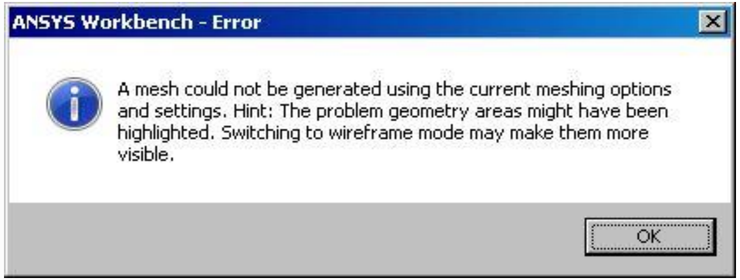
Note: each mesh metric is described in detail in the “Meshing User’s Guide”.

. . . Meshing Troubleshooting

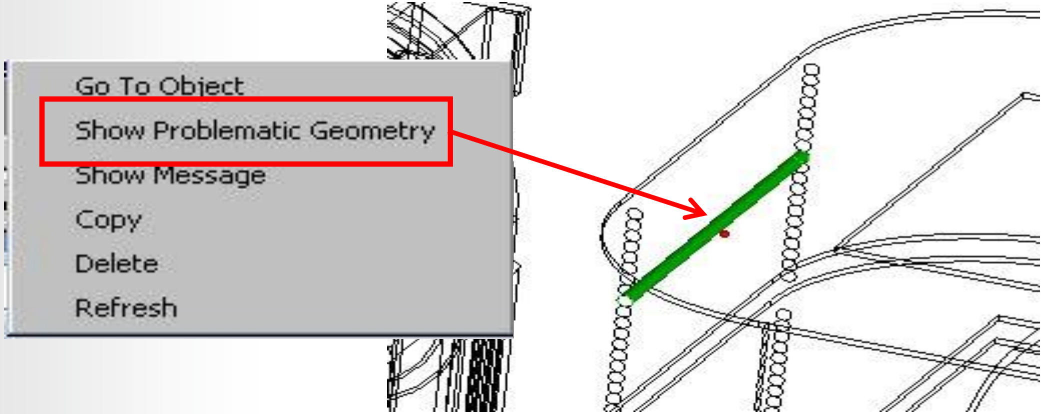
If the mesher is not able to generate a mesh an error message will be returned:

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



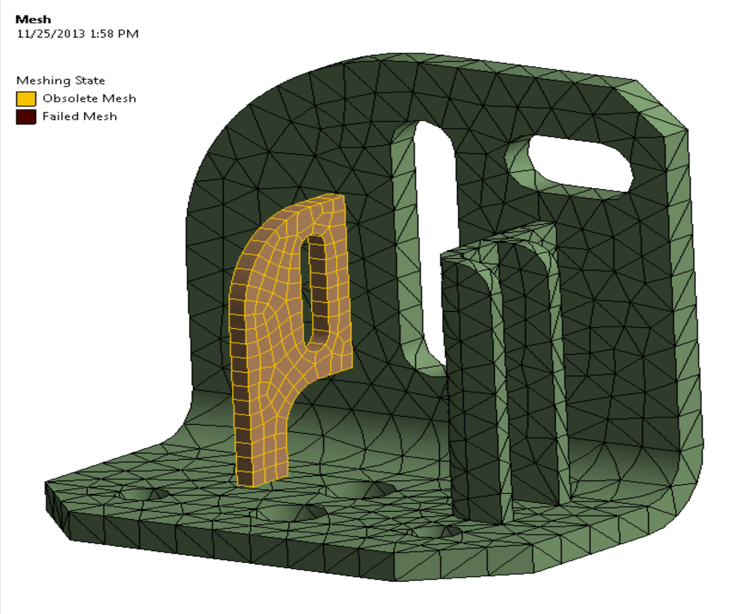


When possible Mechanical can graphically display the problem regions (RMB in the message window). Using a wireframe view will make finding these areas easier.



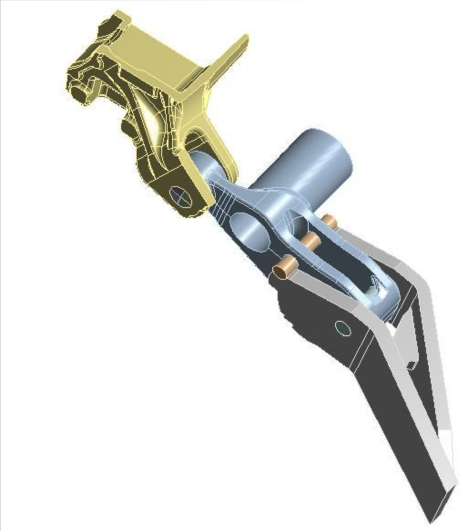
. . . Meshing Troubleshooting

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 colored yellow.



G. Workshop 4.1 – Mesh creation

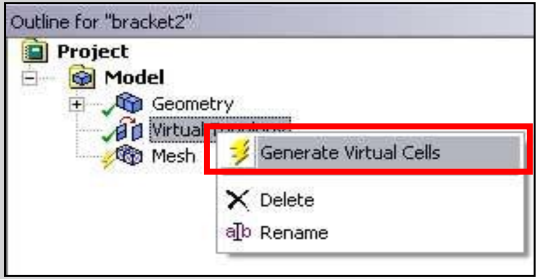
- **Workshop 4.1 – Mesh Creation**
- **Goal:**
  - **Use the various mesh controls to create your mesh.**

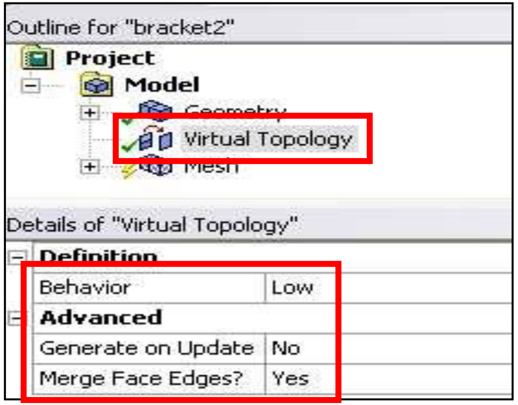


D. Virtual Topology

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. “Virtual Topology” branch is added below the “Model” .

- **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.**

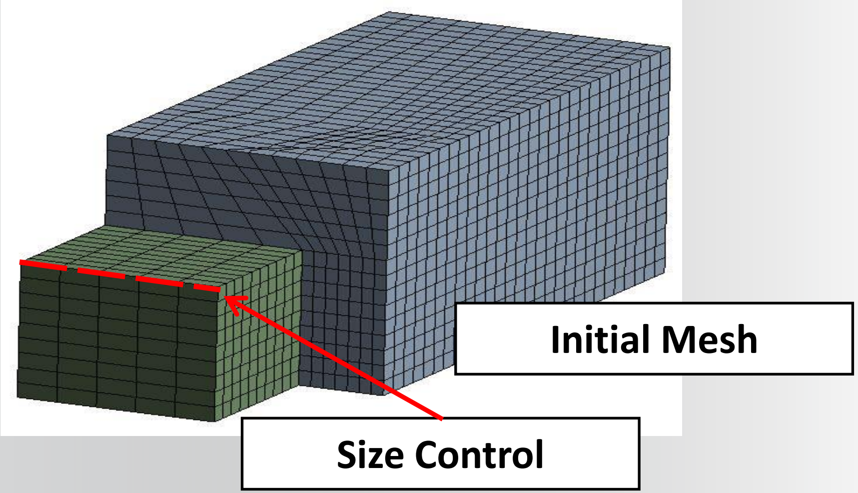


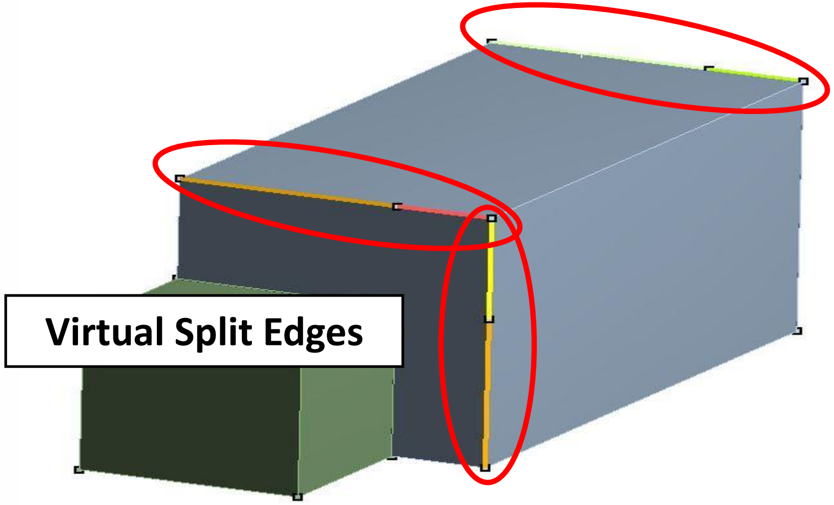


Several Examples Follow . . .

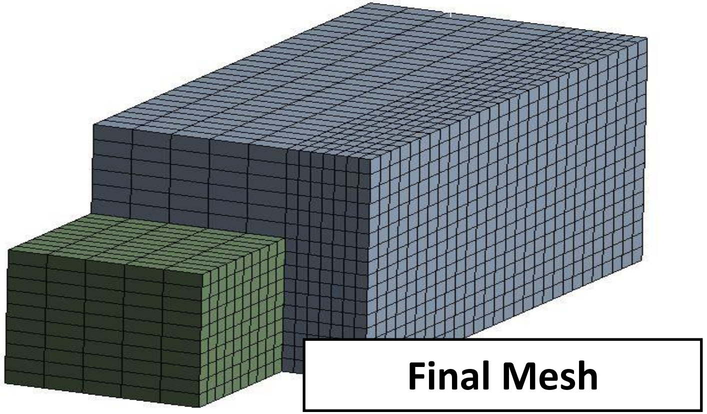
. . . Virtual Topology

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



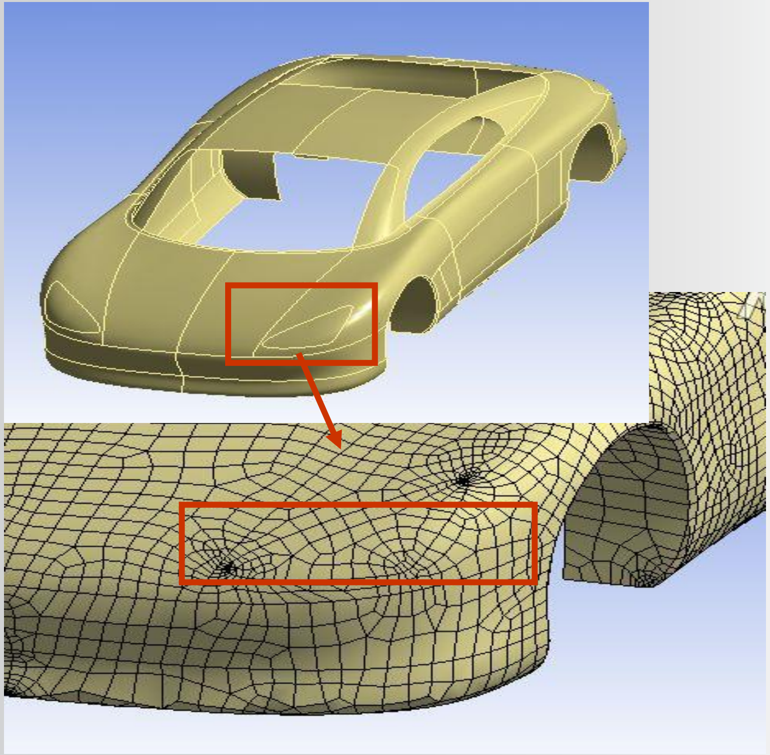


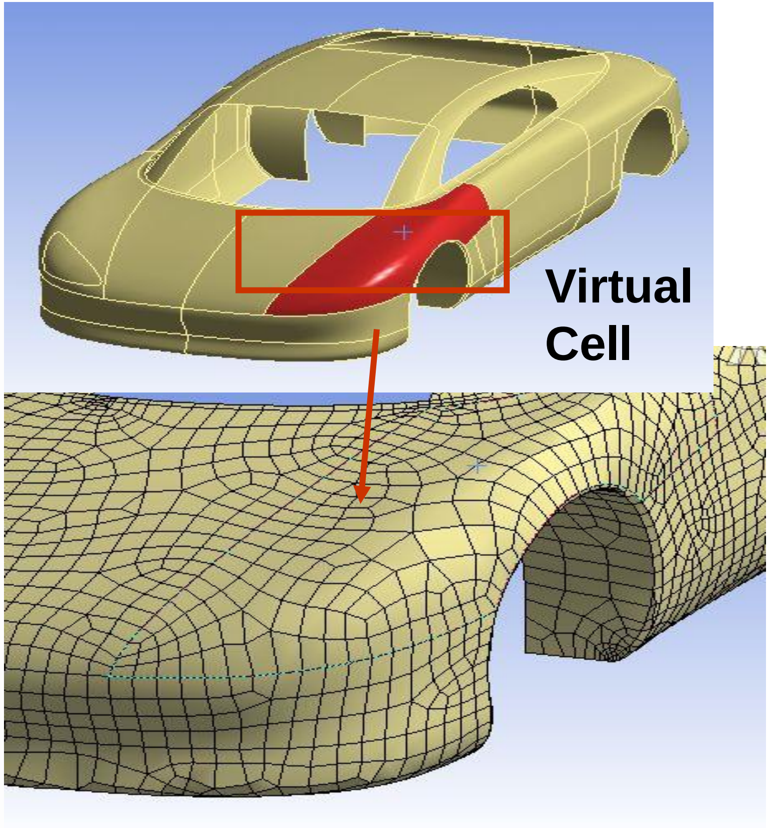
Shown in the upper right, 3 edges are “virtually” split to accommodate improved elements shapes.



. . . Virtual Topology

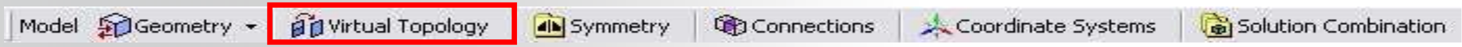
Surface Model Example:





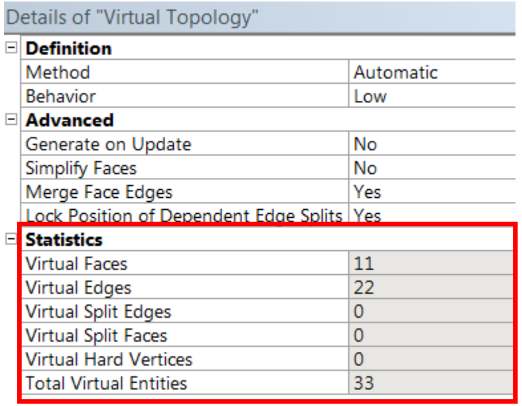
ANSYS

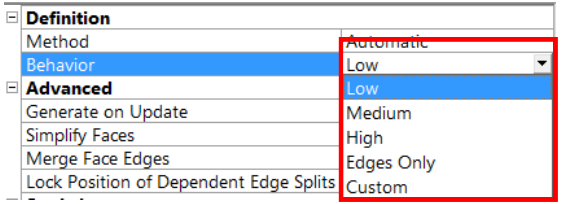
. . . Virtual Topology



"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





. . . Virtual Topology

In some instances it may be desirable to modify topology to allow a specific operation (meshing, load, support…).

- **Split face at vertices**

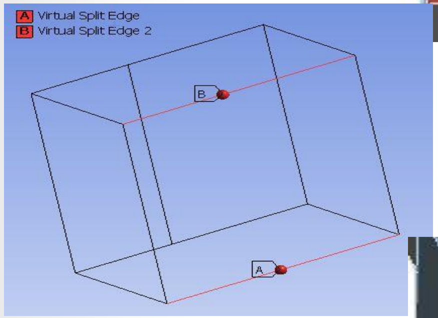
  Split Face at Vertices

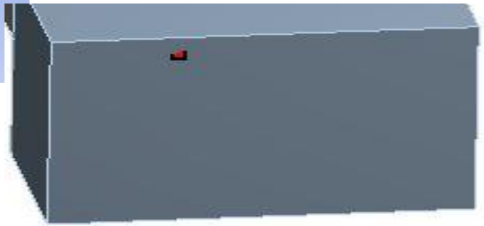
- **Split Edge**

  Split Edge at +   Split Edge

- **Add a hard vertex**

  Hard Vertex at +

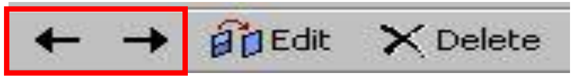


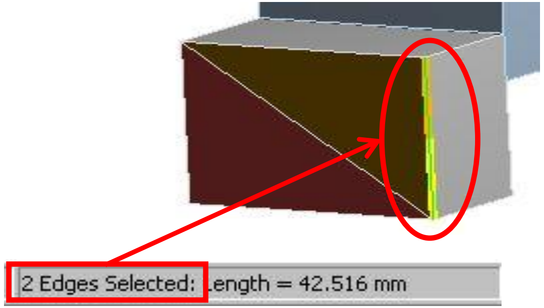


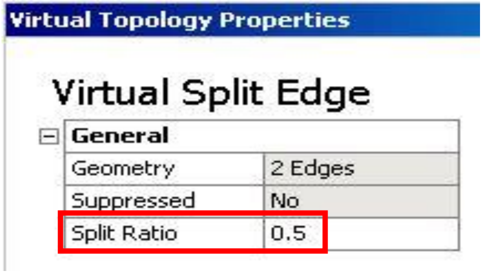
. . . Virtual Topology

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.





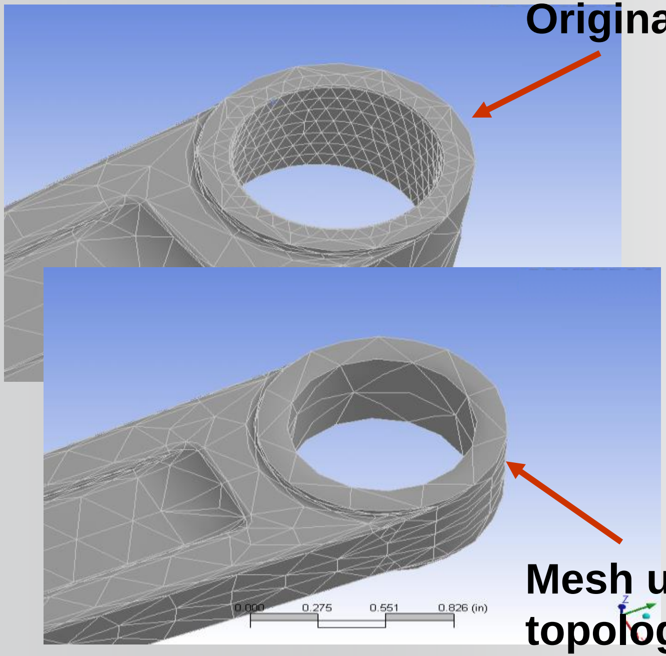


. . . Virtual Topology

Keep in mind that the topology can change!

- Example: a chamfer is added to the top surface in this virtual cell. The interior lines are not recognized anymore.

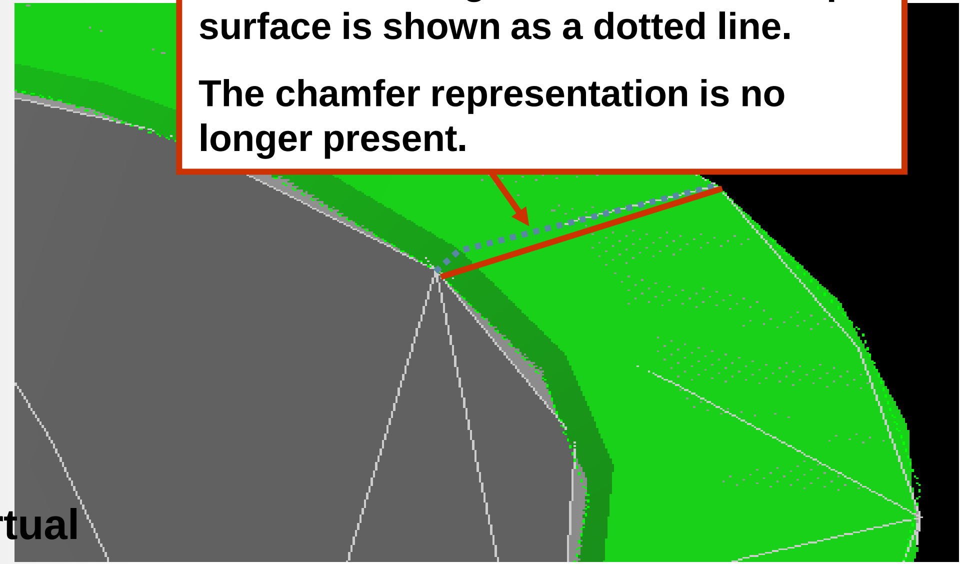
Original mesh



Mesh using virtual topology

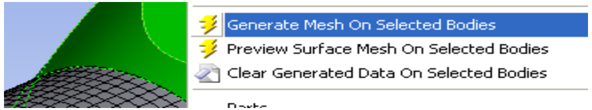
Element’s edge is shown as a solid line and the original chamfer and top surface is shown as a dotted line.

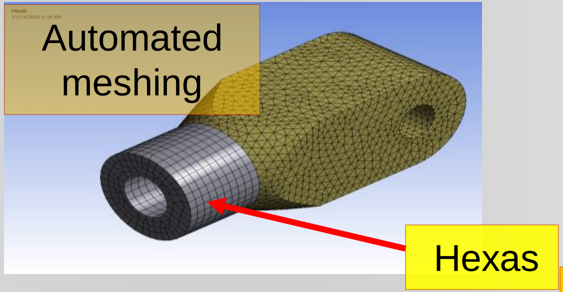
The chamfer representation is no longer present.

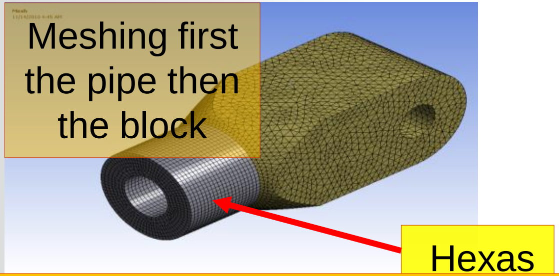


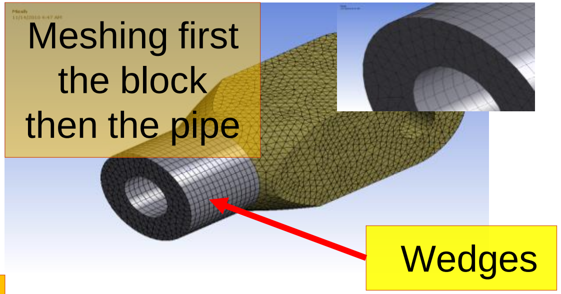
E. Direct Meshing

- Local meshing
- Bodies can be replaced/meshed/remeshed individually
- Subsequent bodies will use the attached face mesh
- The meshing results will depend on the meshing order
- RMB on the body(ies) and generate the mesh locally









Meshed with default meshing settings

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

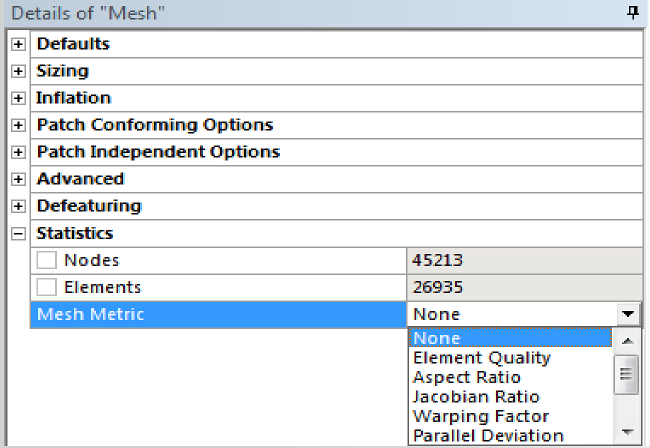
Statistics	
Nodes	24810
Elements	80178
Mesh Metric	Orthogonal Quality
Min	0.172748547853107
Max	0.999836842472065
Average	0.869178740711722
Standard Deviation	0.093626215921133

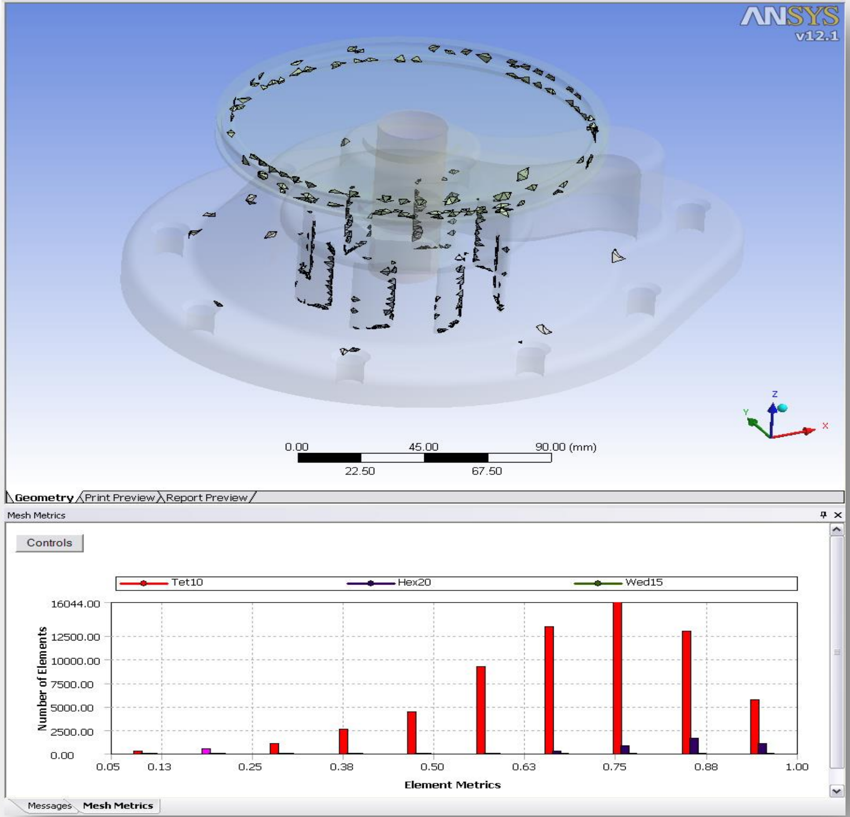
Statistics	
Nodes	13897
Elements	60932
Mesh Metric	Orthogonal Quality
Min	0.336740641439225
Max	0.999369993571416
Average	0.866831224319874
Standard Deviation	8.45868210177912E-02

F. Mesh Quality Criteria

You can check mesh quality using Mesh Metrics

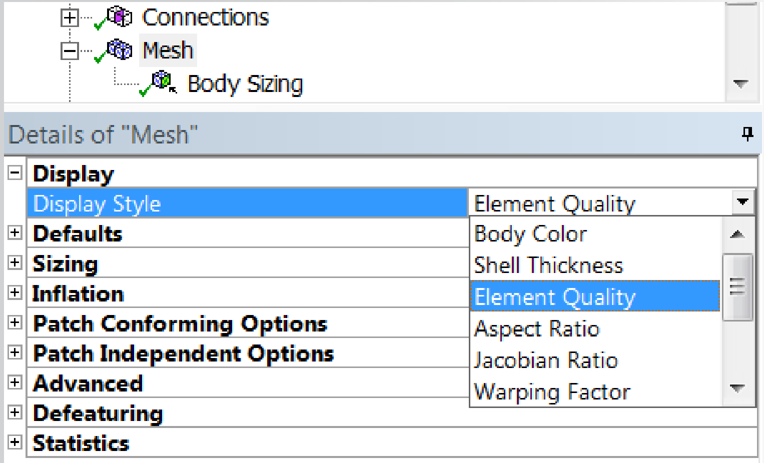
Remember : specific criterias for each physic

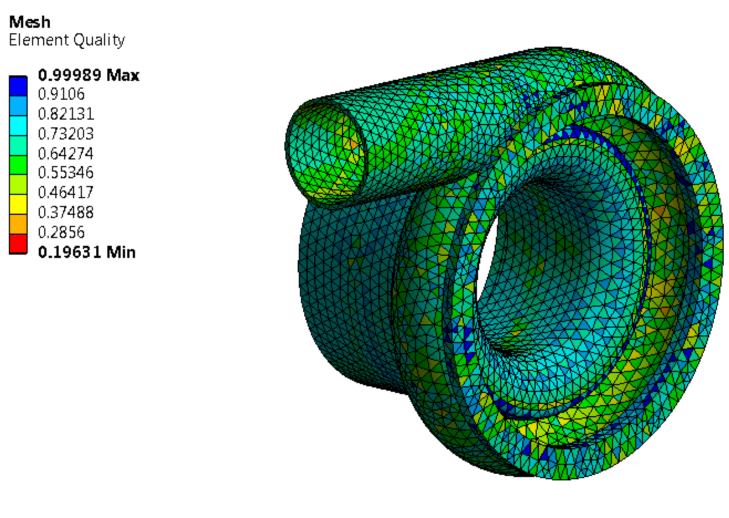




…Mesh Quality Criteria

Users can use display style option in details of mesh. This option able to display mesh in color by quality metrics





…Mesh Quality Criteria

Example of mesh metric : Element Quality :

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

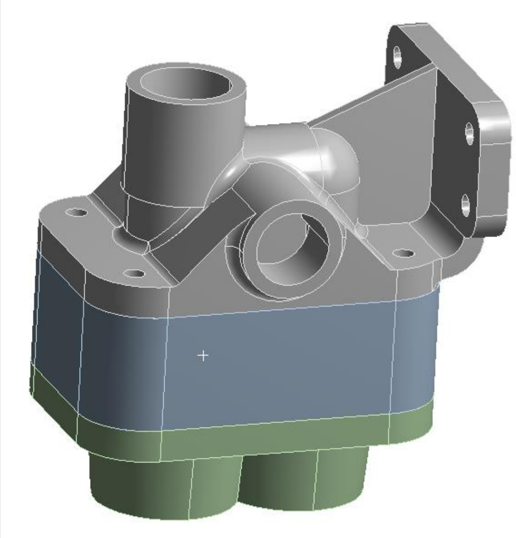
0
Bad

1
Perfect

Note : more information on other mesh metrics are in appendix

G. Workshop 4.2 – Mesh Control

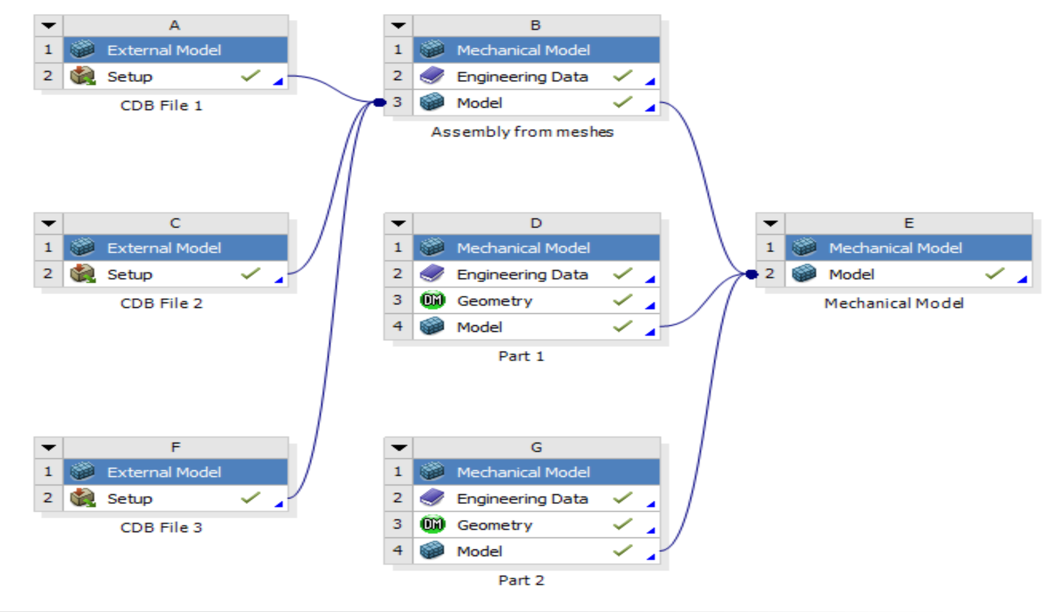
- **Workshop 4.2 – Mesh Control**
- **Goal:**
  - **Use the various mesh controls to enhance the mesh for the model.**



H. APPENDIX

- Model Assembly
- Other Mesh Quality Criteria

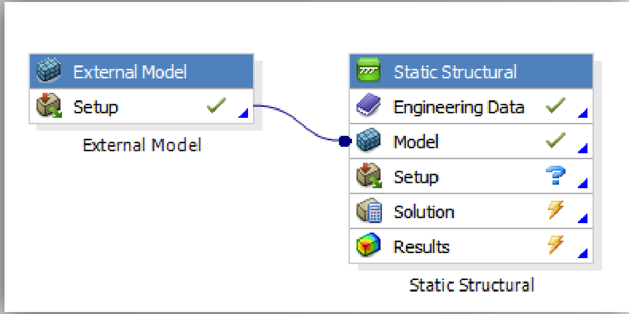
MODEL ASSEMBLY

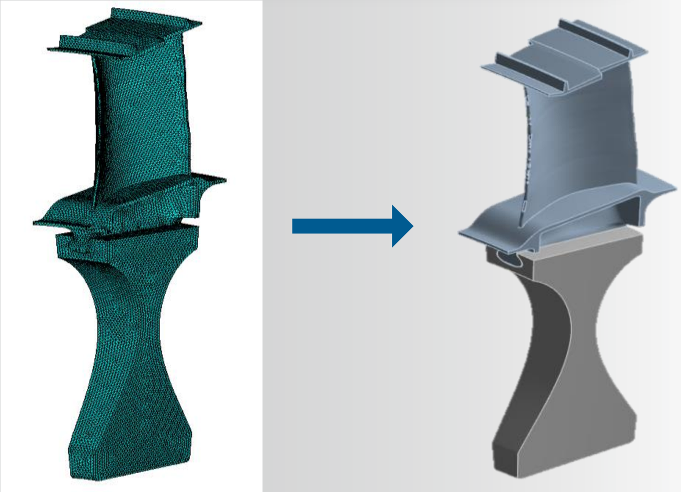


Geometry is not only the starting point of a Workbench based structural simulation.

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

…MODEL ASSEMBLY



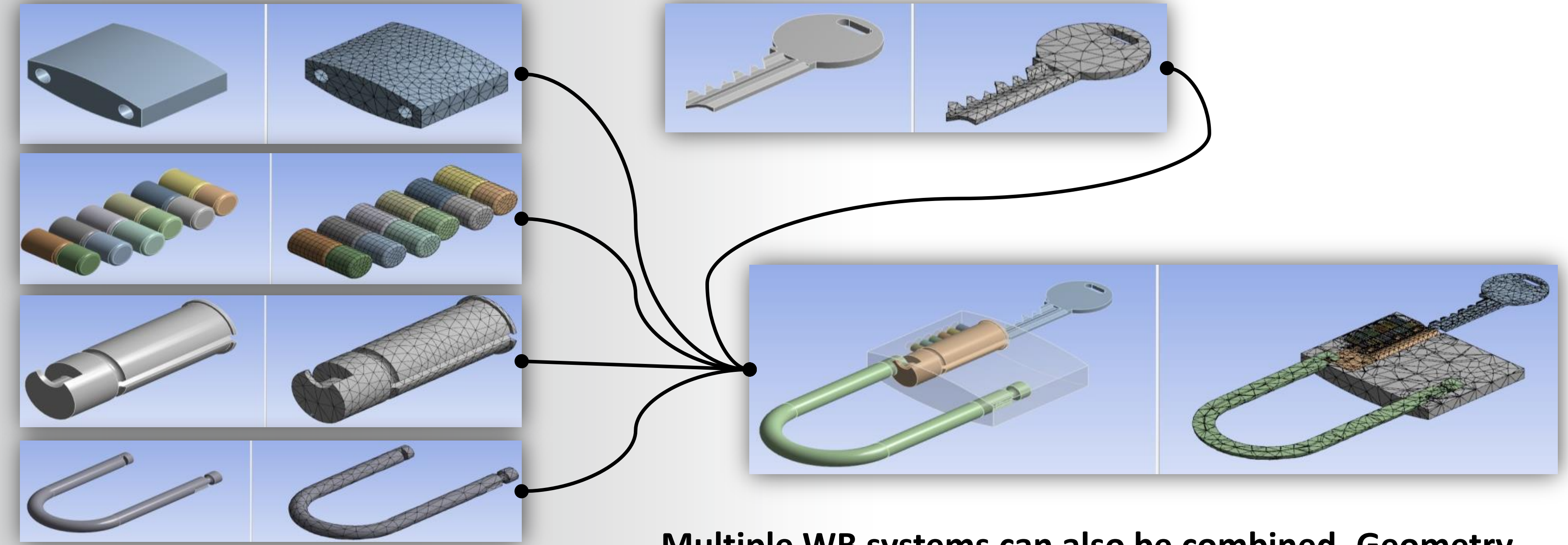


You can import mesh data (solids and shells) from CDB* file into Workbench using the External Model system and also scale, rotate or translate parts.

Contact detection will happen as if you are working with geometry data.

* .cdb is a file of model and database information which contains model data in terms of ANSYS input commands

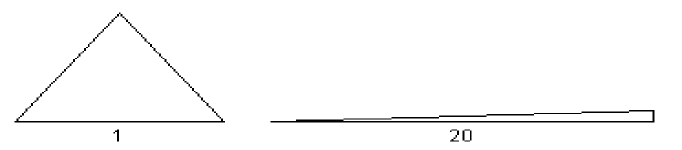
…MODEL ASSEMBLY

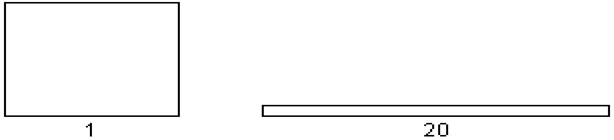


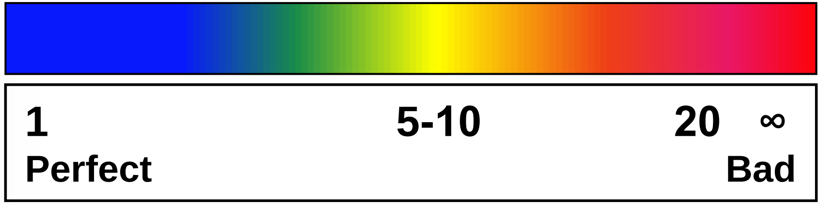
Multiple WB systems can also be combined. Geometry, Mesh and Named Selections are retrieved.

Other Mesh Quality Criteria

Aspect Ratio : Lengthening of element

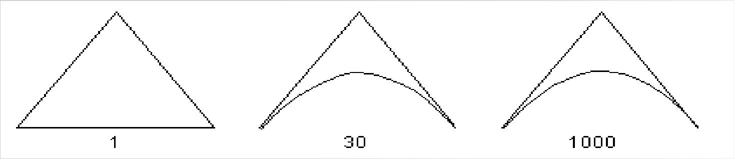


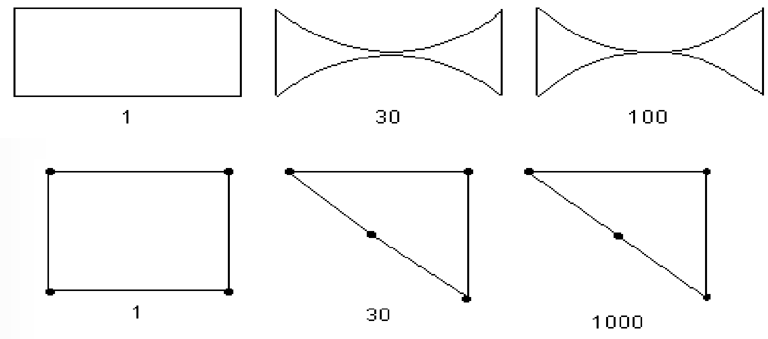




Other Mesh Quality Criteria

Jacobian Ratio :

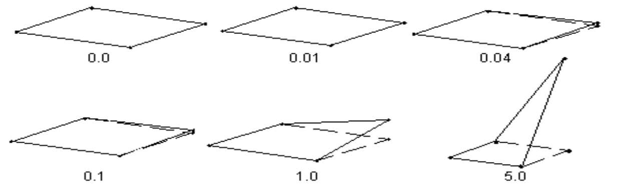


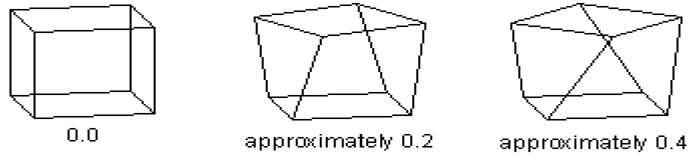


1	**10**	**30**	**∞**
Perfect		**Bad**	

Other Mesh Quality Criteria

Warping Ratio :



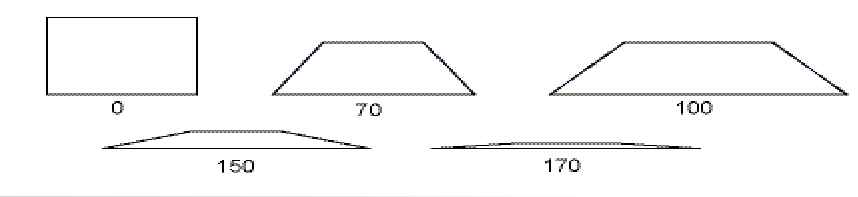


0 **0.1** **1** **∞**
Perfect **Bad**

0 **0.2** **0.4** **∞**
Perfect **Bad**

Other Mesh Quality Criteria

Parallel Deviation :

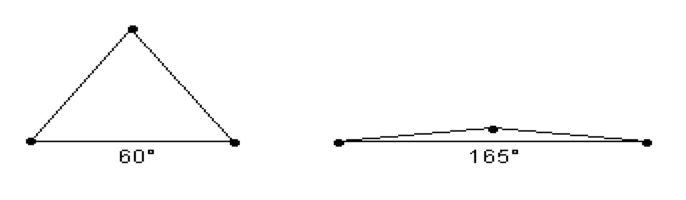


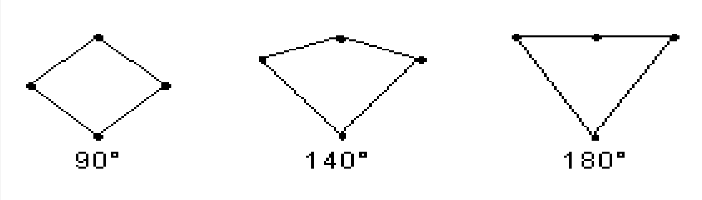
0 **170**

Perfect **Bad**

Other Mesh Quality Criteria

Maximum Corner Deviation :



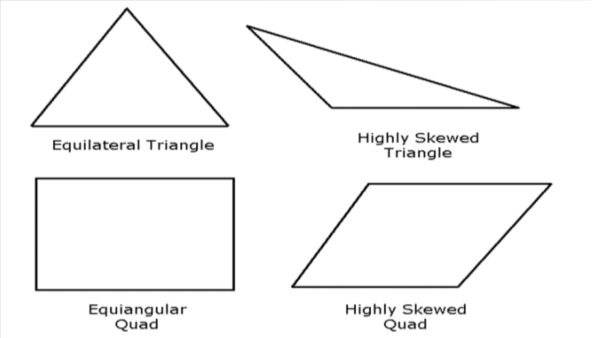


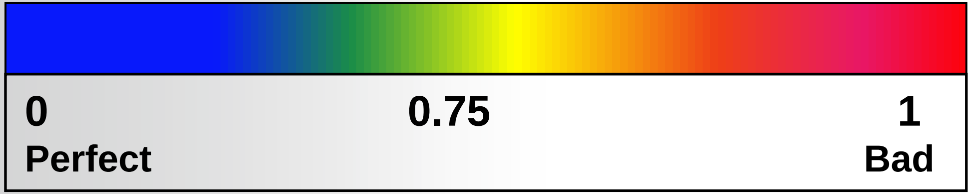
60 **165**
Perfect **Bad**

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

Other Mesh Quality Criteria

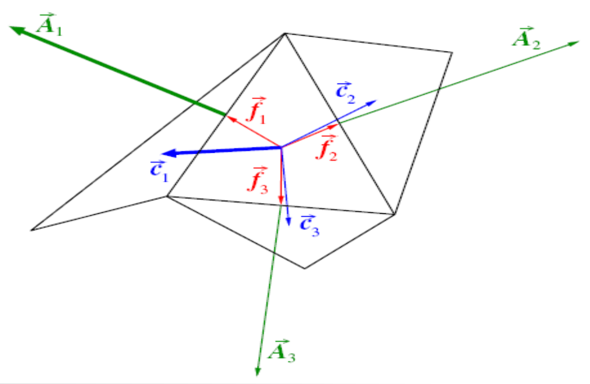
Skewness





Other Mesh Quality Criteria

Orthogonal Quality :



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

