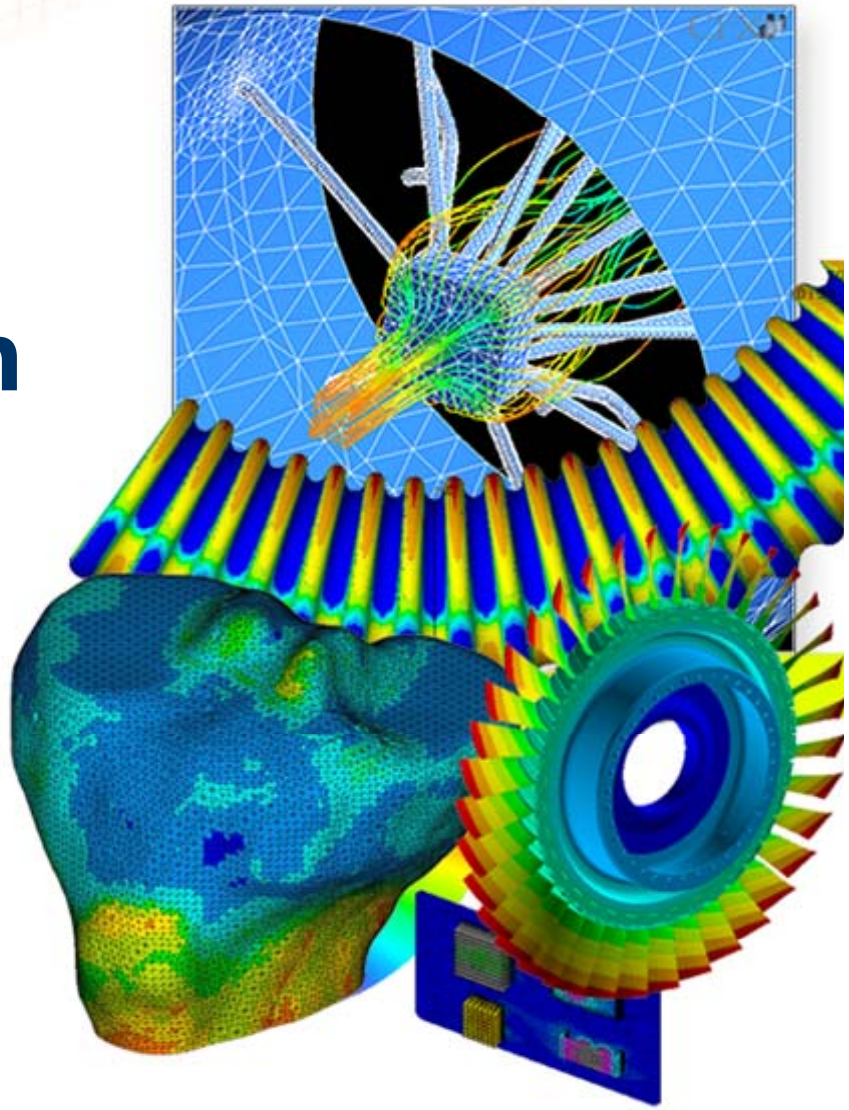
CAE ASSOCIATES

ENGINEERING ADVANTAGE

# CFD Meshing with ANSYS Workbench

*March 14, 2013*



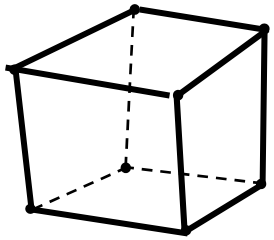
# Agenda

- ANSYS Workbench Mesh Methods
- Efficient Meshing for CFD
- Live Demonstration

# ANSYS Workbench Mesh Methods

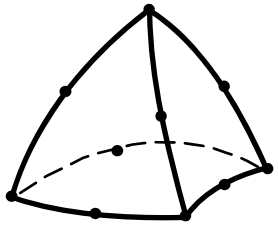
- Mesh methods can be divided into two categories:

- Hexahedral Methods:
  - Sweep
  - Multizone
  - Hex Dominant
  - CutCell (Fluent Only)



Hex Cell

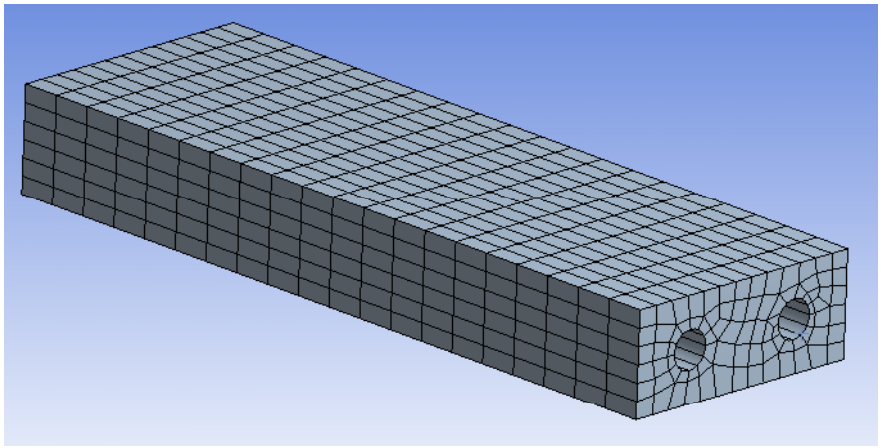
- Tetrahedron Methods:
  - Patch Conforming
  - Patch Independent



Tet Cell

# Mesh Methods

- Sweep Mesh Method
  - Sweep requires topologically consistent source and target faces (same number of vertices per face with a smooth path from the source to the target).
  - Sweep is the default Workbench meshing approach, if a body cannot be swept it is free meshed with tetrahedral elements.
  - Specify manual source (and target faces) when:
    - There exists more than one possible sweep direction
    - You want to sweep mesh a thin solid with a single element through the thickness
    - You want to inflate the mesh near the surface of the geometry



Details of "Sweep Method" - Method

| | |
|---|---|
| **Scope** | |
| Scoping Method | Geometry Selection |
| Geometry | 1 Body |
| **Definition** | |
| Suppressed | No |
| Method | Sweep |
| Element Midside Nodes | Use Global Setting |
| Src/Trg Selection | Automatic |
| Source | |
| Target | |
| Free Face Mesh Type | |
| Type | |
| Sweep Num Divs | Default |
| Sweep Bias Type | No Bias |
| Element Option | Solid |

Automatic
Manual Source
Manual Source and Target
Automatic Thin
Manual Thin

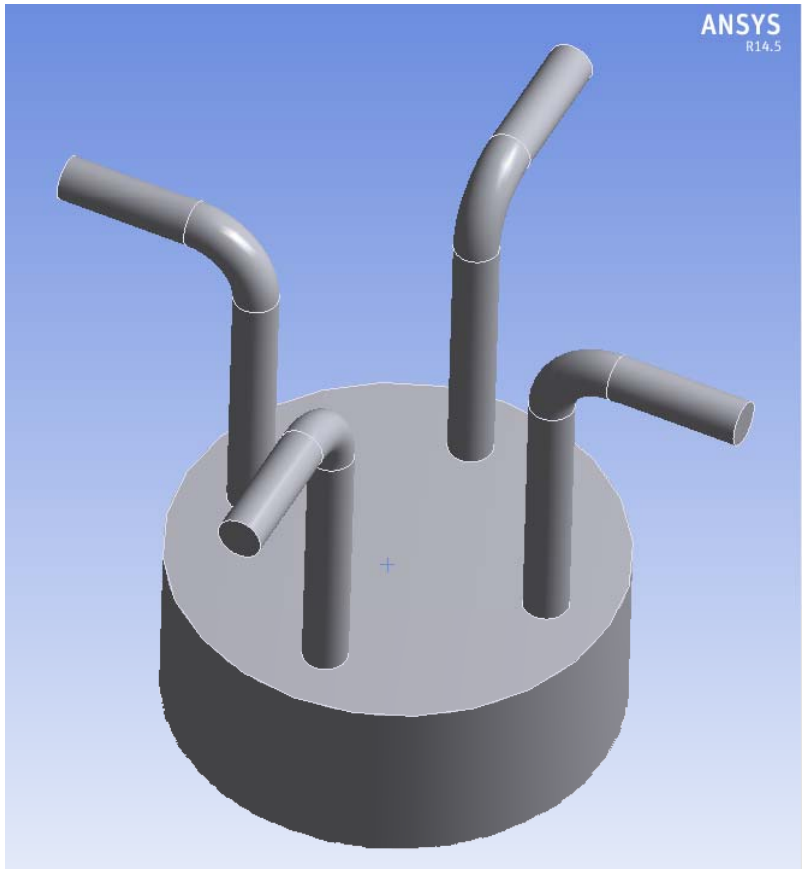
# Mesh Methods

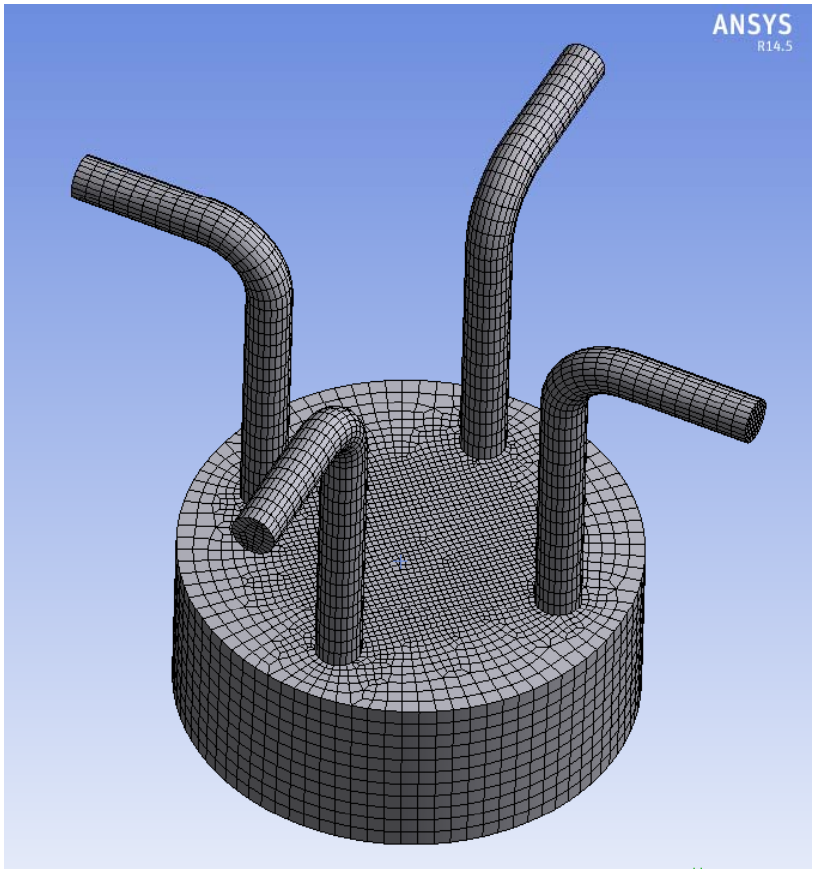
- Sweep Mesh Method
  - The number of elements in the sweep direction along with the bias settings are also set in the sweep details.

| Details of "Sweep Method" - Method | |
|---|---|
| **Scope** | |
| Scoping Method | Geometry Selection |
| Geometry | 1 Body |
| **Definition** | |
| Suppressed | No |
| Method | Sweep |
| Element Midside Nodes | Use Global Setting |
| Src/Trg Selection | Automatic |
| Source | Program Controlled |
| Target | Program Controlled |
| Free Face Mesh Type | Quad/Tri |
| Type | Number of Divisions |
| Sweep Num Divs | 8 |
| Sweep Bias Type | _ ___ _____ ___ _ |
| Sweep Bias | 2. |
| Element Option | Solid |

# Mesh Methods

- The MultiZone Method uses the Hexa blocking method (courtesy of the ICEM Advanced Meshing Module) that internally segments bodies into topologically consistent pieces.



# Mesh Methods

- Tetrahedral Mesh Methods
  - Two algorithms exist for tetrahedral meshing

- Algorithm = Patch Conforming:
  - The mesh must conform to the boundaries of the faces, yielding a very fine mesh in regions with small faces.
  - Default method

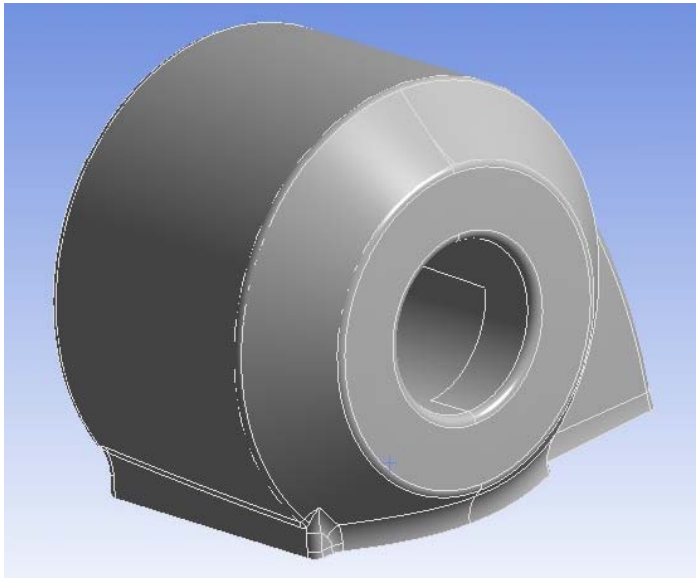
| Details of "Patch Conforming Method" - Method | |
|---|---|
| **Scope** | |
| Scoping Method | Geometry Selection |
| Geometry | 1 Body |
| **Definition** | |
| Suppressed | No |
| Method | Tetrahedrons |
| Algorithm | Patch Conforming |
| Element Midside Nodes | Use Global Setting |

- Algorithm = Patch Independent:
  - The mesh is not required to conform to the boundaries of the faces.
  - This is useful when there are many small faces which would normally produce a very fine mesh.

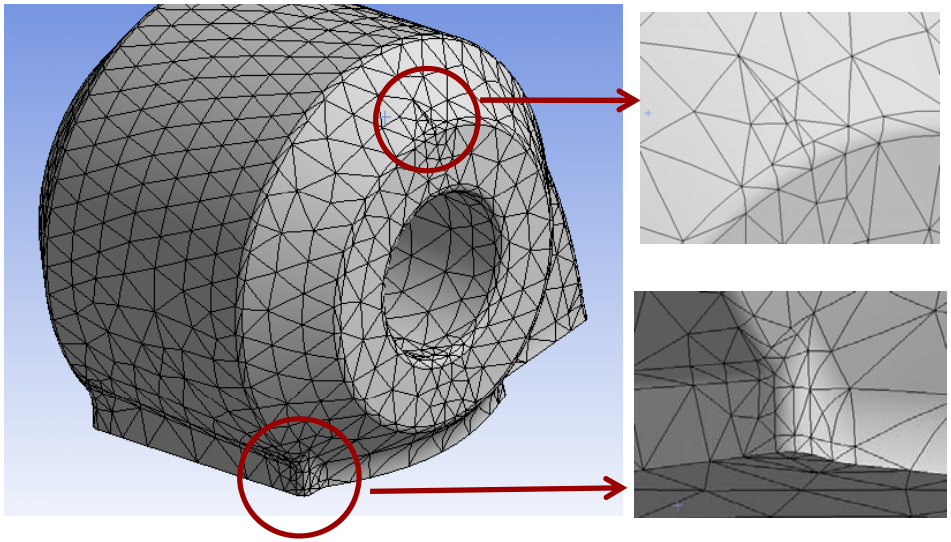
| Details of "Patch Independent" - Method | |
|---|---|
| **Scope** | |
| Scoping Method | Geometry Selection |
| Geometry | 1 Body |
| **Definition** | |
| Suppressed | No |
| Method | Tetrahedrons |
| Algorithm | Patch Independent |
| Element Midside Nodes | Use Global Setting |
| **Advanced** | |

# Mesh Methods

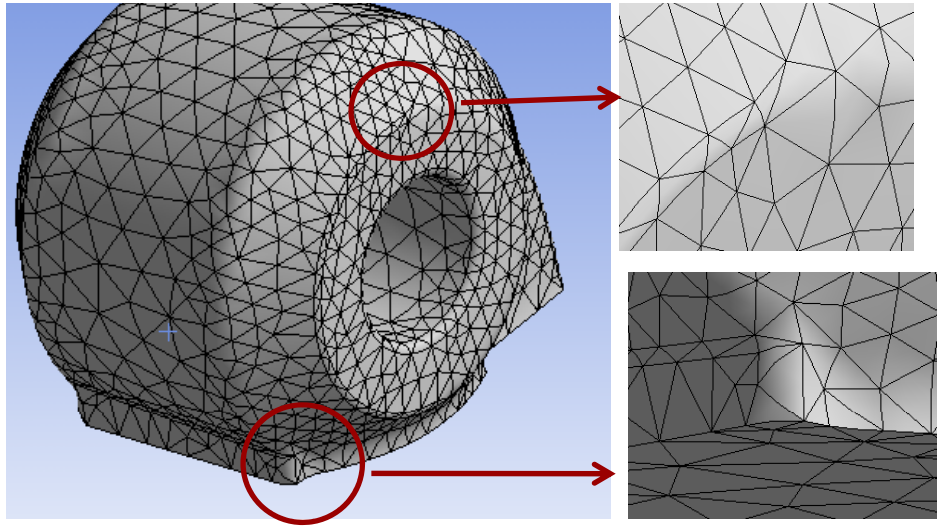
- Tetrahedral Mesh Example:
  - Geometry with very small faces relative to the overall geometry.



Patch conforming mesh: 28,050 elements

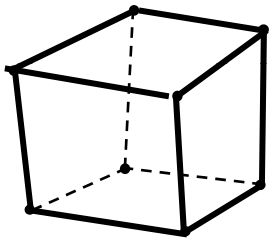


Patch independent mesh: 11,579 elements

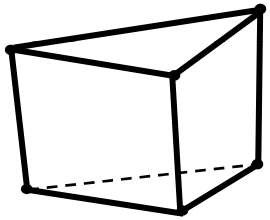


# Efficient CFD Meshing

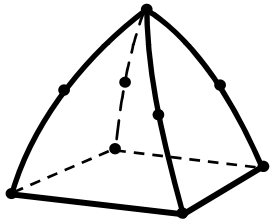
- Why do I want a hex mesh?
  - Elements generally come in a few basic shapes:
    - Hexahedron (hexas, bricks)
    - Pentahedron (wedges, prisms)
    - Tetrahedron
    - Pyramids



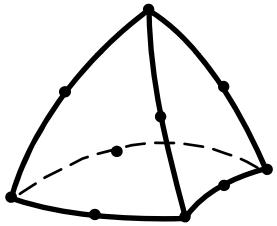
Hex



Wedge



Pyramid

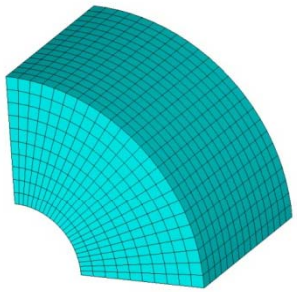


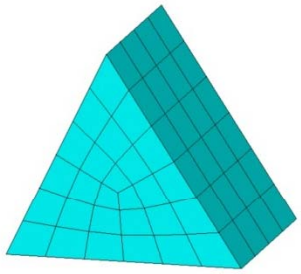
Tet

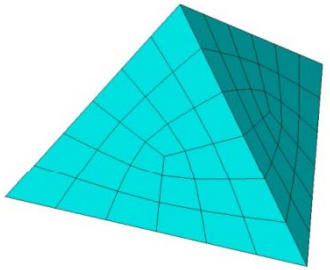
  - Hexahedrons can fill a given volume more efficiently than other mesh shapes
    - It takes approximately 5-6 tetrahedrons to fill 1 hexahedron.
    - Fewer elements lead to faster solution times
  - Hexahedron meshes are generally more uniform
    - Easier control of the element distribution
  - Hexahedrons can be more accurate when aligned with the flow direction
    - Not always possible to align the mesh with the flow direction

# How Do I Get a Hex Mesh?

- What Steps Do I Need to Take?
  - Hex meshes always take more man-time than tetrahedral meshes.
  - Hex meshes are usually created using one or more of the following approaches:
    - Slice and dice the solid geometry to create 6-faced volumes, each face having 3 or 4 boundary curves.
    - Create a 2D mesh of quadrilaterals and sweep or extrude the 2D mesh to create 3D hexas. Typical operations include:
      - Revolve the mesh about an axis.
      - Sweep the mesh along one or more curves.
      - Sweep a surface mesh from one face of a volume to a topologically similar face on the "opposite" face of a volume.
  - Hex meshes can also be created with advanced meshing methods.

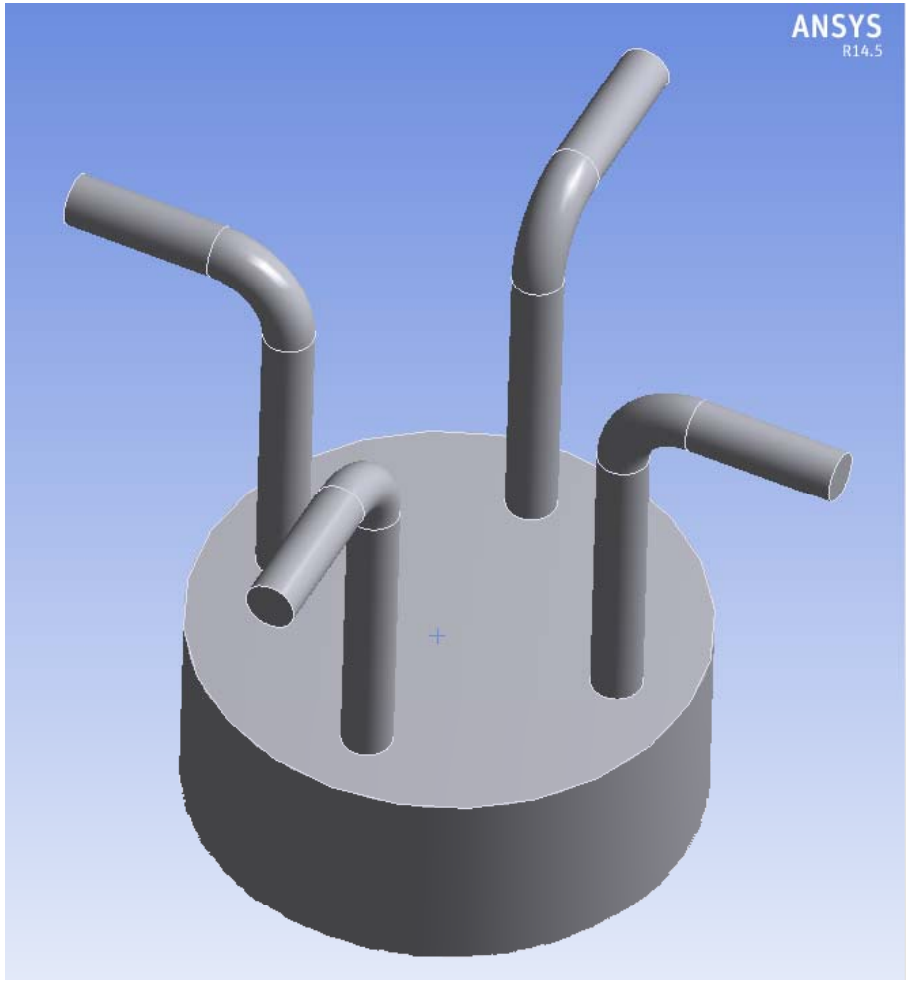






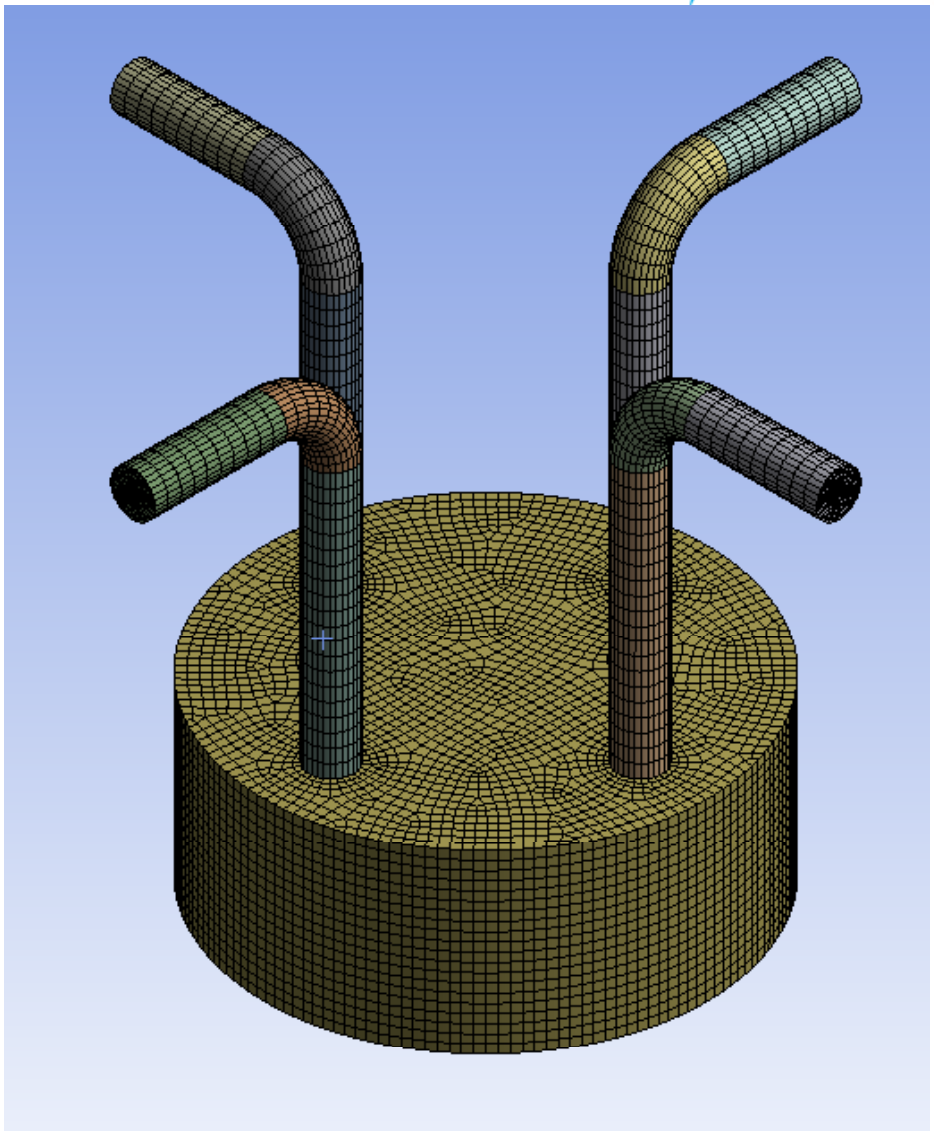
# How Do I Get a Hex Mesh?

- Consider the part shown below. While not sweepable in its present form, an all-hex mesh can be created with some effort by the user.



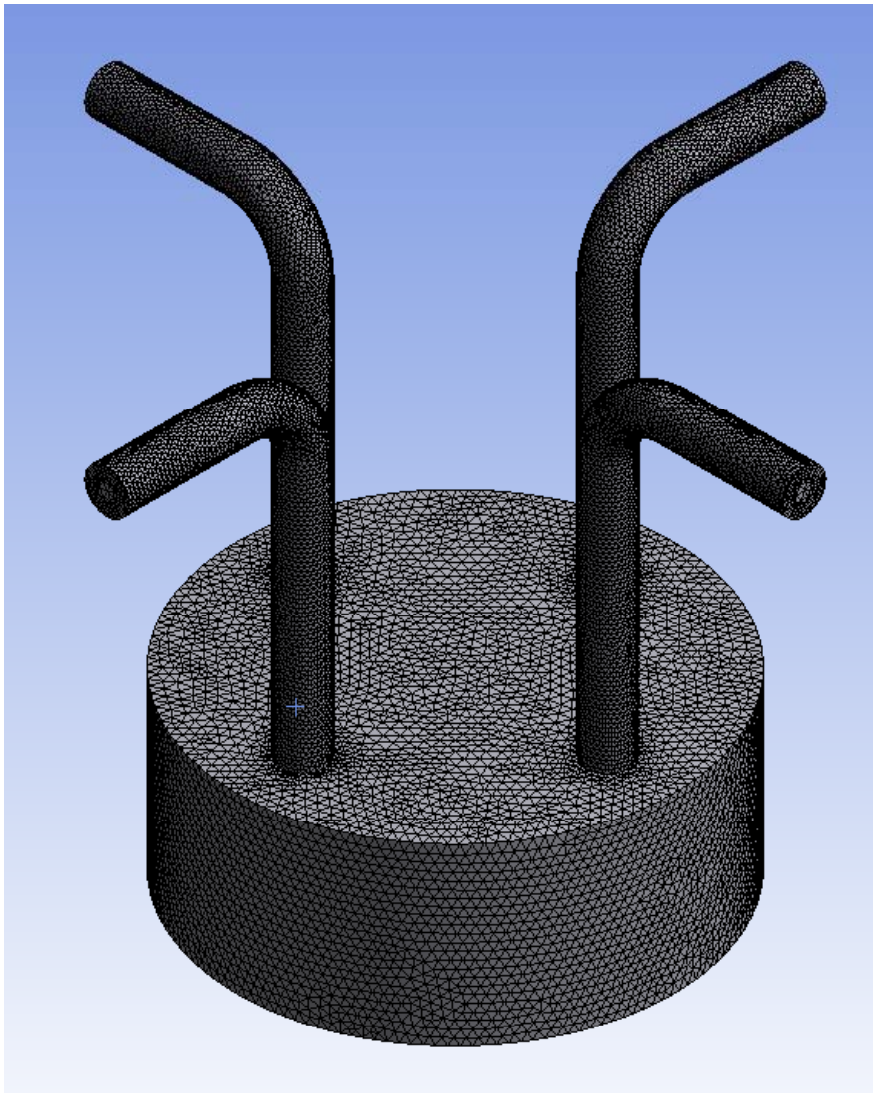
# How Do I Get a Brick Mesh?

- By creating a few slices in DesignModeler and forming the resulting bodies into a part, we can get a collection of sweepable bodies.
- The mesh is generated to capture the inlet pipe curvature and with 5 inflation layers along the walls.
- The resulting mesh contains 125,780 elements.



# How Do I Get a Hex Mesh?

- The corresponding all tetrahedron mesh resulting from the unaltered geometry is shown to the right.
- Again, the mesh is generated to capture the inlet pipe curvature and with 5 inflation layers along the walls. The part is meshed with very little effort.
- The resulting mesh contains 487,354 elements!

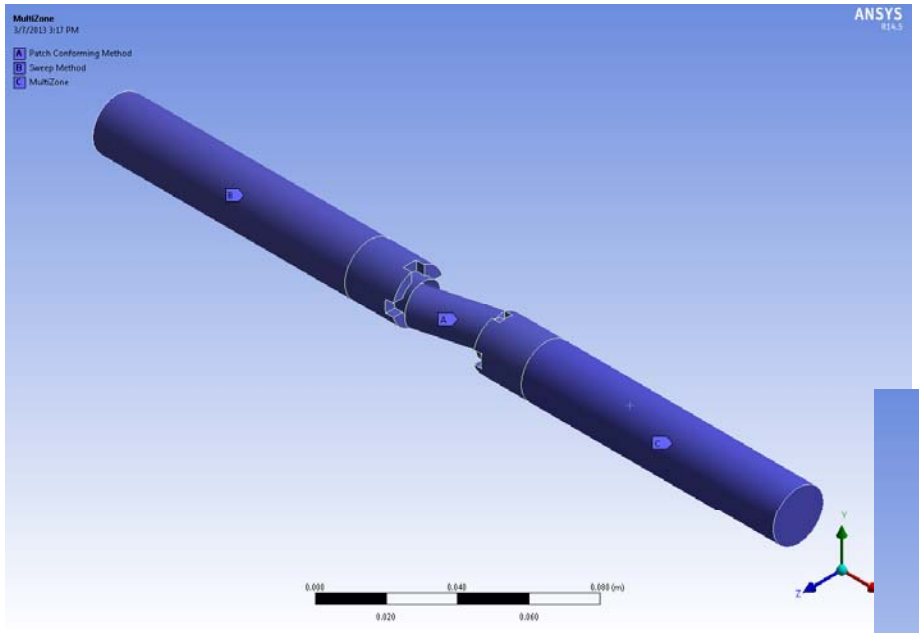


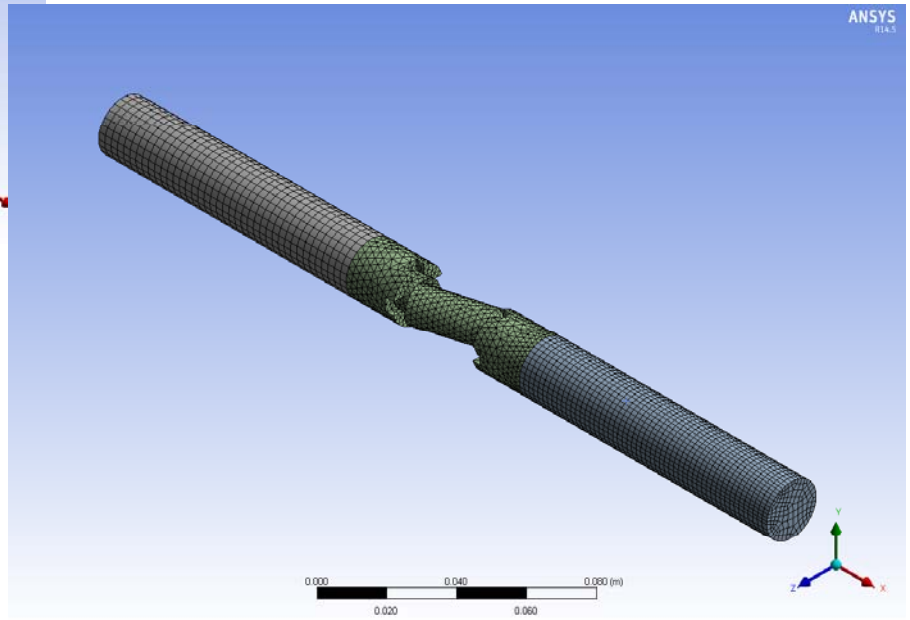
# How Do I Get a Hex Mesh?

- So How Do I Get a Hex Mesh in Workbench?
  - Recall that Several Mesh Methods will create Hex meshes
    - Sweep
    - Hex Dominant
    - Multizone
  - Each method has different geometry requirements
    - Important to understand how they behave!
  - What if I can't get a mesh that's entirely hexas?
    - Selective meshing to the rescue!
    - Try to mesh with as many hexas as possible, then fill the remainder with tets.

# Combining Methods

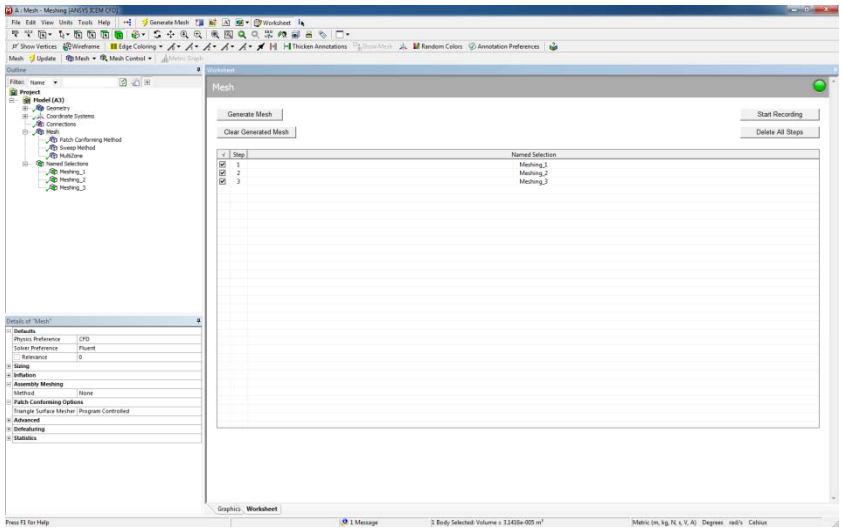
- Mesh methods defined on individual bodies remain valid when those bodies are formed into a part!

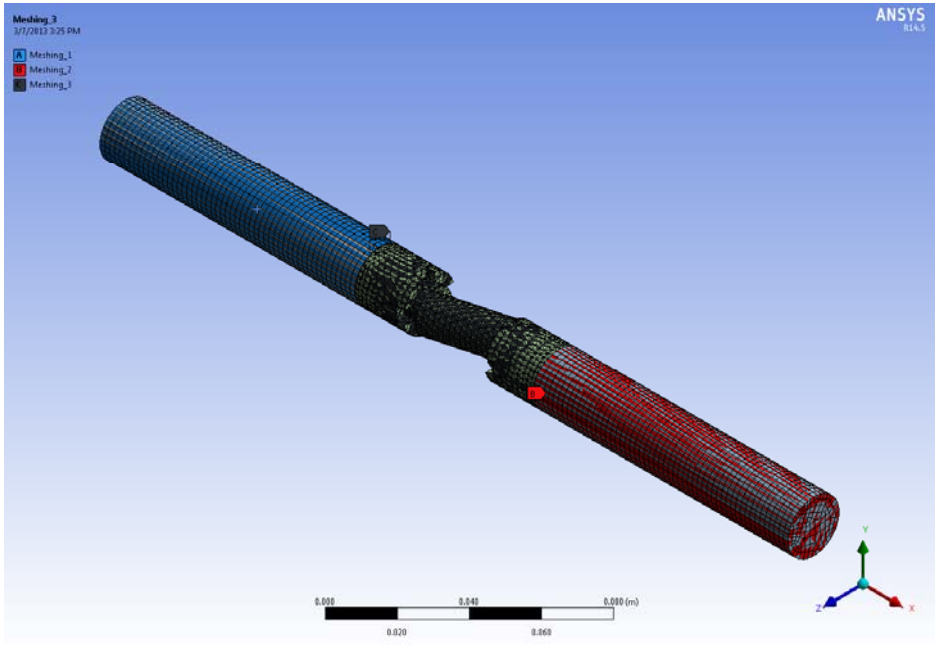


# Selective Meshing

- The Worksheet records the body order for Selective meshing.
  - Named selections are created to control the order of meshing (not sent to the solver)
  - Note: The automatic method is still used for parts without a specified mesh method.



# Summary

- Many different options within ANSYS Meshing for generating an efficient CFD mesh
- Hex meshes are more efficient and controllable
- Ease of use of ANSYS Workbench makes generating high fidelity meshes a simple process