Lecture 8
Mesh Quality

Introduction to ANSYS Meshing
Mesh Quality

What you will learn from this presentation

• Impact of the Mesh Quality on the Solution
• Quality criteria
• Methods for checking the mesh quality
• Tools to improve quality in Meshing
• Pinch
• Virtual topology
Meshing Process in ANSYS Meshing

1. Specify Global Mesh Settings
2. Insert Local Mesh Settings
3. Preview & Generate Mesh
4. Check Mesh Quality
Impact of the Mesh Quality

Good quality mesh means that...

• Mesh quality criteria are within correct range  
  – Orthogonal quality ...
• Mesh is valid for studied physics  
  – Boundary layer ...
• Solution is grid independent
• Important geometric details are well captured

Bad quality mesh can cause;
• Convergence difficulties
• Bad physic description
• Diffuse solution

User must...
• Check quality criteria and improve grid if needed
• Think about model and solver settings before generating the grid
• Perform mesh parametric study, mesh adaption ...

Table of Design Points

<table>
<thead>
<tr>
<th>1</th>
<th>Name</th>
<th>P1 - Sweep Method 3 Sweep Element Size</th>
<th>P2 - Sweep Method 2 Sweep Element Size</th>
<th>P3 - Sweep Method 1 Sweep Element Size</th>
<th>P4 - Face Sizing Element Size</th>
<th>P5 - Edge Sizing Element Size</th>
<th>P6 - Dp</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>m</td>
<td>m</td>
<td>m</td>
<td>m</td>
<td>m</td>
<td>m</td>
<td>m</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>Current 0.04</td>
<td>0.04</td>
<td>0.04</td>
<td>0.02</td>
<td></td>
<td>747.86</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>DP 1 0.02</td>
<td>0.02</td>
<td>0.02</td>
<td>0.01</td>
<td></td>
<td>500.04</td>
</tr>
<tr>
<td></td>
<td>5</td>
<td>DP 2 0.01</td>
<td>0.01</td>
<td>0.01</td>
<td>0.005</td>
<td></td>
<td>361.4</td>
</tr>
<tr>
<td></td>
<td>6</td>
<td>DP 3 0.005</td>
<td>0.005</td>
<td>0.005</td>
<td>0.0025</td>
<td></td>
<td>307.6</td>
</tr>
<tr>
<td></td>
<td>7</td>
<td>DP 4 0.0025</td>
<td>0.0025</td>
<td>0.0025</td>
<td>0.00125</td>
<td></td>
<td>299.86</td>
</tr>
</tbody>
</table>
Impact of the Mesh Quality on the Solution

- Example showing difference between a mesh with cells failing the quality criteria and a good mesh
- Unphysical values in vicinity of poor quality cells
Impact of the Mesh Quality on the Solution

• Diffusion example

V_{z_{\text{MIN}}} \approx -90\text{ft/min}
V_{z_{\text{MAX}}} \approx 600\text{ft/min}

V_{z_{\text{MIN}}} \approx -100\text{ft/min}
V_{z_{\text{MAX}}} \approx 400\text{ft/min}
Grid Dependency

- Solution run with multiple meshes
- Note: For all runs the computed Y+ is valid for wall function (first cell not in laminar zone)
• Hexa cells can be stretched in stream direction to reduce number of cells
• Bias defined on inlet and outlet walls
• Bias defined on inlet edges
  – 16 000 cells (~DP2)
  – Delta P = 310 Pa (~DP3)
Hexa vs. Tetra

- **Hexa:** Concentration in one direction
  - Angles unchanged
- **Tetra:** Concentration in one direction
  - Angles change
- **Prism:** Concentration in one direction
  - Angles unchanged
- **Solution for boundary layer resolution**
  - Hybrid prism/tetra meshes
  - Prism in near-wall region, tetra in volume
  - Automated
  - Reduced CPU-time for good boundary layer resolution
Mesh Statistics and Mesh Metrics

Displays mesh information for Nodes and Elements

List of quality criteria for the Mesh Metric
• Select the required criteria to get details for quality
• It shows minimum, maximum, average and standard deviation

Different physics and different solvers have different requirements for mesh quality

Mesh metrics available in ANSYS Meshing include:
  – Element Quality
  – Aspect Ratio
  – Jacobean Ration
  – Warping Factor
  – Parallel Deviation
  – Maximum Corner Angle
  – Skewness
  – Orthogonal Quality

For Multi-Body Parts, go to corresponding body in Tree Outline to get its separate mesh statistics per part/body
Mesh Quality Metrics

Orthogonal Quality (OQ)

Derived directly from Fluent solver discretization

- For a cell it is the minimum of:
  \[
  \frac{A_i \cdot f_i}{||A_i||\ ||f_i||} \quad \frac{A_i \cdot c_i}{||A_i||\ ||c_i||}
  \]
  computed for each face \( i \)

For the face it is computed as the minimum of

\[
\frac{A_i \cdot e_i}{||A_i||\ ||e_i||}
\]

computed for each edge \( i \)

Where \( A_i \) is the face normal vector and \( f_i \) is a vector from the centroid of the cell to the centroid of that face, and \( c_i \) is a vector from the centroid of the cell to the centroid of the adjacent cell, where \( e_i \) is the vector from the centroid of the face to the centroid of the edge.

At boundaries and internal walls

\( c_i \) is ignored in the computations of OQ

<table>
<thead>
<tr>
<th>Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Worst</td>
</tr>
<tr>
<td>1</td>
<td>Perfect</td>
</tr>
</tbody>
</table>
Mesh Quality Metrics

Skewness

Two methods for determining skewness:

1. Equilateral Volume deviation:

   Skewness = \( \frac{\text{optimal cell size} - \text{cell size}}{\text{optimal cell size}} \)

   Applies only for triangles and tetrahedrons

2. Normalized Angle deviation:

   Skewness = \( \max \left[ \frac{\theta_{\text{max}} - \theta_e}{180 - \theta_e}, \frac{\theta_e - \theta_{\text{min}}}{\theta_e} \right] \)

   Where \( \theta_e \) is the equiangular face/cell (60 for tets and tris, and 90 for quads and hexas)
   - Applies to all cell and face shapes
   - Used for hexa, prisms and pyramids
Mesh Quality

Mesh quality recommendations

Low Orthogonal Quality or high skewness values are not recommended

Generally try to keep minimum orthogonal quality > 0.1, or maximum skewness < 0.95. However these values may be different depending on the physics and the location of the cell.

Fluent reports negative cell volumes if the mesh contains degenerate cells.

**Skewness** mesh metrics spectrum

<table>
<thead>
<tr>
<th>Skewness</th>
<th>Excellent</th>
<th>Very good</th>
<th>Good</th>
<th>Acceptable</th>
<th>Bad</th>
<th>Unacceptable</th>
</tr>
</thead>
<tbody>
<tr>
<td>0-0.25</td>
<td>0.25-0.50</td>
<td>0.50-0.80</td>
<td>0.80-0.94</td>
<td>0.95-0.97</td>
<td>0.98-1.00</td>
<td></td>
</tr>
</tbody>
</table>

**Orthogonal Quality** mesh metrics spectrum

<table>
<thead>
<tr>
<th>Orthogonal Quality</th>
<th>Unacceptable</th>
<th>Bad</th>
<th>Acceptable</th>
<th>Good</th>
<th>Very good</th>
<th>Excellent</th>
</tr>
</thead>
<tbody>
<tr>
<td>0-0.001</td>
<td>0.001-0.14</td>
<td>0.15-0.20</td>
<td>0.20-0.69</td>
<td>0.70-0.95</td>
<td>0.95-1.00</td>
<td></td>
</tr>
</tbody>
</table>
Aspect Ratio

2-D:
• Length / height ratio: $\delta x/\delta y$

3-D
• Area ratio
• Radius ratio of circumscribed / inscribed circle

Limitation for some iterative solvers
• $A < 10 \ldots 100$
• (CFX: $< 1000$)

Large aspect ratio are accepted where there is no strong transverse gradient (boundary layer ...)

$\delta x$
$\delta y$
Smoothness

Checked in solver
• Volume Change in Fluent
  – Available in Adapt/Volume
  – $3D: \sigma_i = V_i / V_{nb}$

• Expansion Factor in CFX
  – Checked during mesh import
  – Ratio of largest to smallest element volumes surrounding a node

Recommendation:
Good: $1.0 < \sigma < 1.5$
Fair: $1.5 < \sigma < 2.5$
Poor: $\sigma > 5 \ldots 20$
Mesh Metric Graph

- Displays Mesh Metrics graph for the element quality distribution
- Different element types are plotted with different color bars
- Can be accessed through menu bar using Metric Graph button
- Axis range can be adjusted using controls button (details next slide)

- Click on bars to view corresponding elements in the graphics window
  - Use to help locate poor quality elements
Mesh Metric Graph Controls

- Elements on Y-Axis can be plotted with two methods;
  - Number of Elements
  - Percentage of Volume/Area
- Options to change the range on either axis
- Specify which element types to include in graph
  - Tet4 = 4 Node Linear Tetrahedron
  - Hex8 = 8 Node Linear Hexahedron
  - Wed6 = 6 Node Linear Wedge (Prism)
  - Pyr5 = 5 Node Linear Pyramid
  - Quad4 = 4 Node Linear Quadrilateral
  - Tri3 = 3 Node Linear Triangle
  - Te10, Hex20, Wed15, Pyr13, Quad8 & Tri6 non-linear elements
Section Planes

Displays internal elements of the mesh

- Elements on either side of plane can be displayed
- Toggle between cut or whole elements display
- Elements on the plane

Edit Section Plane button can be used to drag section plane to new location

- Clicking on “Edit Section Plane” button will make section plane’s anchor to appear

Multiple section planes are allowed

For large meshes, it is advisable to switch to geometry mode (click on geometry in the Tree Outline), create the section plane and then go back to mesh model
Mesh Quality Check for CFX

• The CFX solver calculates 3 important measures of mesh quality at the start of a run and updates them each time the mesh is deformed
• Mesh Orthogonality
• Aspect Ratio
• Expansion Factor
Mesh Quality Check for Fluent

Grid check tools available

- **Check**: Perform various mesh consistency checks
- **Report Quality**: lists worse values of orthogonal quality and aspect ratio
- TUI command `mesh/check-verbosity` sets the level of details in the report

**Domain Extents:**
- x-coordinate: min (m) \(-1.340580\times 10^{-01}\), max (m) \(8.000000\times 10^{-01}\)
- y-coordinate: min (m) \(-2.407051\times 10^{-01}\), max (m) \(1.350000\times 10^{-01}\)
- z-coordinate: min (m) \(-3.500000\times 10^{-02}\), max (m) \(3.500000\times 10^{-02}\)

**Volume statistics:**
- Minimum volume (m³): \(2.087421\times 10^{-08}\)
- Maximum volume (m³): \(3.18742e-07\)
- Total volume (m³): \(5.925829e-03\)

**Face area statistics:**
- Minimum face area (m²): \(6.187846\times 10^{-05}\)
- Maximum face area (m²): \(1.274684e-04\)

**Mesh Quality:**
- Orthogonal Quality ranges from 0 to 1, where values close to 0 correspond to low quality.
- Minimum Orthogonal Quality = 9.9961e-01
- Maximum Aspect Ratio = 2.03929e+01

Factors Affecting Quality

Geometry problems
• Small edge
• Gaps
• Sharp angle

Meshing parameters
• Sizing Function On / Off
• Min size too large
• Inflation parameters
  – Total height
  – Maximum angle
• Hard sizing

Meshing methods
• Patch conformal or patch independent tetra
• Sweep or Multizone
• Cutcell

Geometry cleanup in Design Modeler
  or
Virtual topology & pinch in Meshing

Mesh setting change
Virtual Topology

When to use?
- To merge together a number of small (connected) faces/edges
- To simplify small features in the model
- To simplify load abstraction for mechanical analysis
- To create edge splits for better control of the surface mesh control

Virtual cells modify topology
- Original CAD model remains unchanged
- New faceted geometry is created with virtual topology

Restrictions
- Limited to “developable” surfaces
- Virtual Faces cannot form a closed region
Creating Virtual Topology

• To access VT menu, click on Model and then on Virtual Topology

• Right click on VT menu to access automated and manual VT tools
Automatic Virtual Topology

Automatically creating *Virtual Faces*

- Left Click *Virtual Topology* in *Model Tree*
- Set *Behaviour* in *Details*
  - Controls aggressiveness of automatic VT algorithm
  - Low: merges only the worst faces (and edges)
  - Medium & High: try to merge more faces
- Select if Face Edges shall be merged
- Right Click *Virtual Topology* and click *Generate Virtual Cells*

Manually creating a Virtual Face

- RMB on Model tree and select Insert Virtual Topology
- Select Virtual Topology from the Tree Outline
- Pick faces or edges, RMB and Insert Virtual Cell

*All VT entities created can be seen in different colors if Virtual Topology is selected in Tree Outline*
Virtual Topology: Example

Without Virtual cells:
Edges are respected while creating surface mesh

With Virtual cells:
Small faces are merged to form a single virtual face and edges of the original set of faces are no longer respected for meshing

<table>
<thead>
<tr>
<th>Statistics</th>
<th>Without Virtual cells</th>
<th>With Virtual cells</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nodes</td>
<td>122036</td>
<td>120644</td>
</tr>
<tr>
<td>Elements</td>
<td>640547</td>
<td>635831</td>
</tr>
<tr>
<td>Mesh Metric</td>
<td>Skewness</td>
<td>Skewness</td>
</tr>
<tr>
<td>Min</td>
<td>5.7893566496463E-05</td>
<td>8.0149887573544E-06</td>
</tr>
<tr>
<td>Max</td>
<td>0.98737225533058</td>
<td>0.86029266741795</td>
</tr>
<tr>
<td>Average</td>
<td>0.245608757966614</td>
<td>0.24688413449344</td>
</tr>
<tr>
<td>Standard Deviation</td>
<td>0.131328150948273</td>
<td>0.127824986922467</td>
</tr>
</tbody>
</table>
Project to underlying Geometry

– Virtual topologies are a faceted representation of the original geometry. By default mesh is projected to the facets

– Improved projection can be obtained by projecting back to the underlying geometry
Virtual Topology : Example

Creating edge split

- Select Virtual Topology from the Tree Outline
- Pick the edge(s)
- RMB and select ‘Virtual Split Edge at +’ or ‘Virtual Split Edge’ to split the edge at the location specified by the selection, or to enter the split ratio in the Details window, respectively

With edge splits:

We can add edge constrains to improve the mesh

Edge splits can be moved interactively. Pick the virtual edge, hold the F4 key and move the red node along the edge with the mouse
Pinch

- Pinch control removes small features at the mesh level
  - Slivers
  - Short Edges
  - Sharp Angles

- The Pinch feature works on vertices and edges only

- The Pinch feature is supported for the following mesh methods:
  - Patch Conforming Tetrahedrons
  - Thin Solid Sweeps
  - Hex Dominant meshing
  - Quad Dominant Surface Meshing
  - Triangles Surface meshing

- Not supported for
  - CutCell
  - Patch Independent
  - Multizone
  - General Sweep
**Pinch Control**

Pinch features can be defined 2 ways

- **Automatically**: pinches created based on global pinch tolerance in Mesh Detail
- **Manually**: pinch created one by one by user with local tolerance

All pinches are listed in Model Tree under Mesh menu with methods and local controls.
Vertex-vertex ‘Pinch controls’ will be created on an edge with length less than the specified tolerance.

Will pinch out the slave geometry into the master geometry.

Mesh without Pinch Control

Mesh with Pinch Control
**Pinch: Edge-Edge**

Edge-Edge ‘Pinch controls’ will be created on any face for which two edges are within the proximity of specified tolerance.

Will pinch out the entire or a portion of the slave geometry into the master.

**Mesh without Pinch Control**

**Mesh with Pinch Control**
Workshops 5 (Applications Choice)

5a Mixing Tank*
*DM Required

5b Automotive Aero

5c Combustion Chamber (2d)

5d Assembly Meshing

5e Manifold*
*DM Required