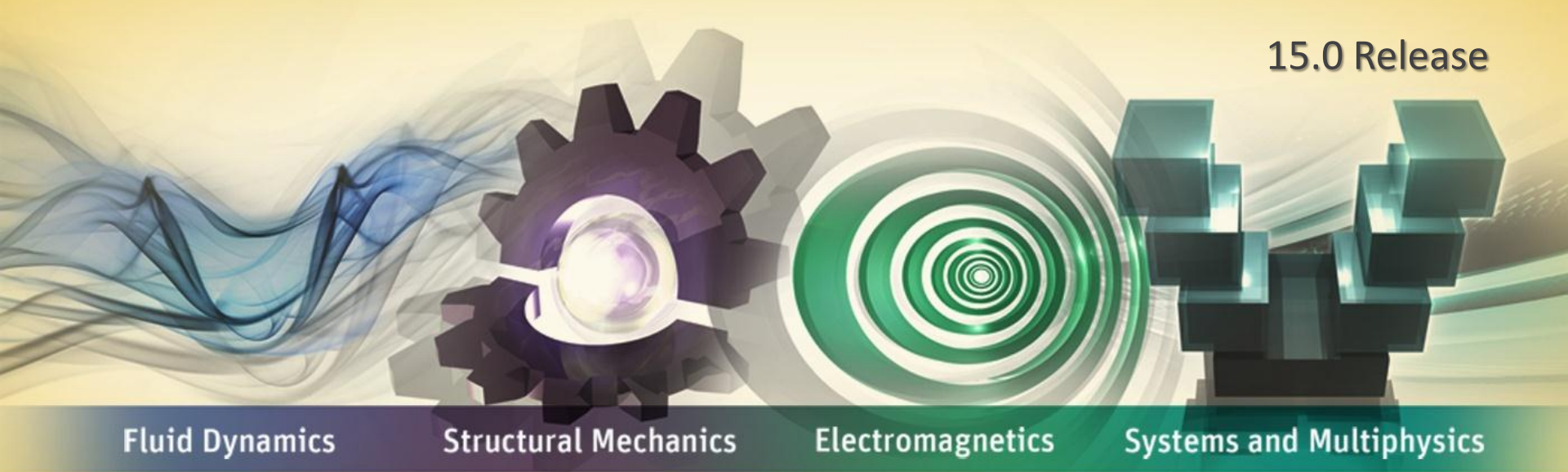


Appendix 4

Additional Volume Meshing Features

15.0 Release

A horizontal banner with a yellow-to-orange gradient background. It features four distinct 3D visualizations: blue fluid flow streamlines, a purple gear with a glowing center, a green concentric circular pattern, and a stack of teal cubes. Below these images is a dark teal bar containing four white text labels.

Fluid Dynamics

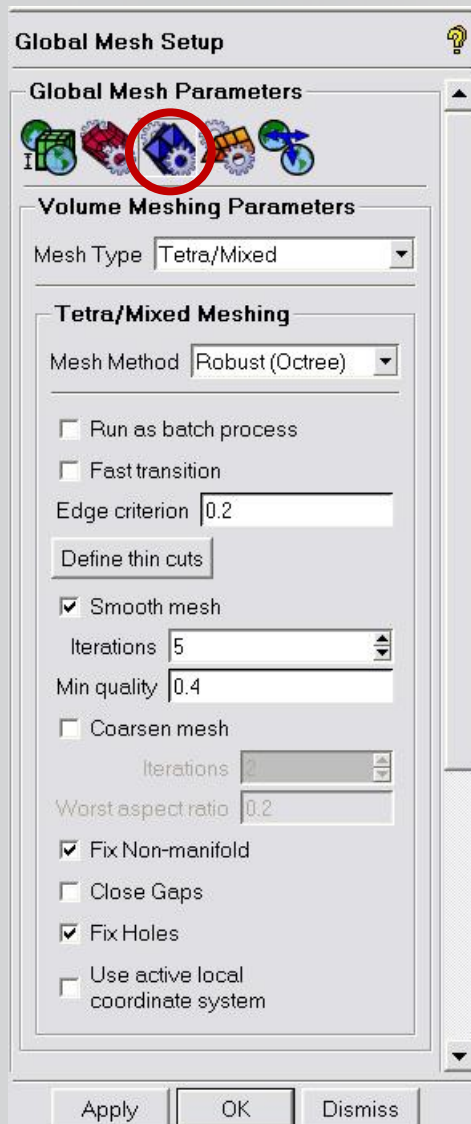
Structural Mechanics

Electromagnetics

Systems and Multiphysics

Introduction to ANSYS ICEM CFD

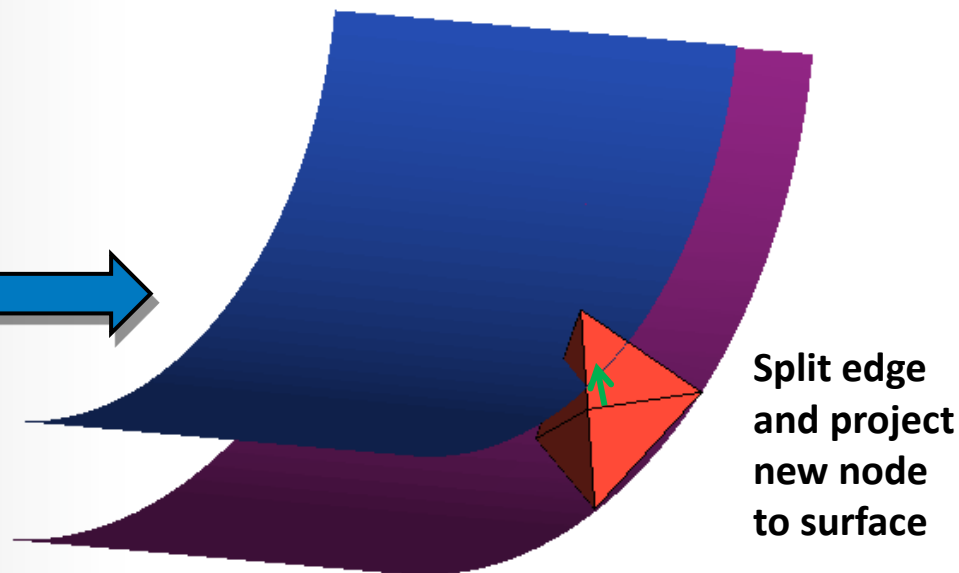
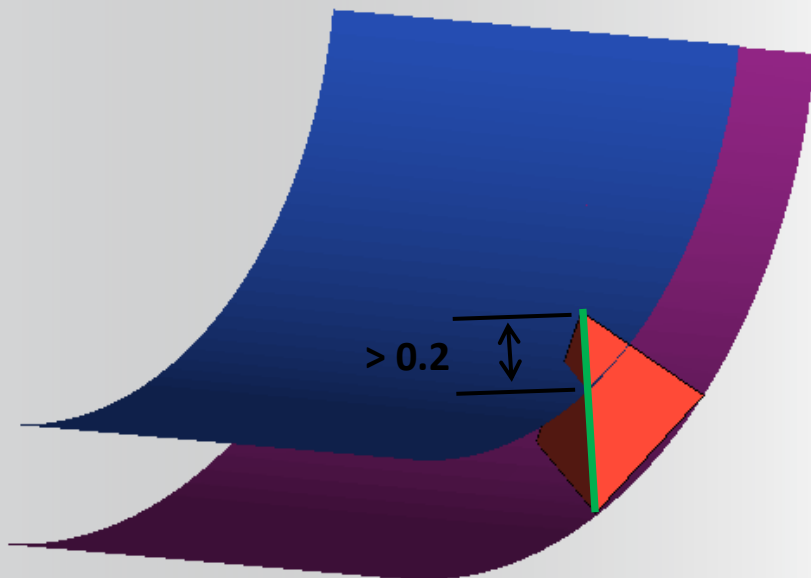
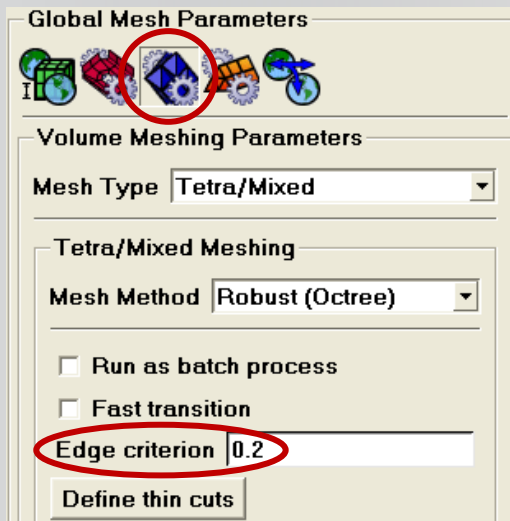
Tetra Octree – Global Options



- **Global Mesh Setup > Volume Meshing parameters**
 - **Run as batch process**
 - Runs as a separate process (tetra.exe). GUI will stay interactive.
 - **Fast Transition**
 - Allows for a slightly faster transition in element size from finer to coarser
 - Reduced element count
 - **Edge Criterion**
 - Split elements at a factor greater than set value to better capture geometry
 - **Define Thin cuts**
 - Tool for resolving thin gaps, sharp angles
 - User selects pairs of opposing parts
 - Resolves elements jumping from one side to another
 - **Smooth**
 - Automatically smoothes after grid generation process
 - **Coarsen**
 - **Fix Non-manifold**
 - Automatically tries to fix elements that jump from surface to another surface
 - For a more detailed description go to **Help > Help Topics > Help Manual > Mesh > Global Mesh Setup > Volume Meshing Parameters > Tetra/Mixed > Robust (Octree)**

- **Edge criterion**

- For **tetra octree** only
- A number 0 – 1
- 0.2 (the default) means if more than 20% of an edge crosses a surface or curve, then split the edge
- Has an effect similar to globally applying a **thin cut**
- Smaller numbers will cause more splitting. The closest node will be projected to the surface, modeling smaller features
- Use prudently. Too small a number can result in random clusters of refined mesh



Tetra Octree Workflow Recommendation

1. Check geometry accuracy and integrity
 - Visually scan surfaces to see if curvature is too choppy or if gaps are large
 - Can use build diagnostic topology with no “filtering” to indicate gaps and holes by the yellow curves
 - Must use a reasonable tolerance, indicating a maximum allowable gap
 - Large gaps also show after running tetra if leakage is reported
2. Move surfaces into intended part names
 - Possibly move curves and points later, but surfaces are most important
3. Obtain necessary curves and points
 - Needed for nodes to align at hard corners
 - Build topology with “filtering” feature angle (30 - 45) or extract curves from surfaces and points from curves, then delete excess
 - Scan model and delete curves/points at more gradual angles where surfaces meet
4. Create material point(s)
 - Defines volumes to keep mesh (all regions initially meshed)
 - Material point must be INSIDE volume, not on surfaces
 - ORFN point is a reserved name which instructs to cut away that region

5. Set mesh sizes

- Globally first
 - Maximum size
 - Curvature/proximity refinement
- Locally
 - Part mesh setup spreadsheet
 - Surface mesh setup
 - Curve Mesh setup
 - Densities

6. Compute mesh

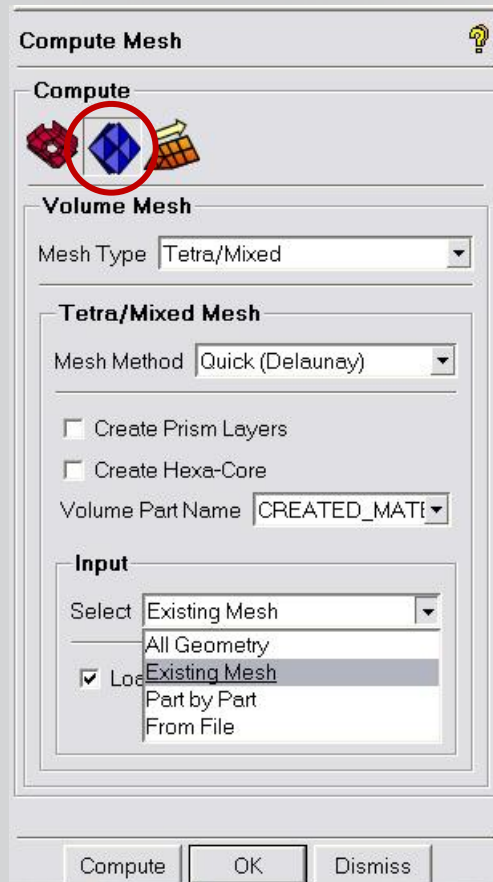
7. Smooth

- Approximately 0.2 minimum quality for tetra

8. Check errors and possible problems

- See ICEM CFD help for descriptions of each check
- Also discussed in *Mesh Preparation* presentation
- Edit mesh directly to fix problems or change a mesh size and remesh

Compute Mesh – Delaunay, Adv. Front, TGrid



• Run Options:

- Similar options as octree except cannot mesh to partial geometry and partial mesh (option: *Use existing mesh parts*)
- *Create Prism Layers* available for both
- *Hexa-Core* not available for *Advancing Front*
- *Volume Part Name*
 - For newly created tetras
 - Can choose *Inherited* to use material points
- *Input*
 - *All Geometry*
 - Will run shell mesh first with no user input/editing
 - Using parameters from *Model/Part/Surface/Curve Mesh Setup*
 - Review *Shell Meshing* chapter
 - If doubtful as to shell mesh quality, run *Shell Mesh* first, then use *Existing Mesh* Input method
 - *Existing Mesh*
 - Most common method. Surfaces already meshed
 - *Part by Part*
 - Meshes each part separately. Nodes are not connected
 - *From File*
 - Saves memory. Surface mesh does not need to be loaded

Densities with No Volume

- Must use **width** parameter
 - Width is the number of tetra layers away from the density which will keep the same size as the density mesh size until allowed to grow
 - Width displays for point and line densities but not plane densities
 - Only tetra octree respects **width** control

Mesh



Create Density

Name density.0

Size .1

Ratio 0

Width 4

Density Location

From

☒ Points ☐ Entity bounds

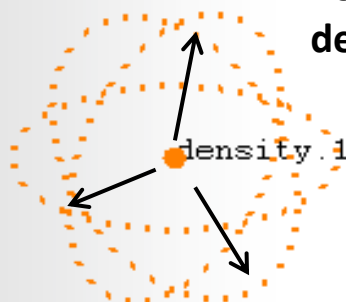
Points ...

Apply

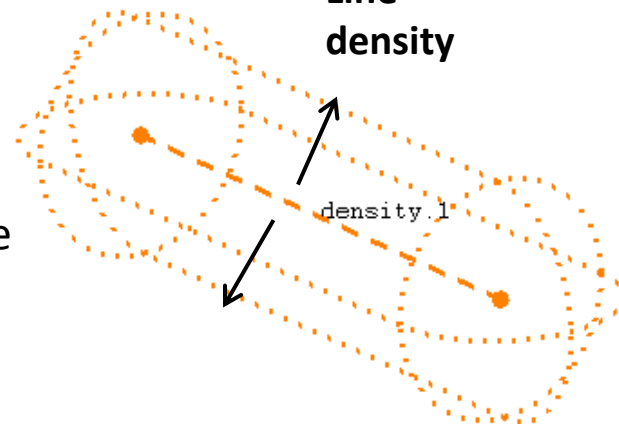
OK

Dismiss

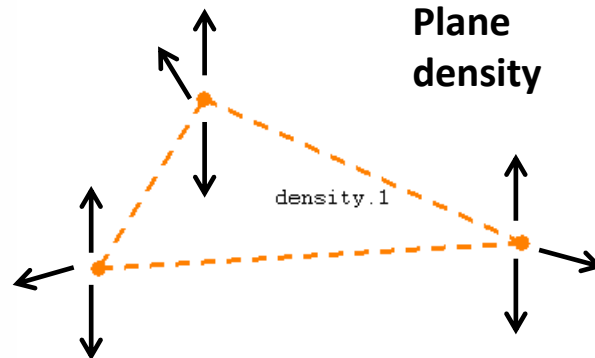
Point
density



Line
density

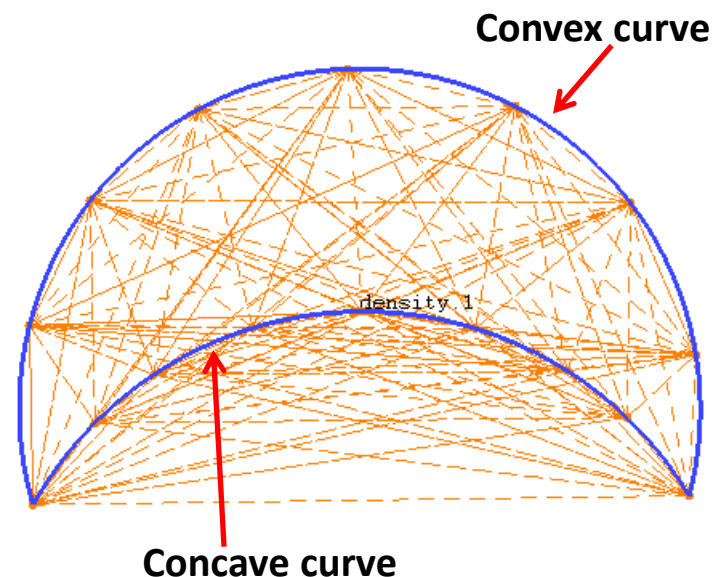
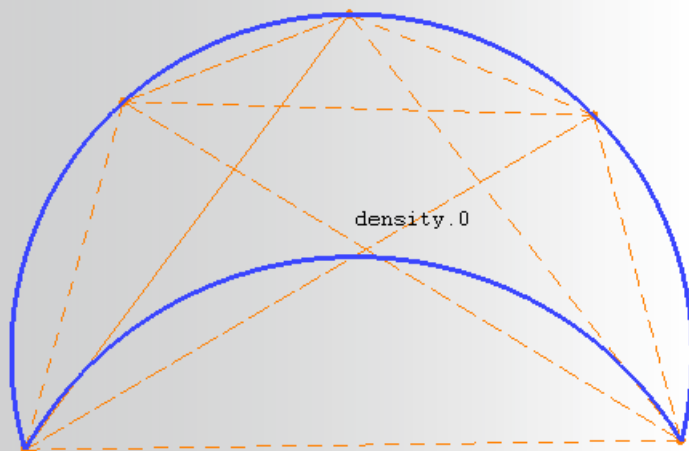


Plane
density



Complicated Density Shapes

- You can select any number of points to model curvature better but only convex shapes are handled



- Need to create multiple densities to model concave shapes without excess elements generated

