Lecture 4

Meshing in Mechanical

Introduction to ANSYS Mechanical
In this chapter controlling meshing operations is described.

Topics:
A. Global Meshing Controls
B. Local Meshing Controls
C. Meshing Troubleshooting
D. Virtual Topology

The capabilities described in this section are generally applicable to the ANSYS DesignSpace Entra licenses and above and are noted in the lower-left hand tables.
Meshing in Mechanical

- The nodes and elements representing the geometry model make up the mesh:
  - A “default” mesh is automatically generated during initiation of the solution.
  - The user can “generate” the mesh prior to solving to verify mesh control settings.
  - A finer mesh produces more precise answers but also increases CPU time and memory requirements.
A. Global Meshing Controls

- Physics Based Meshing allows the user to specify the mesh based on the physics to be solved. Choosing the physics type will set controls such as:
  - Solid element mid-side nodes
  - Element shape checking
  - Transitioning

- Physics preferences can be:
  - Mechanical
  - Electromagnetics
  - CFD
  - Explicit

- Note: Some mesh controls are intended for non-Mechanical applications (CFD, EMAG, etc). Only mechanical mesh controls are discussed in this course.
• Basic meshing controls are available under the “Defaults” group in the “Mesh” branch
  – The user has control with a single slider bar
    • “Relevance” setting between –100 and +100

- Relevance = coarse mesh

+ Relevance = fine mesh
Introduction to ANSYS Mechanical

... Global Meshing Controls

- **Sizing Section:**
  - The controls in this group set the basic size defaults for the initial mesh. Local controls (described later), can be used to override these values in specific regions of the model.
  - 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 element size used for the entire model.
- **Initial Size seed:** Initial mesh size is based either on the entire assembly or on each individual part.
- **Smoothing:** Attempts to improve element quality by moving nodes. Number of smoothing iterations can be controlled (Low, Medium, High).
- **Transition:** Controls the rate at which adjacent elements will grow (Slow, Fast)

### Details of "Mesh"

<table>
<thead>
<tr>
<th>Defaults</th>
<th>Mechanical</th>
</tr>
</thead>
<tbody>
<tr>
<td>Relevance</td>
<td>0</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Sizing</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Use Advanced Size Function</td>
<td>Off</td>
</tr>
<tr>
<td>Relevance Center</td>
<td>Coarse</td>
</tr>
<tr>
<td>Element Size</td>
<td>Default</td>
</tr>
<tr>
<td>Initial Size Seed</td>
<td>Active Assembly</td>
</tr>
<tr>
<td>Smoothing</td>
<td>Medium</td>
</tr>
<tr>
<td>Transition</td>
<td>Fast</td>
</tr>
<tr>
<td>Span Angle Center</td>
<td>Coarse</td>
</tr>
<tr>
<td>Minimum Edge Length</td>
<td>10.0 mm</td>
</tr>
</tbody>
</table>
Advanced Size Functions: 4 settings to control basic mesh sizing.

- **Curvature**: The curvature size function examines curvature on edges and faces and sets element sizes so as not to violate the maximum size or the curvature angle (automatically computed or defined by the user).

- **Proximity**: The proximity size function allows you to specify the minimum number of element layers created in regions that constitute “gaps” in the model (features).

- **Fixed**: The fixed size function does not refine the mesh based on curvature or proximity. Rather, you specify minimum and maximum sizes and gradation is provided between sizes based on a specified growth rate.

- **Note**: Users may accept default settings for these options or specify their own (described next).
**Curvature settings:**

- **Normal angle:** the maximum allowable angle that one element edge is allowed to span (default based on relevance and span angle center settings).
- **Min Size:** the minimum element edge size that the mesher will create.
- **Max Face Size:** Maximum size the surface mesher will allow.
- **Max Size:** Maximum size the volume mesher will allow.
- **Growth Rate:** Specifies the increase in element size for each succeeding layer progressing from an edge. A value of 1.2 represents a 20% increase. Settings from 1 to 5 with a default determined by relevance and transition settings.

### Sizing

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

Curvature = 20 deg.  
Curvature = 75 deg.
**Global Meshing Controls**

- **Proximity Settings:**
  - **Proximity Accuracy:** Set between 0 and 1 (0.5=default). Controls the search range used with the max size and cells across gap settings. A setting of 0 is faster, a setting of 1 is more accurate.
  - **Num Cells Across Gap:** specifies the number of element layers to be generated in the gap sections (i.e. between features).

<table>
<thead>
<tr>
<th>Sizing</th>
<th>Use Advanced Size Function</th>
<th>On: Proximity</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Relevance Center</td>
<td>Coarse</td>
</tr>
<tr>
<td></td>
<td>Initial Size Seed</td>
<td>Active Assembly</td>
</tr>
<tr>
<td></td>
<td>Smoothing</td>
<td>Medium</td>
</tr>
<tr>
<td></td>
<td>Transition</td>
<td>Fast</td>
</tr>
<tr>
<td></td>
<td>Span Angle Center</td>
<td>Coarse</td>
</tr>
<tr>
<td></td>
<td>Proximity Accuracy</td>
<td>0.5</td>
</tr>
<tr>
<td></td>
<td>Num Cells Across Gap</td>
<td>Default (3)</td>
</tr>
<tr>
<td></td>
<td>Min Size</td>
<td>Default (5.8756e-002 mm)</td>
</tr>
<tr>
<td></td>
<td>Max Face Size</td>
<td>5.0 mm</td>
</tr>
<tr>
<td></td>
<td>Max Size</td>
<td>Default (11.7510 mm)</td>
</tr>
<tr>
<td></td>
<td>Growth Rate</td>
<td>Default (1.850 )</td>
</tr>
<tr>
<td></td>
<td>Minimum Edge Length</td>
<td>1.250 mm</td>
</tr>
</tbody>
</table>

[Image showing meshing with two different settings for Num Cells = 2 and Num Cells = 5]
Introduction to ANSYS Mechanical

... 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.

- **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.
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
  - Inflation Control
  - Pinch Control
  - Gap Tool (EMAG only, not covered)
Method Control: Provides the user with options as to how solid bodies are meshed:

- Automatic (default):
  - Body will be swept if possible. Otherwise, the “Patch Conforming” mesher under “Tetrahedrons” is used.

Continued . . .
... Local Meshing Controls : Method (continued)

- **Tetrahedrons:**
  - An all Tetrahedron mesh is generated.

- **Patch Conforming:**
  - All face boundaries are respected when mesh is created.

- **Patch Independent Meshing:**
  - Faces and their boundaries may or may not be respected during meshing operations.
  - The exception is when a boundary condition is applied to a surface, its boundaries are respected.
**Hex Dominant**: Creates a free hex dominant mesh. Useful for meshing bodies that cannot be swept.

- Recommended for meshing bodies with large interior volumes.
- Not recommended for thin or highly complex shapes.

**Free Face Mesh Type**: determines the mesh shape to be used to fill the body (Quad/Tri or All Quad).

### 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:
- Tetrahedrons – 443 (9%)
- Hexahedron – 2801 (62%)
- Wedge – 124 (2%)
- Pyramid – 1107 (24%)
Introduction to ANSYS Mechanical

... Local Meshing Controls : Method (continued)

- Sweep :
  - **Sweep-mesh (hex and possible wedge) elements.**
  - **Type** : Number of Divisions or Element Size in the sweep direction.
  - **Sweep Bias Type** : Bias spacing in sweep direction.
  - **Src/Trg Selection** : Manually select the start/end faces for sweeping or allow the mesher to choose.
  - **Automatic/Manual Thin Model** – One hex or wedge through the thickness. Can choose between Solid Shell (SOLSH190) element and a Solid element (Solid185). A solid shell element is useful for thin structures with a single element through the thickness (e.g. sheet metal).

---

### Details of "Sweep Method" - Method

<table>
<thead>
<tr>
<th>Scope</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>Scoping Method</td>
<td>Geometry Selection</td>
</tr>
<tr>
<td>Geometry</td>
<td>1 Body</td>
</tr>
</tbody>
</table>

| 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 |

---

### Details of "Sweep Method" - Method

<table>
<thead>
<tr>
<th>Scope</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>Scoping Method</td>
<td>Geometry Selection</td>
</tr>
<tr>
<td>Geometry</td>
<td>1 Body</td>
</tr>
</tbody>
</table>

| Suppressed | No |
| Method | Sweep |
| Element Midside Nodes | Dropped |
| Src/Trg Selection | Automatic Thin |
| Source | Program Controlled |
| Free Face Mesh Type | Quad/Tri |
| Sweep Num Divs | Default |
| Element Option | Solid Shell |
• **MultiZone Method:**
  - A patch independent mesher that automatically decomposes solid geometry to accomplish sweep meshing (like a user might slice a model for meshing).

• **Mapped Mesh Type:** controls the shapes used for fill regions.

• **Free Mesh Type:** if set, allows tet meshes in the fill regions. Can set to “not allowed” if all hex is desired.
• Sizing:
  – “Element Size” specifies average element edge length or number of divisions (choices depend on geometry selection).
  – “Soft” control may be overridden by other mesh controls. “Hard” may not.
  – Mesh biasing is available.

• Sphere of Influence sizing, see next page.

<table>
<thead>
<tr>
<th>Entity</th>
<th>Element Size</th>
<th># of Elem. Division</th>
<th>Sphere of Influence</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bodies</td>
<td>x</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>Faces</td>
<td>x</td>
<td></td>
<td>x</td>
</tr>
<tr>
<td>Edges</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>Vertices</td>
<td>X</td>
<td></td>
<td>X</td>
</tr>
</tbody>
</table>
• **Sphere of Influence:**
  - Center is located using local coordinate system.
  - All scoped entities within the sphere are affected by size settings.

“Sphere of Influence” (shown in red) has been defined. Elements lying in that sphere for that scoped entity will have a given average element size.
• Contact Sizing: generates similar-sized elements on contact faces for face/face or face/edge contact region.
  - “Element Size” or “Relevance” can be specified.
  - Choose “Contact Sizing” from the “Mesh Control” menu and specify the contact region.
  - Or drag and drop a Contact Region object onto the “Mesh” object.

In this example, the contact region between the two parts has a Contact Sizing Type Relevance is specified. Note that the mesh is now consistent at the contact region.
• Element refinement divides existing mesh
  - An ‘initial’ mesh is created with global and local size controls first, then element refinement is performed at the specified location(s).
  - Refinement range is 1 to 3 (minimum to maximum). Refinement splits the edges of the elements in the ‘initial’ mesh in half. Refinement level controls the number of iterations this is performed.

For example shown, the left side has refinement level of 2 whereas the right side is left untouched with default mesh settings.
• Mapped Face Meshing: generates structured meshes on surfaces:
  – In example below, mapped face meshing on the outer face provides a more uniform mesh pattern.

• Mapped quad or tri mesh also available for surface bodies.
• See next slide for advanced options . . . .
Local Mesh 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.

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 Mesh 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 Mesh Controls

- Pinch: allows the removal of small features by “pinching” out small edges and vertices (only).
  - Master: geometry that retains the original geometry profile.
  - Slave: geometry that changes to move toward the master.
  - Can be automatic (Mesh level) or local (add Pinch branch).

Note: a global pinch control can be set in the mesh branch details “Defeaturing” section.
C. Meshing Troubleshooting

- Mesh Metrics: can be 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” of the ANSYS documentation.
• If the mesher is not able to generate satisfactory elements, an error message will be returned:

- The problematic geometry will be highlighted on the screen, and a named selection group “Problematic Geometry” will be created, so the user may review the model.
• Meshing failures can be caused by a number of things:
  – Inconsistent sizing controls specified on surfaces, which would result in the creation of poorly-shaped elements
  – Difficult CAD geometry, such as small slivers or twisted surfaces
  – Stricter shape checking (“Aggressive” setting in Mesh branch)

• Some ways to avoid meshing failures:
  – Specify more reasonable sizing controls on geometry
  – Specify smaller sizing controls to allow the mesher to create better-shaped elements
  – In the CAD system, use hidden line removal plots to see sliver or unwanted geometry and remove them
  – Use virtual cells to combine sliver or very small surfaces
    – *This option will be discussed next*
D. Virtual Topology

• Virtual Topology: combines surfaces and edges for meshing control:
  - “Virtual Topology” branch is added to the “Model” branch.
  - A “Virtual Cell” is a group of adjacent surfaces that “acts” as a single surface.
  - Interior lines of original surfaces will no longer be honored by meshing process.
  - For other operations such as applying Loads and Supports, a virtual cell can be referenced as a single entity.
  - Virtual cells can be generated automatically via RMB:
    • The “Behavior” controls the aggressiveness of the “Merge Face Edges?” setting for auto generation.

• Example . . .
• Consider the example below:
• 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.

Element’s edge is shown as a solid line and the original chamfer and top surface is shown as a dotted blue line.
The chamfer representation is no longer present.
In addition to creating virtual faces, edges can be split to form virtual edges to aid in various meshing operations.

- Virtual Split Edge at +: splits at the selection point along the edge.

- Virtual Split Edge: requires a fractional entry indicating the position along the edge where the split will be located (e.g. 0.5 results in the line split in half).
Workshop 8.1

Meshing Evaluation

Introduction to ANSYS Mechanical
Introduction to ANSYS Mechanical

Goals

• In this workshop an arm from a mechanism will be solved using several different meshes for comparison.
• Our goal is to explore how meshing changes can have dramatic effects on the quality of the results obtained.
Assumptions

- In the loading conditions being simulated the arm is experiencing both tensile and bending loads as shown here.
- Our area of interest is the web section that reinforces the interior of the arm.
Open the Project page.

From the Units menu verify:

- Project units are set to “Metric (kg, mm, s, C, mA, mV).
- “Display Values in Project Units” is checked (on).
1. From the Toolbox double click “Static Structural” to create a new system.

2. RMB the geometry cell and “Import Geometry” and browse to “Mesh_Arm.stp”.

3. Double click the “Model” cell to open the Mechanical application.
4. Set the working unit system:
   • “Units > Metric (mm, kg, N, s, mV, mA”).

5. Apply the tensile force on the arm:
   a. Highlight the smaller interior cylindrical face.
   b. RMB > Insert > Force
   c. In the detail window choose the component method and enter 5000N in the Y direction.
6. Apply the bending force to the arm:
   a. Highlight the circular face at the base of the smaller end of the arm.
   b. “RMB > Insert > Force”.
   c. In the detail window choose the component method and enter 1000N in the -Z direction.
7. **Apply the fixed support on the arm:**
   a. Select the larger diameter interior cylindrical face.
   b. RMB > Insert > Fixed Support.
8. Mesh the arm using all default settings:
   a. Highlight the mesh branch.
   b. “RMB > Generate Mesh”.

Inspection of the completed mesh shows a very coarse result. In real applications we would likely refine the mesh before solving.
9. Check the element quality:
   a. Highlight the Mesh branch.
   b. In the Statistics details set “Mesh Metric” to “Element Quality”.

The element quality plot shows that many elements are of a relatively low quality. However, to illustrate some of the practices and tools we’ll solve the model as it is.
10. Request Results:
   a. Highlight the “Solution” branch (A6).
   b. RMB > Insert > Stress > Equivalent Von Mises Stress.
   c. RMB > Insert > Stress > Error.

11. Solve the model:
   a. Click Solve.
12. View Initial Results:
The stress result shows one of the web sections may be an area of concern. In reviewing the error plot however we can see there is a rapid transition from high to low energy in adjacent elements. This is an indication that mesh refinement is recommended.
Re-Meshing

At this point there are numerous mesh controls we could employ to improve the mesh. We’ll focus on the potential problem area indicated in the results using several meshing controls.

13. Change the global mesh settings:
   a. Highlight the mesh branch.
   b. In the details change the “Relevance Center” to “Medium”.

![Image of ANSYS mesh settings](image.png)
14. Add a mesh size control:
   a. Highlight the 5 faces shown below.
   b. RMB > Insert > Sizing.
   c. In the details set the element size to 3mm.
15. Remesh the model:
   a. Highlight the “Mesh” branch.
   b. RMB > Generate Mesh.

The new mesh shows we’ve accomplished refinement around the region of interest.
16. Again reviewing the element quality metric from the mesh statistics detail a definite improvement can be seen.
Results

- Solve the model and review results as before.
- A comparison of stresses from our original mesh shows the maximum value has gone from approximately 38 to 43 MPa.
Results

- Compare error plots for the region of interest.

- It’s clear the refinement has reduced the rapid transition in energy values when compared to the original mesh.
Conclusion

- Notice that there are still areas of high energy transition in the model. Our mesh refinement has addressed our stated goal but not the entire model.
Using the Convergence Tool

- ANSYS provides the option to refine the mesh automatically with a goal of convergence on a specific result.
Using the Convergence Tool

- Maintain the global mesh “medium” relevance center
- Delete the 3mm mesh sizing tool applied to surfaces
- Insert a convergence tool on the Equivalent stress result for the whole part and set the Allowable Change to 10%
- Select the Solution branch in the model tree and refer to the Details section... Change the Max Refinement Loops to 3
- Solve the model and observe the results tracked by the convergence tool
- What happens and why?
Using the Convergence Tool

• Edit the part in DesignModeler and add 5mm fillets to the edges shown
Using the Convergence Tool

- In DesignModeler: File >> Refresh Inputs then Exit
- In Workbench: Refresh the project and then return to Mechanical window – you should see your fillets added
- You may have to reassign your BC’s
- Solve the model again and observe the convergence behavior
- What happens and why?