**Workshop plan**
Week 1 - Quick start in ANSYS
Week 2 – Geometry – DesignModeler (DM)
**Week 3 – Mesh – Meshing**
Week 4 – Solver – FLUENT/CFX
Week 5 - Post Processing
Week 6 – Import/Export, Data Visualisation
Week 7 – Exam

**Exercise plan**
- Week 1 – Steady problems
- Week 2 – Unsteady problems
- **Week 3 – Playing with mesh / Convection**
- Week 4 – Turbulent Flows
- Week 5 – Turbomachinery
- Week 6 – Complex tasks, multiphase flows
- Week 7 – Exam
What will you learn today:

- Process for pre-processing using ANSYS tools
- What is the ANSYS Meshing?
- Meshing Fundamentals
- How to launch ANSYS Meshing?
- ANSYS Meshing interface
- Geometry concepts
- Meshing methods

Solution - Tutorial #3: Meshing/Convection

Preprocessing Workflow

What is ANSYS Meshing

ANSYS Meshing is a component of ANSYS Workbench
- Meshing platform
- Combines and builds on strengths of pre-processing offerings from ANSYS:
  - ICEM CFD, TOSCA (Fluent Meshing), CFX Mesh, Gambit

Able to adapt and create Meshes for different Physics and Solvers
- CFD: Fluent, CFX and POLYFLOW
- Mechanical: Explicit dynamics, Implicit
- Electromagnetic

Integrates directly with other WB systems
Meshing Fundamentals

**Purpose of the Mesh**
- Equations are solved at cell/nodal locations
  - Domain is required to be divided into discrete cells (meshed)

**Mesh Requirements**
- Efficiency & Accuracy
- Refine (smaller cells) for high solution gradients and fine geometric detail.
- Coarse mesh (larger cells) elsewhere.
- Quality
  - Solution accuracy & stability deteriorates as mesh cells deviate from ideal shape

Meshing Process in Ansys Meshing

1. Specify Global Mesh Settings
   - Physics, Sizing, Inflation, Pinch, ...
2. Insert Local Mesh Settings
   - Sizing, Refine, Pinch, Inflation, ...
3. Preview & Generate Mesh
   - Preview surface mesh, Inflation
4. Check Mesh Quality
   - Mesh metrics, Charts

Launching ANSYS Meshing

- ANSYS Meshing is launched within Workbench
  - 2 ways:
    1. From Analysis Systems
       - Fluid Flow (RANS), Fluid Flow (CFX)
    2. From Component Systems
       - Mesh

Double-click Mesh in the System
- Right-click and select Edit
Geometry Configuration – Multiple Parts
- Geometry composed of multiple parts
  - No connection between parts (no face sharing)

- Each part meshed independently
- Results in a conformal interface. Meshes do not match. No node connections.

Geometry Configuration – Multi-body Parts
- Geometry composed of multiple bodies in a part
  - Depend on “Shared Topology method” (in DM)
    - None
    - Automatic

- Faces in contact are imprinted & fused to form a single face shared between the 2 bodies
- Results in a conformal mesh
- Common face acts as interior

Geometry Configuration – Multiple – body Parts
- Geometry composed of multiple bodies in a part
  - Imprints

- Faces are imprinted on each side of the face
- Contact region is automatically ensured
- For similar mesh on these faces, use “Match Control” Results in a conformal mesh
- Non-conformal interface

Contact Region

Grid Interface (Plane)
Which method to choose?

Why Multiple Methods?
- Choice depends on:
  - Physics
  - Geometry
  - Resources
- Mesh could require just one or a combination of methods.

Methods & Algorithms for – Tetrahedral Meshing

2 algorithms available
- Patch Conforming
- Patch Independent
Tetrahedrons Method: Control

- Patch independent - sizing

- Automatic curvature & proximity refinement option

- Defeaturing Control
  - Set Mesh Based Defeaturing On
  - Set Defeaturing Tolerance
  - Assign Named Selections to selectively preserve geometry

Tetrahedrons Method: Algorithm comparison

- Tetrahedrons mesh - smooth growth rate
- Tetrahedrons mesh - rapid growth rate

Important DM Concepts: Body States

- Active & Frozen
  - Bodies can exist in one of two states
    - Active
    - Frozen
  - Active bodies merge automatically with bodies in contact or overlapping
  - Frozen bodies remain independent

- Why use frozen bodies?
  - Meshing: It is often more efficient to mesh a series of smaller topologically simpler bodies than one large complex shape
  - Solver: Different physical models and boundary conditions can be applied to different areas of the model if they are defined as separate bodies
Hex Meshing

3 methods available
- Sweep
- Multizone
- Hex Dominant (not recommended for CFD)

Hexa Mesh - Introduction

- Reduced element count
  - Reduced run time
- Elements aligned in direction of flow
  - Reduced numerical error

Initial Requirements
- Clean geometry
- May require geometric decomposition

Sweep Meshing

- Generates hex/wedge elements
- Meshes source surfaces
  - Sweeps through to the target
- Body must have topologically identical source and target faces
- Side faces must be mappable
  - A sweep path must be identified
- Only one source and one target face is allowed
  - Alternative "thin" sweep algorithms can have multiple source & target faces

To access the sweep
- Insert Method
- Set to Sweep
Automatic Method

Mesh Method & Behavior
- Combination of Tetrahedron Patch Conforming and Sweep Method
- Automatically identifies non-separable bodies and creates sweep mesh
- All non-separable bodies meshed using tetrahedron Patch Conformal method
- Compatible with inflation

To access it:
- Default method
- Insert method → Set to Automatic

3 methods available
- Quadrilateral Dominant
- Triangles
- Multizone Quad/Tri

Advanced size function & local size controls are supported.
**2D Meshing**

**Control**
- Mapped Surface Meshes
  - Local mesh controls
  - Fully mapped surface meshes
  - Specified edge sizing/interpolation

**Inflation**
- Boundary edges are inflated
- Global & local inflation controls are supported

**2D Mesh Solver Guidelines**

**ANSYS Fluent**
- For a 2D analysis in Fluent, generate the mesh in the XZ plane
- For asymmetric applications use 2-0 and make sure that the domain is asymmetric about the X-axis
- In ANSYS Meshing, by default, a thickness is defined for a surface body and is shown when the view is not normal to the XY plane
  - This happens graphically—no thickness will be shown when the mesh is exported into Fluent 2D solver

**ANSYS CFX**
- For 2D analysis in CFX, create a volume mesh (using layers)
  - 1 element thick in the symmetry direction, i.e., 1 layer
  - This block for Planner 2D
  - This Wedge (6:5) for 2D Ax-symmetric

**Meshing Multiple Bodies / Selective Meshing**

**What is?**
- Selectively picking bodies and meshing them incrementally

**Why?**
- Bodies can be meshed individually
- Mesh seeding from meshed bodies influences neighboring bodies (user has control)
- Automated meshing can be used at any time to mesh all remaining bodies
- When controls are added, only affected body meshes require remeshing
- Selective body updating
- Extensive mesh method interoperability
Selective Meshing

Local Meshing
- Clear mesh on individual bodies
- Generate mesh on individual bodies
  - Subsequent bodies will use the attached face mesh
  - The meshing results (all types) will depend on the meshing order
  - Adjust mesh controls - to remesh only affected body
- Select body(s)
  - Right click

Selective Meshing
- Use it to record the order of meshing to automate future cases
- Right-click Mesh in the Outline to access it
- A Worksheet is generated
  - Named Selections are automatically created for each meshed body for reference in the Worksheet

Selective Body Updating
- Remeshing only bodies that have changed
- Access option through Task > Options
  - All geometry updated, all bodies remeshed
  - Assumptions for topological change (add/delete) (allow)
  - Non-Asymmetrically: Assumes no topological change fixture

Example: Geometric change to block
Global Mesh Controls

In this lecture we will learn about:
- Introduction to Global Mesh Controls
- Defaults
- General Sizing Controls & Advanced Size Functions
- Global Inflation
- Assembly Meshing Controls
- Statistics
In this lecture we will learn about:

- Local mesh controls (Mesh sizing, Refinement, Match control, Inflation, etc)
- How to apply local controls?
- Effect of local controls on mesh

Preprocessing Workflow

Meshing Process in ANSYS Meshing
**Local Mesh Controls**

- Control the mesh locally
- Depends on the "Mesh Method" used

**Non-CutCell meshing local controls**

- Cell Size
- Number of Elements
- Element Size

- Elements
- Number of Elements
- Element Size

**CutCell meshing local controls**

- Cell Size
- Number of Elements
- Element Size

- Elements
- Number of Elements
- Element Size

**Sizing**

- Recommended for locally defining the mesh sizes
- You can only scope sizing to one geometry entity type at a time
  - For example: you can apply sizing to a number of edges or a number of faces, but not a mix of edges and faces.

**Types of Sizing options**

- Element size: specifies average element edge length on lines, faces or edges
- Number of divisions: specifies number of elements on edge(s)
- Body of influence: specifies average element size within a body
- Sizing options vary depending on the entity type chosen

**Questions?**
Preprocessing Workflow

Meshing Process in ANSYS Meshing

Impact of the Mesh Quality

- Good quality mesh means that...
  - Mesh quality criteria are within correct range
  - Mesh is valid for studied physics
  - Solution is grid independent
  - Important geometric details are well captured
- Bad quality mesh can cause:
  - Convergence difficulties
  - Bad physics description
  - Diffuse solution
- User must...
  - Check quality criteria and improve grid if necessary
  - Think about model and solver settings before
  - Perform mesh parametric study, mesh adaption,
Impact of the Mesh Quality on the Solution

Mesh Statistics and Mesh Metrics

Mesh Quality Metrics

Orthogonal Quality (OQ)

For the face it is computed as the minimum of
\[ \sum_{i=1}^{n} \frac{|A_i|}{|P_i|} \]
computed for each edge.

At boundaries and internal walls of is ignored in the computations of OQ.
Mesh Quality Metrics

Skewness
Two methods for determining skewness:
1. Equilateral Volume deviation:
   Skewness = \frac{\text{actual cell size} - \text{optimal cell size}}{\text{optimal cell size}}
   Applies only for triangles and tetrahedrons
   \[ \text{Skewness} = \frac{a - b}{b} \]
2. Normalized Angle deviation:
   Skewness = \frac{1 - \cos(\theta)}{2}
   Applies to all cell and face shapes
   - Used for hex, prisms and pyramids
   \[ \text{Skewness} = \frac{1 - \cos(\theta)}{2} \]

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
PLANT reports negative cell volumes if the mesh contains degenerate cells

Skewness mesh metrics spectrum
<table>
<thead>
<tr>
<th>Excellent</th>
<th>Very good</th>
<th>Good</th>
<th>Acceptable</th>
<th>Bad</th>
<th>Unacceptable</th>
</tr>
</thead>
<tbody>
<tr>
<td>0-0.15</td>
<td>0.15-0.50</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>
</tr>
</tbody>
</table>

Orthogonal Quality mesh metrics spectrum
<table>
<thead>
<tr>
<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>
</tr>
</tbody>
</table>

Aspect Ratio

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

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

Limitation for some iterative solvers
- A \times 10^{-12} \leq \delta x
- (CFlx < 1000)

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

- Volume Change in Fluent
  - Available in adapts/volume
  - 3D: \( \Delta V/V_0 \)

- 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: \( \sigma > 2.5 \)

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 sections in the Tree view) before the meshing, then create the section planes and then go back to mesh modes.

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

- Use 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:
- Tet: 4 Node Linear Tetrahedron
- Hex: 8 Node Linear Hexahedron
- Wed: 6 Node Linear Wedge (Prism)
- Pyr: 5 Node Linear Pyramid
- Quad: 4 Node Linear Quadrilateral
- Tri: 3 Node Linear Triangle
- Tet, Hex, Wed, Pyr, Quad, Tri, non-linear elements

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

<table>
<thead>
<tr>
<th>Metric Name</th>
<th>All Cells</th>
<th>Mean Orthogonality Angle (Degrees)</th>
<th>Median Orthogonality Angle (Degrees)</th>
<th>Maximum Orthogonality Angle (Degrees)</th>
<th>Minimum Orthogonality Angle (Degrees)</th>
<th>Mean Orthogonality Ratio</th>
<th>Median Orthogonality Ratio</th>
<th>Maximum Orthogonality Ratio</th>
<th>Minimum Orthogonality Ratio</th>
<th>Mean Orthogonality Measure</th>
<th>Median Orthogonality Measure</th>
<th>Maximum Orthogonality Measure</th>
<th>Minimum Orthogonality Measure</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Grid check tools available:
- Check: Perform various mesh checks
  - Report Quality: lists worst quality and aspect ratio

TIB command: meshcheck
Questions?

Assembly Meshing

Meshing Process in ANSYS Meshing

- Specify Global Mesh Controls (Activate Assembly Meshing)
- Insert Local Mesh Controls
- Generate Assembly Mesh
- Check Mesh Quality

Assembly Meshing

- Mesher engine can be used as a single process.
- Mesh methods covered so far are part or body based methods.
- Not compatible with part-body methods.
- Two algorithms available:
  - CutCell & Tetrahedrons

Access

- Assembly Meshing is accessible only when CFD and Fluent are enabled.
- When activated, replace Home by CutCell or Tetrahedrons.

Note: Some global and local controls are only available for Assembly Meshing (e.g., Match Control).