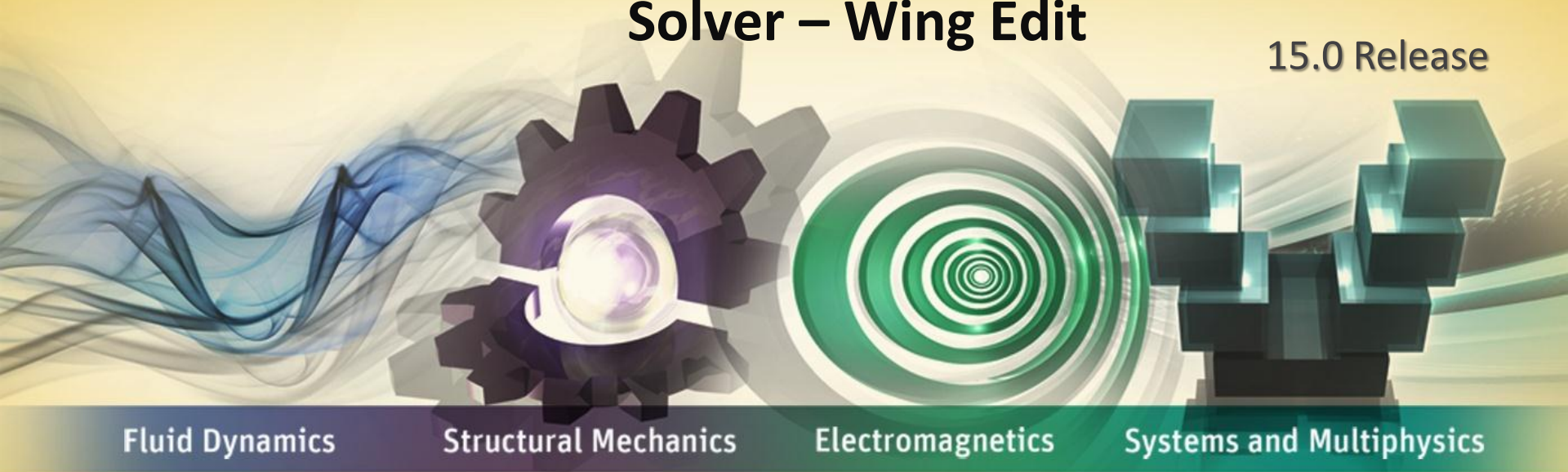


Workshop 7.1 Mesh Preparation and Output to Solver – Wing Edit

15.0 Release

A horizontal banner with a yellow-to-orange gradient background. It features four main sections: 'Fluid Dynamics' with a blue flow visualization, 'Structural Mechanics' with a purple gear, 'Electromagnetics' with a green concentric circle, and 'Systems and Multiphysics' with a 3D block structure. The text '15.0 Release' is in the top right.

Fluid Dynamics

Structural Mechanics

Electromagnetics

Systems and Multiphysics

Introduction to ANSYS ICEM CFD

- Wing Edit model

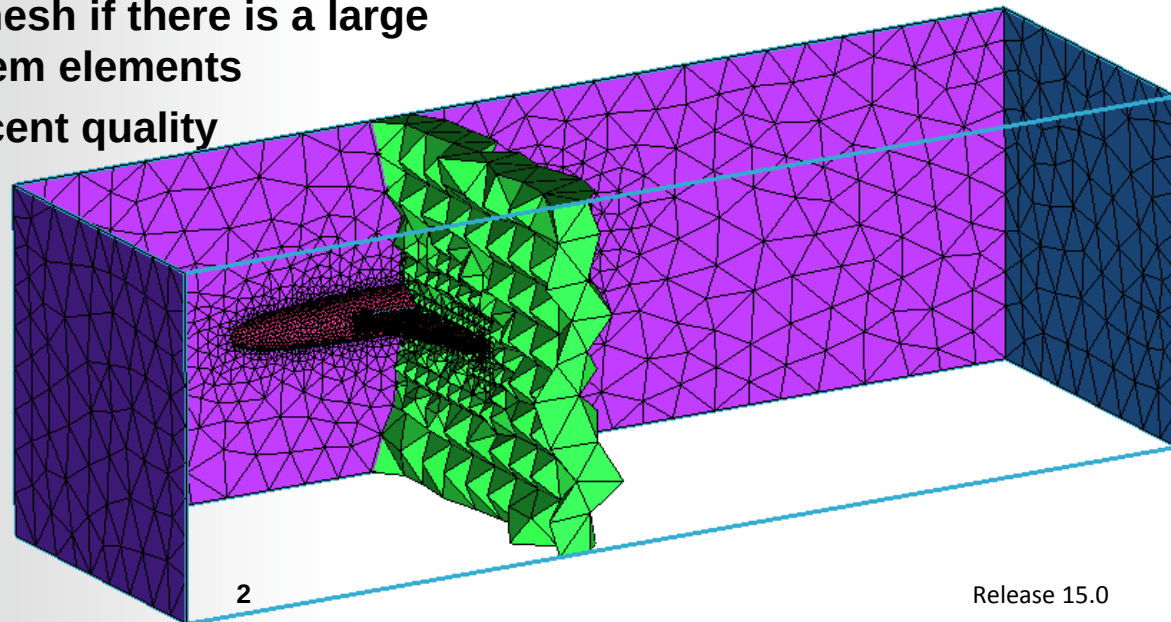
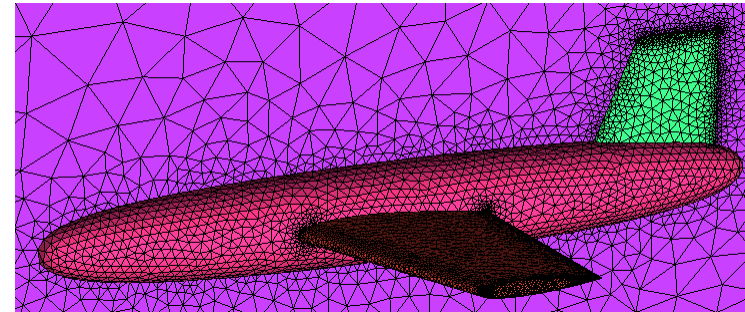
- The model will be meshed with Robust (Octree) method
- Two general steps are necessary before writing to the solver

1. Check mesh for errors and possible problems

- Repair problems at mesh level if a small number of problem elements
- Change mesh size or fix geometry problem and remesh if there is a large number of problem elements

2. Smooth mesh to decent quality

- Write input file for Fluent



Wing Edit Workshop

File Edit View Info Settings Help

New Project...
Open Project...
Save Project...
Save Project As...
Close Project...
Change Working Dir...

Geometry
Mesh
Blocking
Attributes
Parameters
Cartesian

Import Model
Import Geometry
Import Mesh
Export Geometry
Export Mesh

Replay Scripts
Exit

Browse for Folder

New working directory

C:\users\matt\training\ANSYS-ICEM-CFD-Intro_14.0_1st_edi

WingEdit

Clamp
EngineBlock
Femur
HVAC_Hybrid
Stamping
Valve
WaterJacketSection
Wingbody
WingEdit

OK

Cancel

- **File > Change Working Dir...**

- Choose **WingEdit**
- **OK**

- **Use the quick icon to open geometry**

- Choose **WingEdit.tin**
- **Open**



Open

Look in: WingEdit

WingEdit.tin

My Recent Documents
Desktop
My Documents

File name: WingEdit.tin

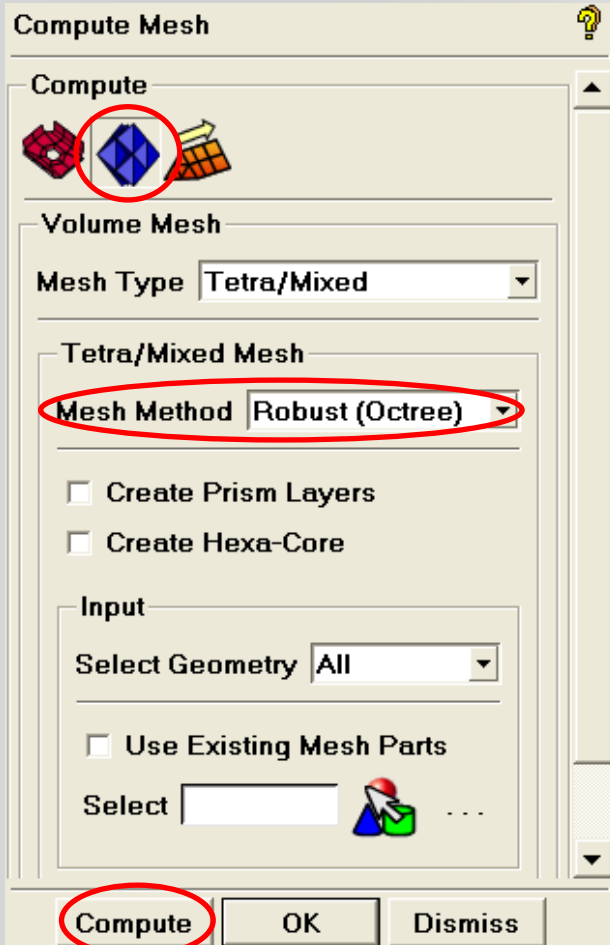
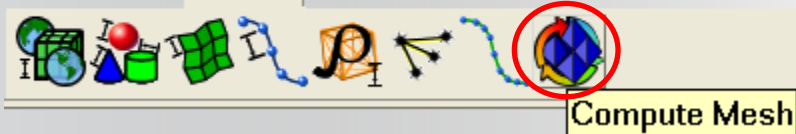
Files of type: Geometry Files (*.tin,*.tin.gz)

Open

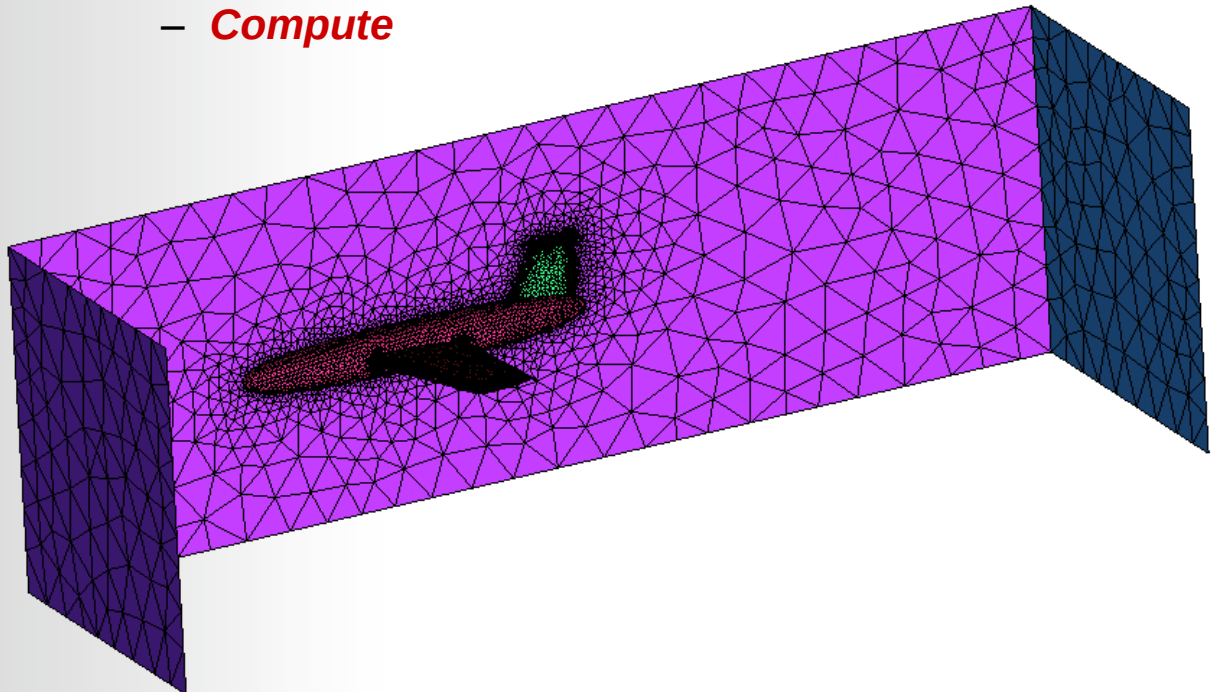
Cancel

Compute Mesh

Mesh

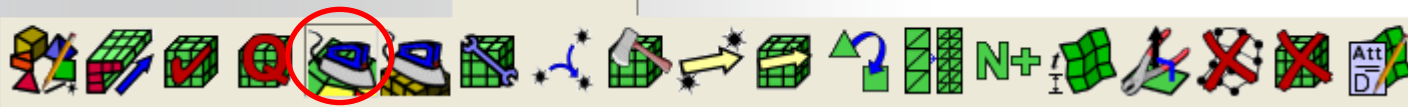


- Global and Part sizes have already been set
- Select **Mesh > Compute Mesh > Volume Mesh**
 - Set **Mesh Method > Robust (Octree)** (default)
 - **Compute**



Mesh Smoothing

Edit Mesh



Smooth Elements Globally

Quality

Smoothing iterations: 5

Up to value: 0.4

Criterion: Quality

Smooth Mesh Type

	Smooth	Freeze	Float
TETRA_4	☒	☐	☐
TRI_3	☒	☐	☐

Smooth Parts/Subsets

Method: All parts

Refresh Histogram

Advanced Options

Laplace

☐ Laplace smoothing

Criterion

☒ Pure ☐ Edge length

Apply OK Dismiss

- Mesh Smoothing

- *Edit Mesh > Smooth Mesh Globally*
- Specify **5 Smoothing iterations** (default)
- **Up to quality = 0.4**
- Keep all other defaults
- **Apply**
- Right-click in quality histogram and select **Replot** to modify its display as shown
 - Set **Min X value = 0**, **Max X value = 1**, **Max Y height = 15**
 - **Accept**

Replot

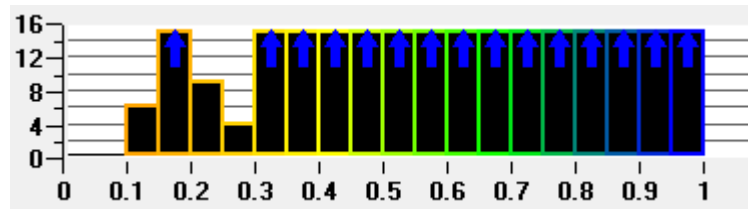
Min X value: 0.0

Max X value: 1.0

Max Y height: 15

Num bars: 20

Accept Reset Cancel

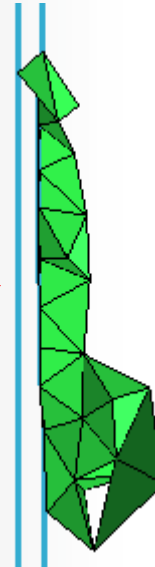
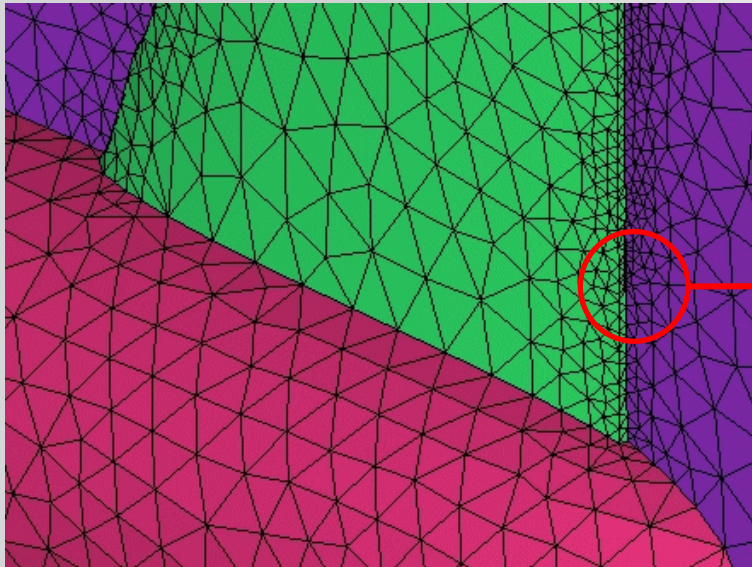
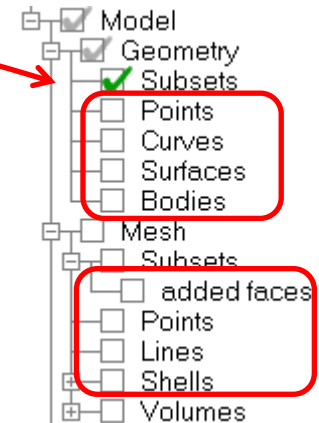
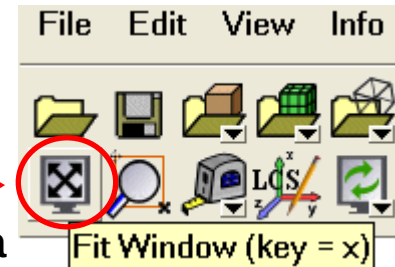
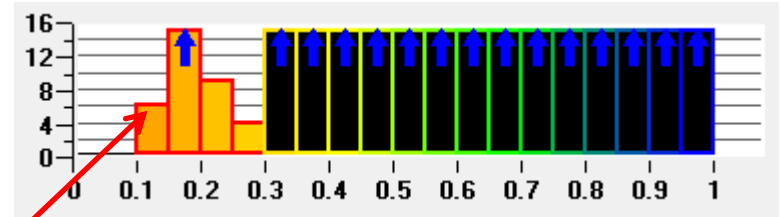


Quality histogram after smoothing and replot

Reviewing the mesh

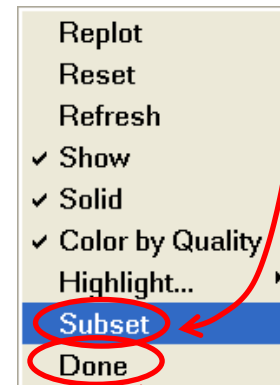
• Quality Histogram

- Left-click on the columns up to **0.3** in the histogram to highlight the elements within those ranges
- Turn off all items under **Geometry** and **Mesh**
- Use the **Fit Window** button to zoom in on the problem
 - Turn on **Curves** again and zoom in or out a bit to get a reference for the location



- Right click in the histogram and choose **Subset** to enter the elements into a subset

- You won't be able to edit them until they are in a subset

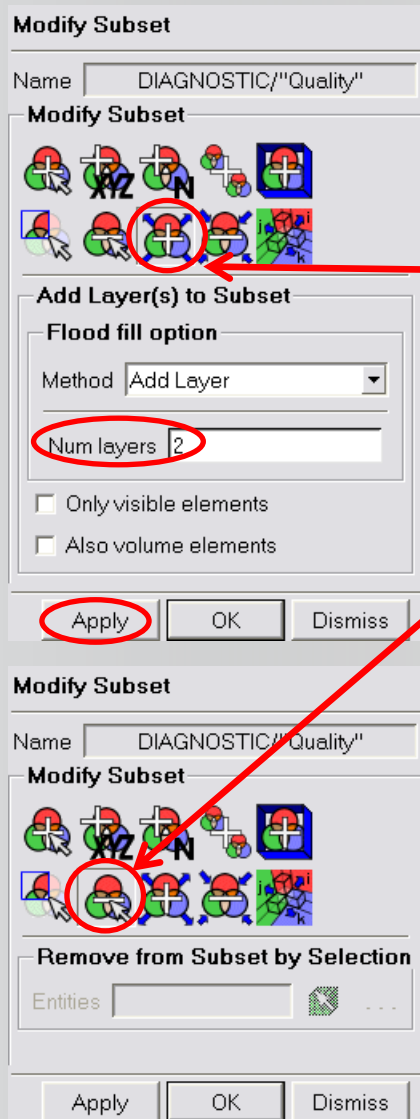
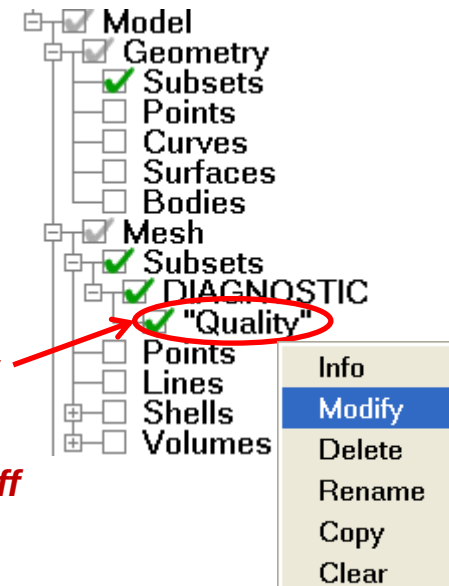


- Right click again and choose **Done** to close histogram

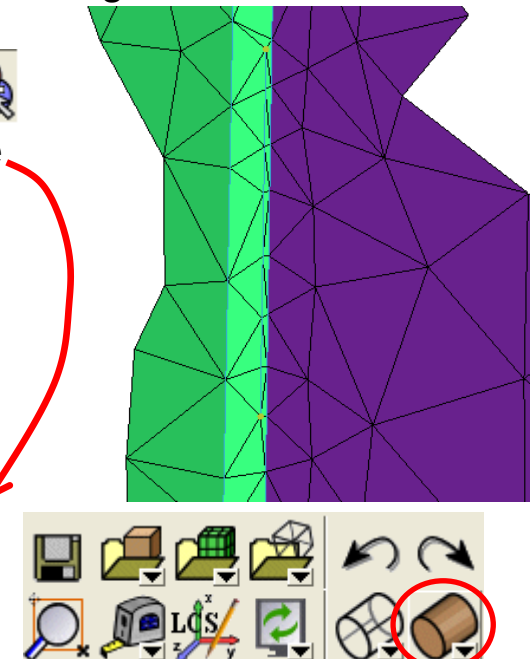
Create/Modify subset

• Using a Subset

- The elements are placed into a subset given the name of the smoothing criteria – “**Quality**” in this case under the **Mesh** branch
- Right-click on the subset name and select **Modify**
- Select **Add Layer(s) to Subset**
- Add **2** layers with **Also volume elements** turned off
 - **Apply**
 - This adds additional elements for ease in seeing surrounding element structure
- Select **Remove from Subset by Selection**
- Click on **Select all volume elements** from the **Select mesh elements** toolbar. Middle click
 - To only visualize and edit surface elements
 - Very difficult to view and edit volume elements and usually not necessary



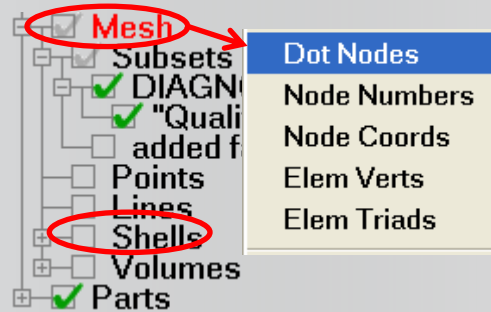
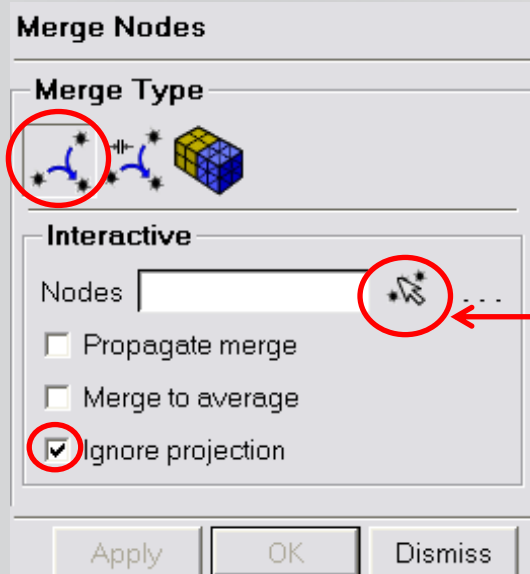
- Display in **Solid** using the utility icon



Merge Nodes

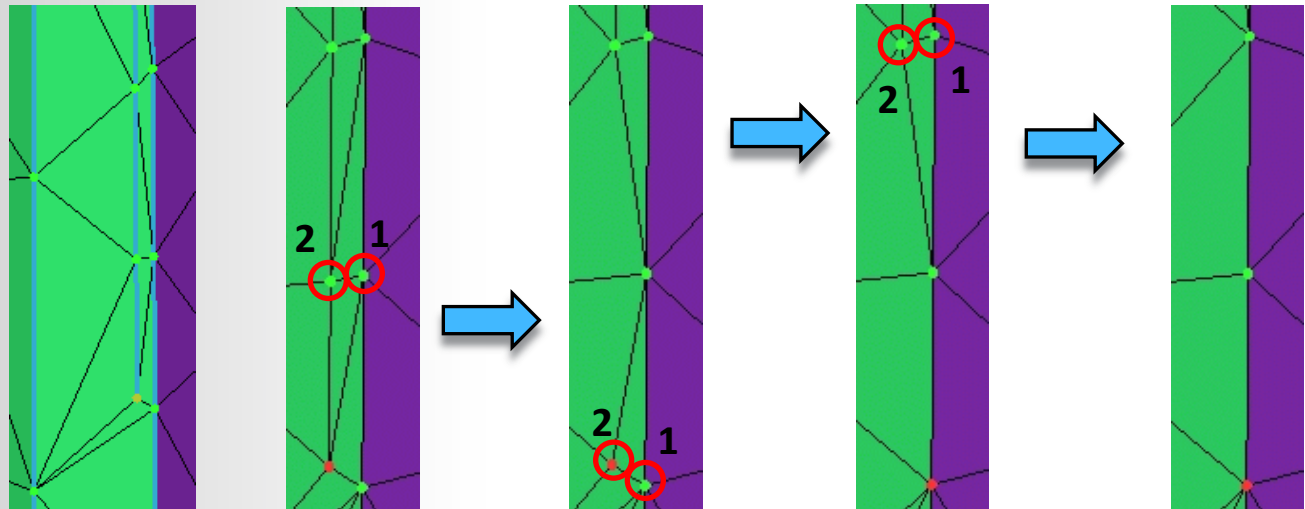


• Merging Nodes



Dot nodes colors nodes according to association

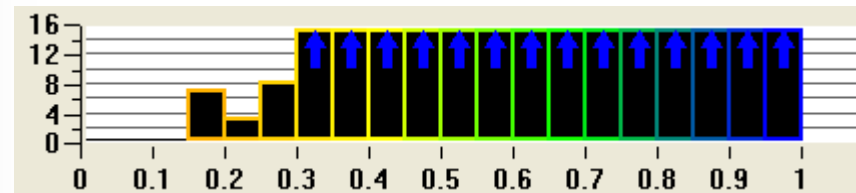
- **Edit Mesh > Merge Nodes > Merge Interactive**
- Activate **Ignore projection** to allow removed node to un-project (when OFF, doesn't allow 2 curve nodes or 2 point nodes to merge)
- With the subset displayed, make sure **Shells** are OFF
- Select  2 nodes from the screen – the first node is kept, the second is removed, and **middle mouse click**
- Right-click on **Mesh** and select **Dot Nodes** to see association
- Continue for length of problem (~8 or 9 locations)



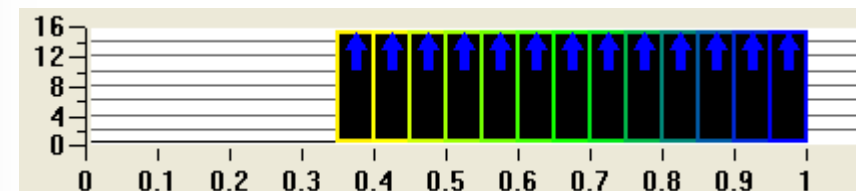
Mesh Smoothing

• Mesh Smoothing

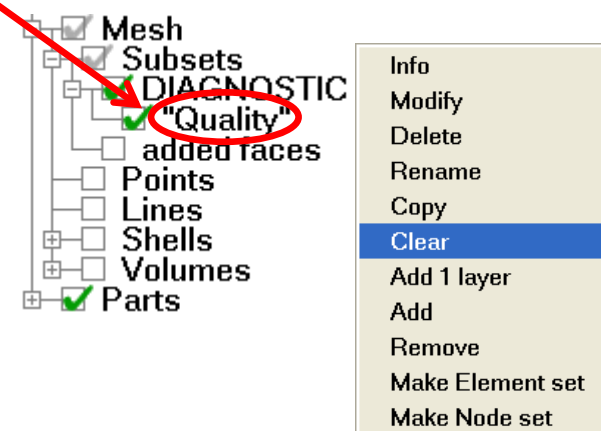
- Always smooth after manual editing
- **Edit Mesh > Smooth Mesh Globally**
- Use previous settings
- **Apply**
- Verify that lowest quality (< 0.3) elements are removed from the left side
- Right-click on the subset name “Quality” from the model tree and select **Clear** or **Delete** (deletes subset name, not elements)
- Manual node movement is also common using **Edit Mesh > Move Nodes > Interactive**
 - Preserves projection to geometry types
 - Remember to smooth again...



After merging but before smoothing



After merging and smoothing

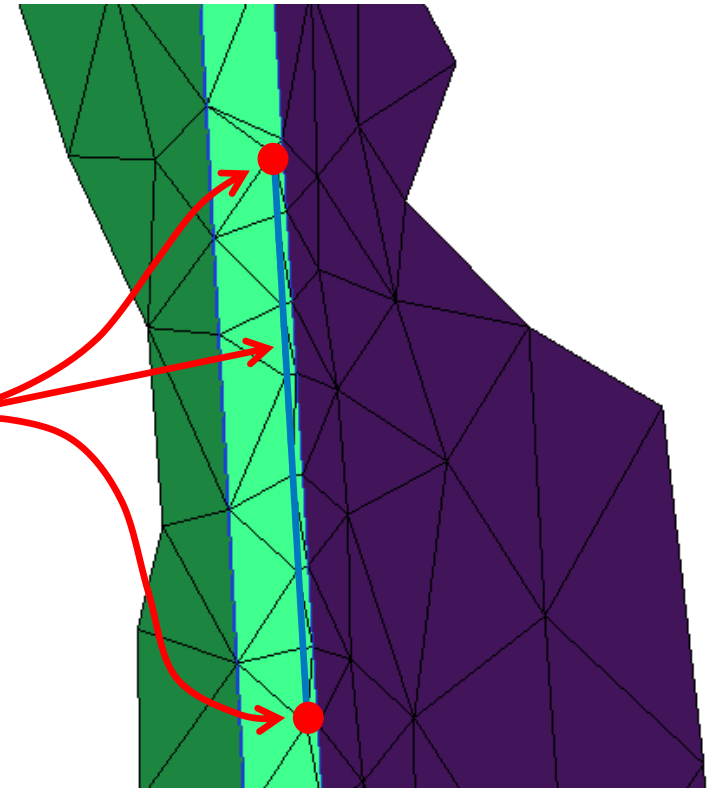


Alternative Method

- Instead of editing the mesh, you could just inspect the source of the problem, fix this, and remesh
 - The source of the problem is the close curve (and end points)
 - These curve and points cause mesh nodes to constrain to them
 - They are not needed since there is no feature to model here (the surface is flat)
 - *Geometry > Delete Curve*
 - *Geometry > Delete Point*



- Delete the one curve and 2 points
- Remesh
- Smooth



Check Mesh

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

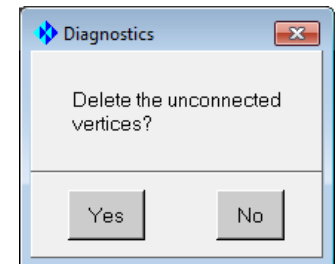
☐ Create subsets

☒ Check/fix each

Apply OK Dismiss

The defaults are different between 2D and 3D mesh

- It is always necessary to check the mesh for errors and possible problems after editing or as a last step before output to solver
 - Select **Edit Mesh > Check Mesh**
 - Use the default set of checks and **Apply**
 - For **Duplicate elements** select **Fix** from the popup
 - This will delete the duplicates
 - These are duplicate line elements that were moved on top of each other from the node merging process
 - For **Delete the unconnected vertices** select **Yes**
 - It's always safe to delete unconnected nodes
 - These are normally deleted during the Save mesh process anyway
 - The smoother often leaves the disconnected nodes

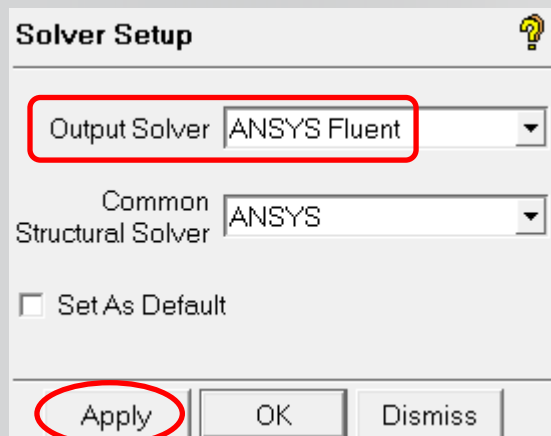


Select Solver



- Select Solver

- Select **Output -> Select Solver**
- Set **Output Solver** to **ANSYS Fluent**



- The **Output Solver** selection can write to any solver ICEM CFD is capable of writing
- The **Common Structural Solver** selection only applies to 5 special structural solvers which allow extended setup in the **Properties**, **Constraints**, **Loads**, and **Solver Options** tabs
 - **Ansys**, **Nastran**, **LS-Dyna**, **Abaqus**, and **Autodyn**

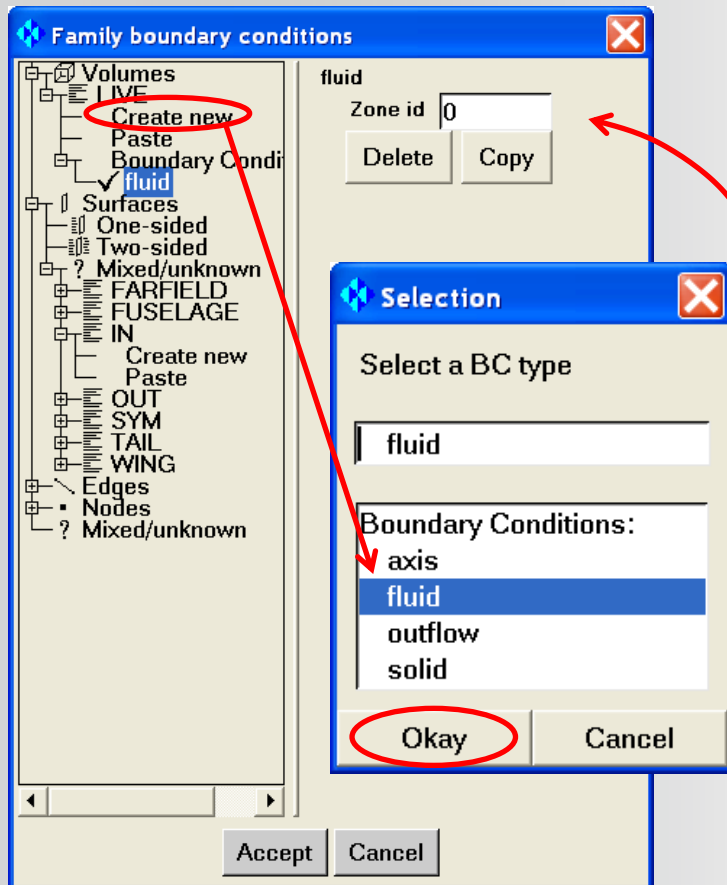
- **Apply**

Define Boundary Conditions



• Define boundary conditions

- Select **Output -> Boundary conditions**
- Expand **Volumes**, then **LIVE**, until you see **Create new** and **Paste**
- Parts are ordered in the tree according to geometry and mesh dimension types (0D, 1D, 2D, 3D) within that part
 - Any part containing mixed dimensions (ex. curves (1D) and tri's (2D)) will be grouped into **mixed/unknown**
- Click on **Create New**
- Select the appropriate boundary condition (**fluid**) from the pop up window
- Click **Okay**
- Different solvers have different choices. If the BC has parameters that can be entered, they will appear at the right side
- It is not required to set any BC's to write to the selected solver



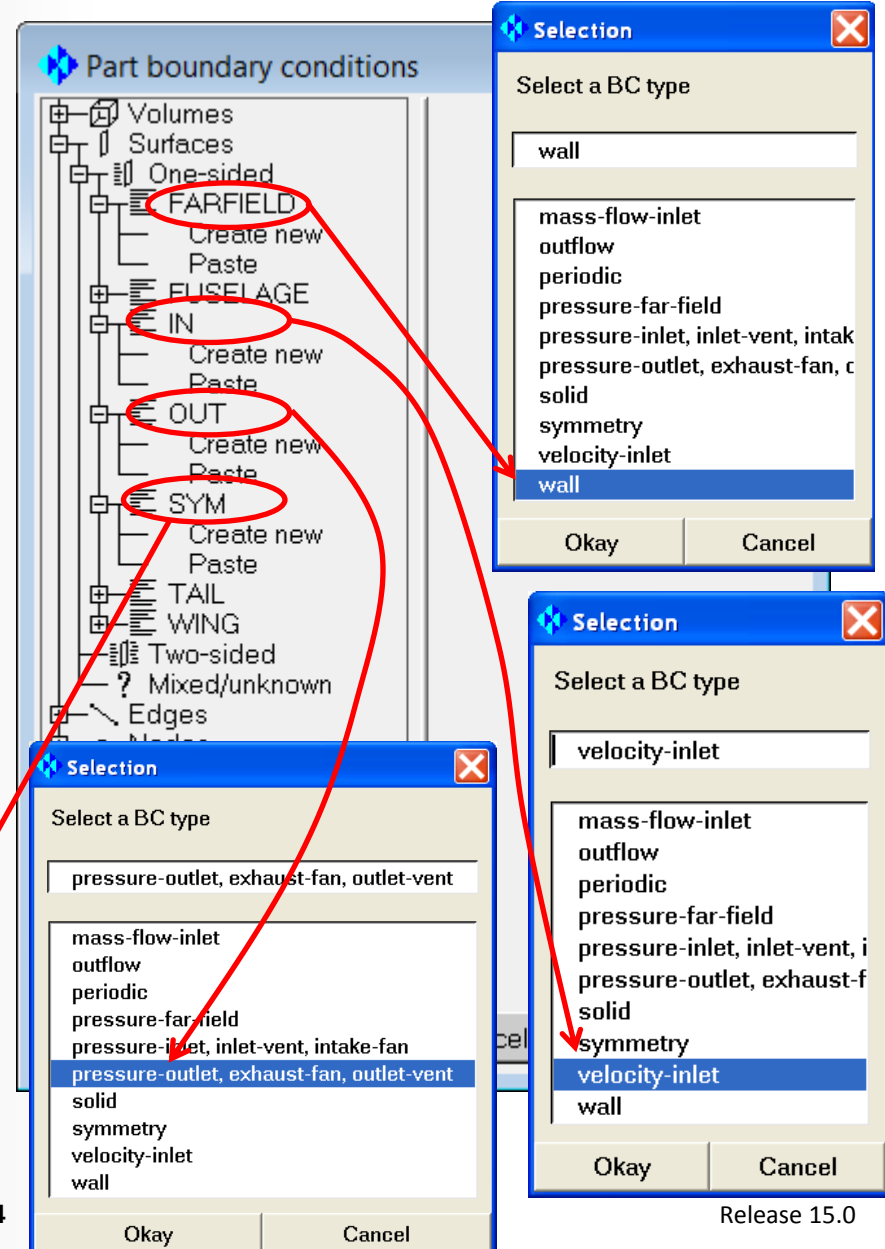
Boundary Conditions - Continued



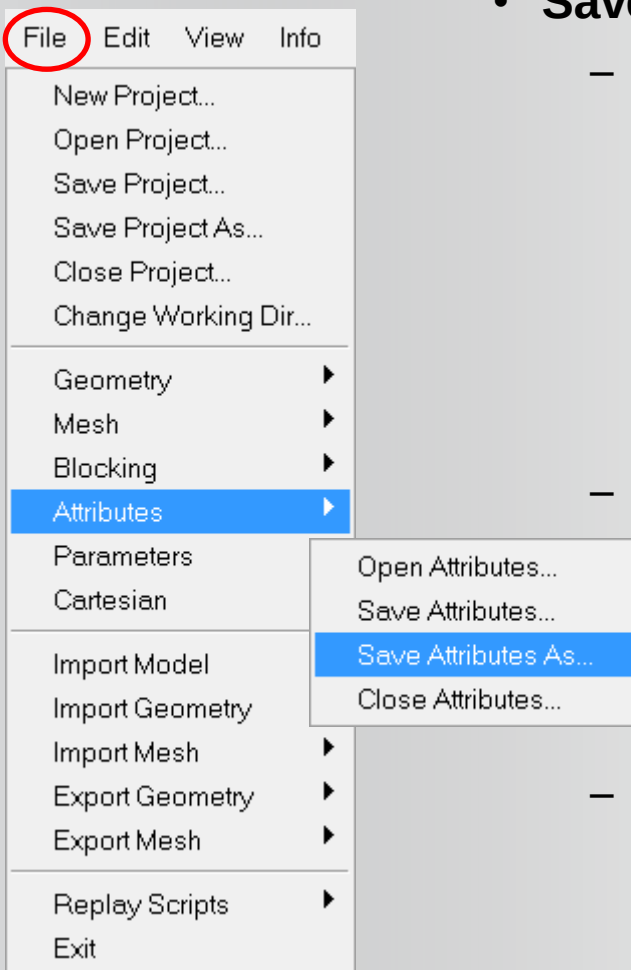
Output

• Continue with assigning surface boundary conditions

- Expand under **FARFIELD** under **Surfaces** list (under **one-sided**)
- Press **Create new**
- Select **wall**, then **Okay**
- Set part **IN** to **velocity-inlet**
- Set **OUT** to **pressure-outlet, exhaust-fan, outlet-vent**
- Set **SYM** to **symmetry**
- Press **Accept** when done

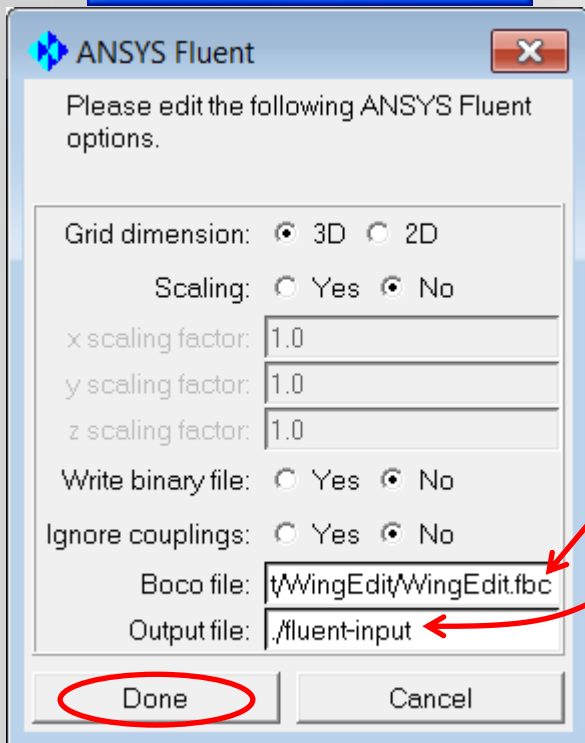
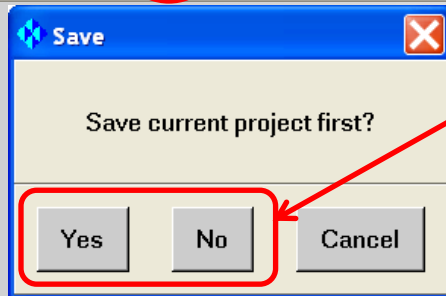
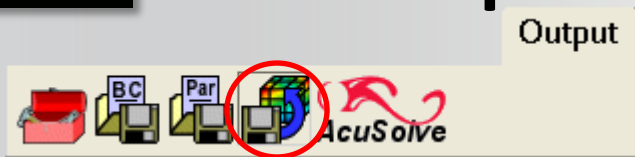


Save Files



- Save boundary condition file and mesh
 - **File > Attributes > Save Attributes As...**
 - Saving attributes saves an attribute file (***.atr**) and boundary condition file (***.fbc**)
 - The attributes file (***.atr**) is for the 5 common structural solvers and the boundary condition file (***.fbc**) is for all others including fluid solvers
 - Save to any name you wish, such as **WingEdit.fbc**
 - **File > Mesh > Save Mesh As...**
 - The mesh has been edited, and the translator to Fluent reads from the file on disk, not from current memory, so you will need to save the mesh
 - Save to any name you wish, such as **WingEdit.uns**
 - When you write the mesh to the solver, it will ask if you want to save the project
 - It is safe to save the project, which saves all files, but the boundary condition file and the mesh are the needed files, so if you have just saved these, it is not necessary to save the project

Write Input



- Write input file

- Select **Output -> Write input**
- When prompted to save the project, it is not necessary if you just saved the boundary conditions and mesh, but you may want to do so to make sure all files have been saved, such as the geometry (***.tin**)
- You will next be prompted to select the mesh (***.uns**) to write out. Select the mesh saved in the previous step (**WingEdit.uns**) or the file named **{project}.uns** if you saved the project.
- The ANSYS Fluent panel pops up
 - This final options window is different for every solver
- Take note of the boundary condition file name
- The output file will be saved in the working directory. The default name of the file is **fluent.msh**. The file name and location can be changed by typing it in the **Output file** field
- Click **Done**
- **Fluent-input.msh** file is created in the working directory

- The mesh can be read into Fluent using **File > Read > Mesh** (or **Case**)
- Take note that **parts** in ICEM CFD become **zones** in Fluent

