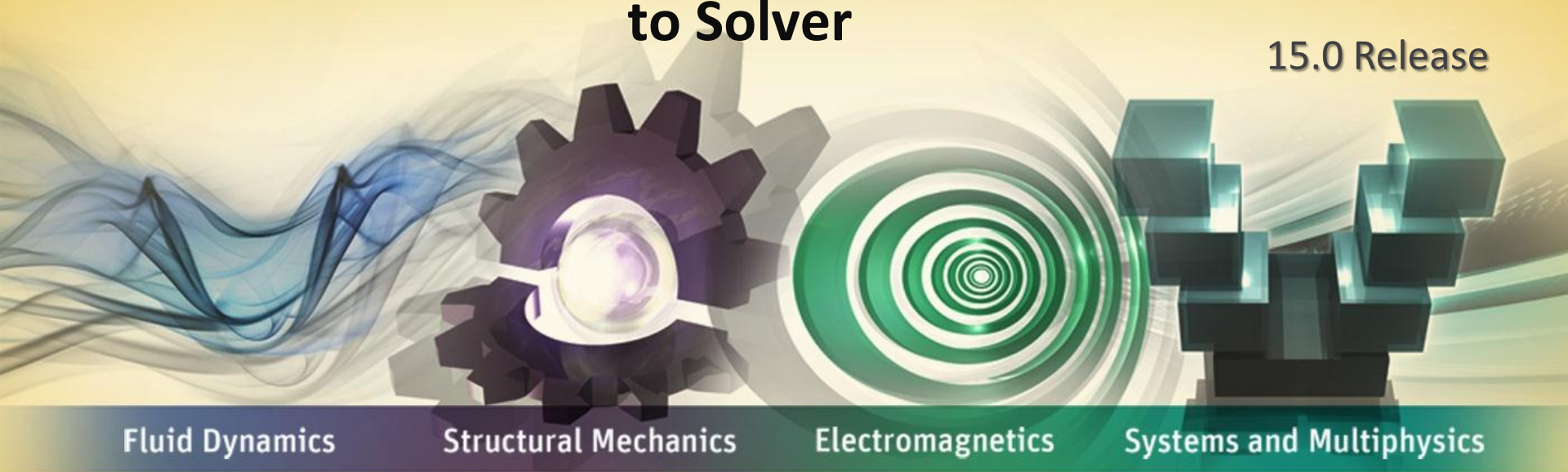


Lecture 6

Mesh Preparation Before Output to Solver

15.0 Release



The banner features four distinct visual elements representing different simulation domains. From left to right: 1. Fluid Dynamics: Represented by flowing blue and white streamlines. 2. Structural Mechanics: Represented by a dark purple gear with a glowing white center. 3. Electromagnetics: Represented by concentric green and white circles. 4. Systems and Multiphysics: Represented by a stack of teal and black 3D cubes. Below these elements is a dark teal bar containing the text labels for each domain.

Fluid Dynamics

Structural Mechanics

Electromagnetics

Systems and Multiphysics

Introduction to ANSYS ICEM CFD

Mesh Preparation Before Output to Solver

Edit Mesh



- Many tools in the **Edit Mesh** tab
 - Smooth, check errors, coarsen, refine, individual element editing
- Two important things should always be done before outputting the mesh to the solver
 1. Make sure the mesh is error-free by going through the **Error** and **Possible Problem** checks
 - Fix any problems
 2. Smooth the mesh to an acceptable quality
 - Fix any problems which cannot smooth to the minimum quality
- These two steps are not necessary with a structured hexa mesh because they are usually handled with the tools in the **Blocking** tab
- When fixing errors or low quality problems, there are always two basic choices:
 1. If there are few problems, edit the elements directly to fix
 2. If there are many problems, inspect the source of the problem, such as a mesh size or geometry problem, then fix this and remesh
- Either choice involves the use of mesh subsets to visualize the problem elements

Mesh Checks

Edit Mesh



Check Mesh

Error	Possible Problems
☒ Duplicate element	☒ Multiple edges
☒ Uncovered faces	☒ Triangle boxes
☒ Missing internal faces	☐ 2 single edges
☐ Periodic problems	☐ Single-multiple edges
☒ Volume orientations	☐ Stand-alone surface mesh
☒ Surface orientations	☒ Single edges
☒ Hanging elements	☐ Delaunay violation
☐ Penetrating elements	☐ Overlapping elements
☐ Disconnected bar elements	☒ Non-manifold vertices
	☒ Unconnected vertices

Set defaults

Elements to check
☒ All ☐ Active parts

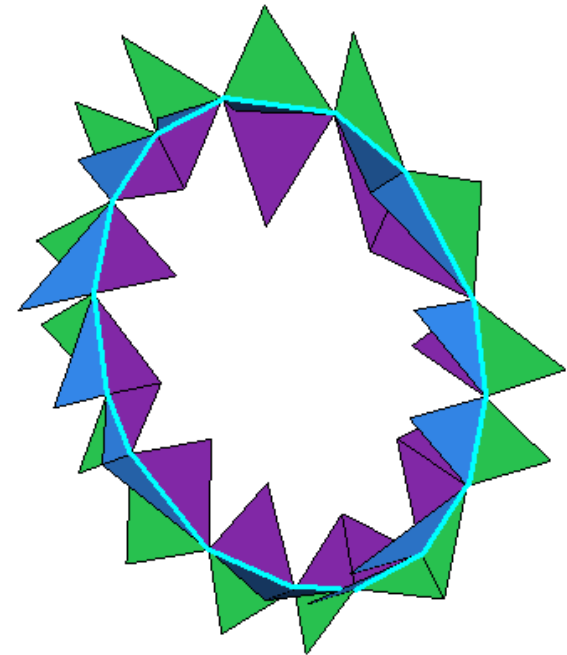
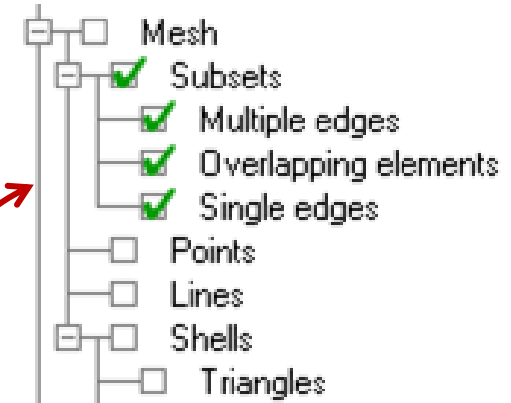
Check mode
☐ Create subsets
☒ Check/fix each

Apply OK Dismiss

- To diagnose mesh connectivity problems
 - **Errors** – most likely to cause problems in:
 - Translation to solver
 - Solver mesh input
 - Solution convergence/run
 - **Possible Problems** – “Unclean surface mesh”
 - Strange configurations of elements, sharp peaks
 - Unwanted holes/gaps
 - May result in incorrect solution
 - Definitions of errors and possible problems found in appendix and help
 - Can check any combination of errors/possible problems
 - Individually select
 - Clicking on **Error** or **Possible Problems** headings will select all options in column – selecting again will de-select all
 - **Set Defaults** will select the most common checks for the current mesh type (2D or 3D)
 - **Check Mode**
 - **Create Subsets** – creates a subset of elements for each problem found (will run through all selected checks)
 - **Check/Fix Each** – offers automatic fixing of some problems (needs user decision after each problem found)

Mesh Checks – Check Mode

- If **Create Subsets** was selected
 - Will go through all checked criteria without interruption
 - Elements that have a particular error/problem are put into a subset with the same diagnostic type name
 - Subsets activated in **Model Tree**
 - Turn off all parts or shells to view subsets
- If **Check/Fix Each** was selected
 - Will be prompted with options one criteria at a time
 - **Fix:** Automatically fix the error/problem
 - Recommended only for **Duplicate Elements, Uncovered Faces, Missing Internal Faces, Volume Orientations, Unconnected Vertices**
 - **Create Subset**
 - **Ignore**
 - For example, multiple edges may be legitimate t-junctions; single edges may be legitimate free edges



Valid multiple edges make T-junctions and usually form closed loops

Edit Mesh

Quality Metrics

Mesh types to check

	Yes	No
LINE_2	☐	☒
TETRA_4	☒	☐
TRI_3	☒	☐
PENTA_6	☒	☐
QUAD_4	☒	☐

Elements to check

☒ All ☐ Active parts
☐ Visible subsets ☐ Visible subsets and active parts

Refresh Histogram

Quality type

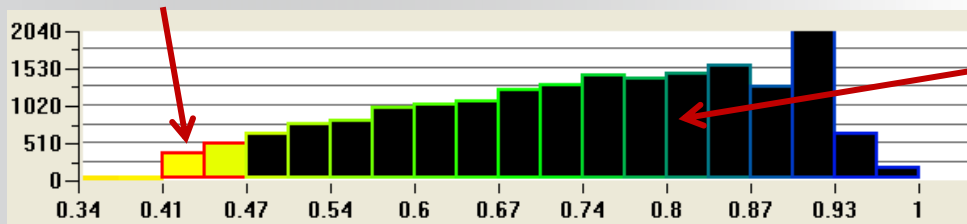
Criterion:

Apply OK Dismiss

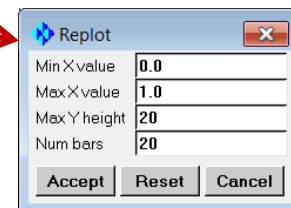
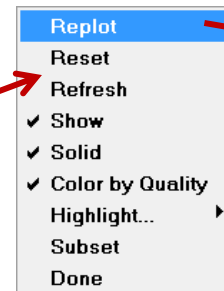


- A diagnostic check of individual element quality
 - **Mesh types to check** – Only checks selected types
 - 1D (Line elements)
 - 2D (Tri and/or Quad)
 - 3D (Tetra, Penta, Hexa and/or Pyramid)
 - **Elements to check** – By part and subset
 - All
 - Active parts
 - Visible subsets
 - Visible subsets and active parts
 - **Refresh Histogram** – Refreshes the histogram displayed without the extra time of recomputing quality
 - **Quality type** – Specifies the Quality criterion for display
 - 49 quality criteria available (cannot smooth by these criterion)
 - Some checks don't apply to all element types

Selecting a histogram bar will display the elements in that range



Right click in histogram for more options



Change X, Y axes

Release 15.0

Mesh Smoothing

Edit Mesh



Smooth Elements Globally

Quality

Smoothing iterations: 5
 Up to value: 0.20
 Criterion: Aspect ratio

Smooth Mesh Type

	Smooth	Freeze	Float
TETRA_4	☒	☐	☐
TRI_3	☒	☐	☐
PENTA_6	☒	☐	☐
QUAD_4	☒	☐	☐

Smooth Parts/Subsets

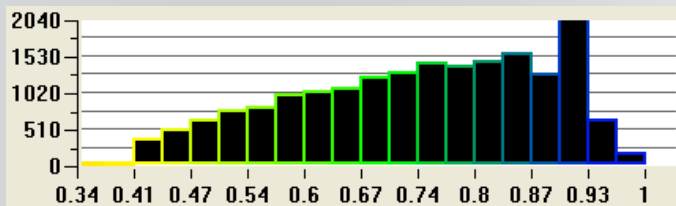
Method: All parts

Refresh Histogram

Apply

OK

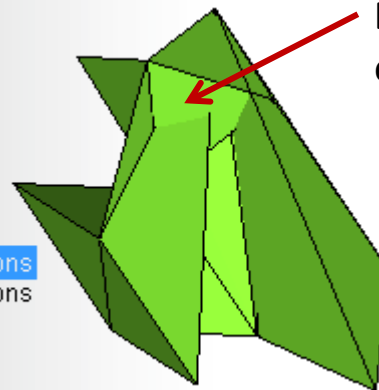
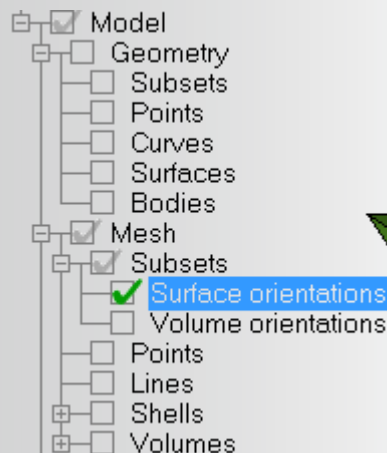
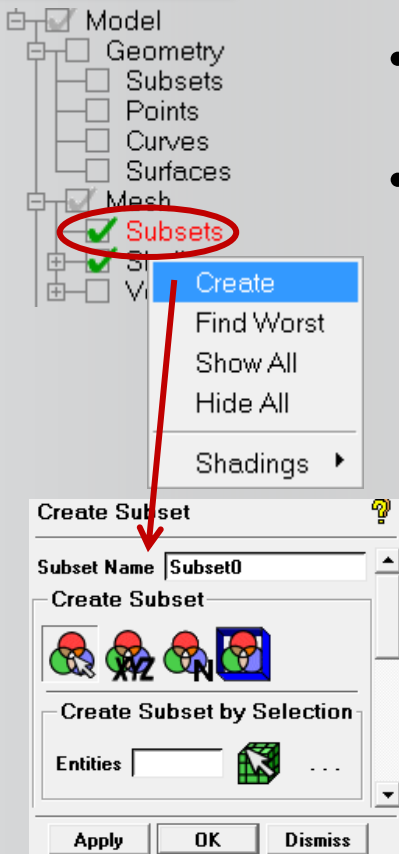
Dismiss



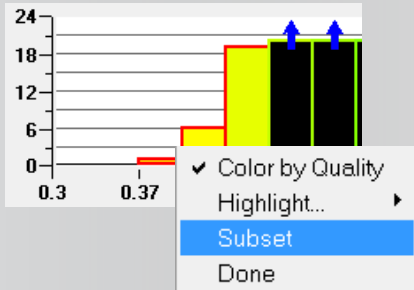
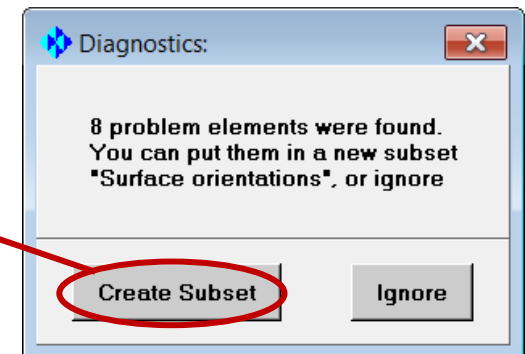
- Automatically improve element quality
 - All element types
 - Necessary to have geometry loaded
 - Nodes are moved to improve the element quality
 - Automatic node movement constrained by node projection type to geometry type – e.g. curve nodes will be constrained to move only on curves
 - Histogram is automatically displayed/updated after smoothing
- User chooses:
 - **Criterion** – 9 to choose from
 - **Up to value**
 - **Smooth Mesh Type**
 - **Smooth**: Element types actively smoothed; quality of type appears as part of histogram.
 - **Freeze**: Nodes are held in place during the smoothing process. These elements not shown in histogram
 - **Float**: Nodes are freed to move along with adjacent smoothed elements, but quality ignored; not shown in histogram
 - Example: After running prism - Freeze Prisms and Pyramids while smoothing Tetra. Float surface elements

Mesh Subsets

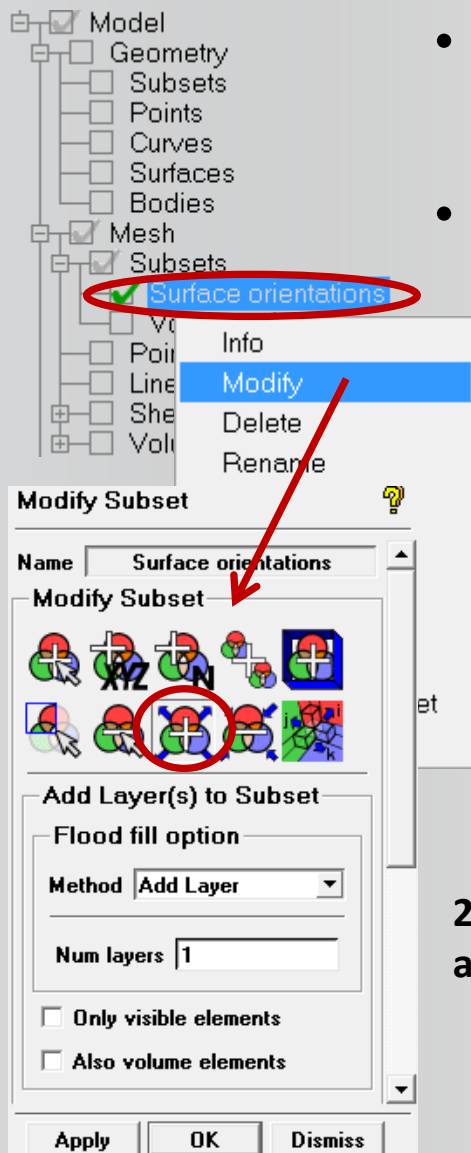
- A mesh subset is a portion of the mesh grouped for viewing purposes
- Very necessary for diagnosing problem elements and editing elements
 - Operations are much quicker than with the entire mesh turned on
 - Can see interior problems not seen with entire mesh displayed
 - 4 ways to created a subset:
 - Right click on **Mesh>Subsets>Create** in Model tree then select elements
 - Highlight elements in smoother or quality histogram, and select **Subset**
 - When checking **errors and possible problems**, choose **Create Subset**
 - In cut plane mode, press **Create mesh subset**



Surface orientation errors found
Element passing through another element



Modifying Subsets



- Once a subset is created, there are many tools to add and subtract elements to the subset for visualizing the problems and to ease working with the mesh edit tools
- Right click on the subset name in the model tree and choose **Modify**
 - Most common to add a couple layers to better visualize the element configurations in the problem location
 - Many other tools to add or subtract elements
 - Add by selection
 - Add near element number or node number
 - Restrict to selected
 - Subtract by selection
 - Remove layers

