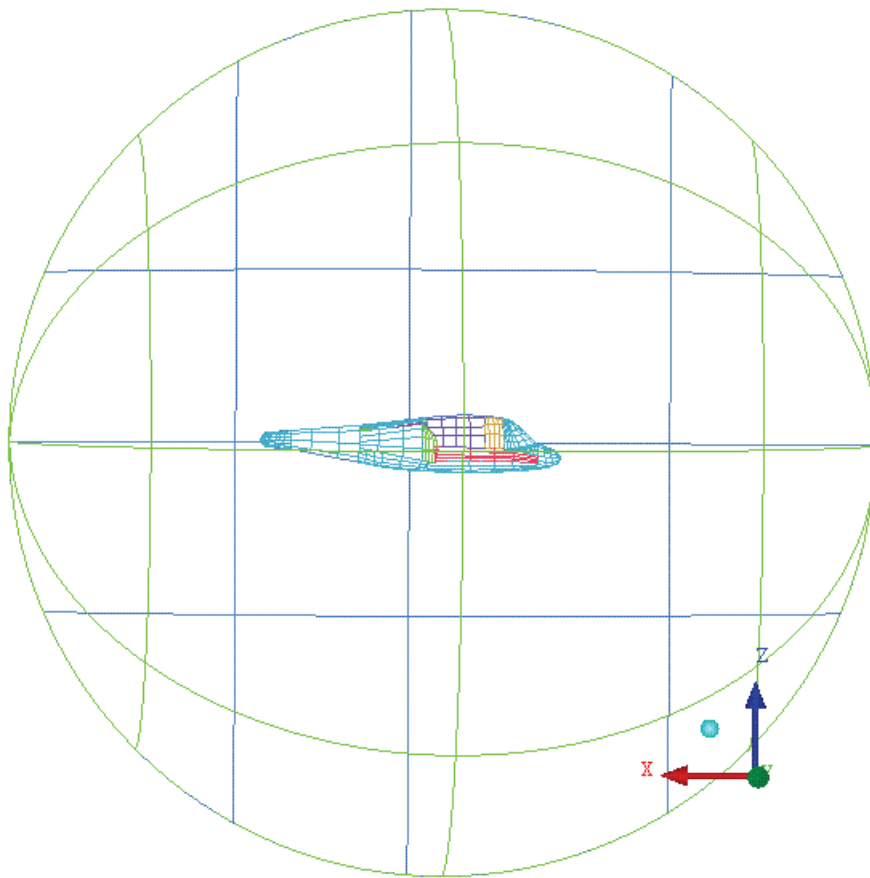

Tetra/Prism Mesh Generation for a Helicopter

This tutorial demonstrates the generation of the Tetra/Prism mesh for a helicopter.

The far-field used in the tutorial is smaller than normal to reduce the runtime.

Figure: Helicopter Geometry



This tutorial demonstrates how to do the following:

- Extract feature curves from the symmetry plane.
- Create the material point.
- Generate the Octree mesh.
- Manipulate the mesh display using cut planes.
- Generate the Delaunay mesh.
- Smooth the mesh.
- Verify and save the mesh.

Preparation

- Step 1: Preparing the Geometry
- Step 2: Creating a Material Point
- Step 3: Generating the Octree Mesh
- Step 4: Generating the Delaunay Mesh
- Step 5: Smoothing the Mesh
- Step 6: Saving the Project

Preparation



1. Copy the input geometry file (**helicopter.tin**) from the ANSYS installation directory under **v140/icemcfd/Samples/CFD_Tutorial_Files/Helicopter** to the working directory.
2. Start ANSYS ICEM CFD and open the geometry (**helicopter.tin**).

File > Geometry > Open Geometry...

Step 1: Preparing the Geometry

In most cases, you would put the parts comprising the helicopter fuselage into a single part and then build topology to extract feature curves. In this tutorial, you will retain each surface in its own part to better illustrate the patch-independence. Also, you will skip the build topology step and instead extract the feature curve from the symmetry plane.

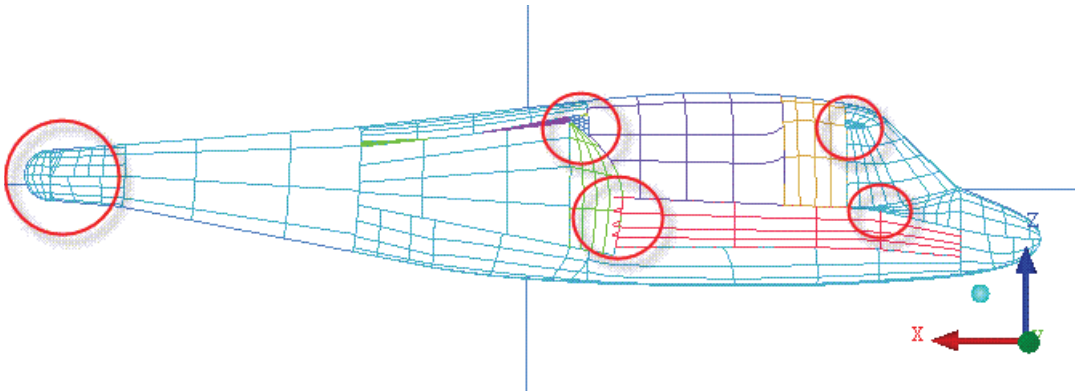
1. Enable **Surfaces** under **Geometry** in the display control tree.

 **Geometry**  **Surfaces**



2. Zoom in to the helicopter fuselage and examine the geometry.

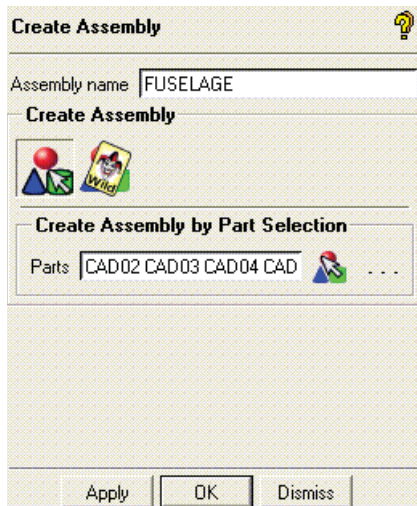
The overlapping surfaces and slivers are shown in *Figure: Overlapped Surfaces and Slivers* (p. 108).

Figure: Overlapped Surfaces and Slivers



3. Create an assembly for all the fuselage surfaces.

 **Parts**  **Create Assembly**



a. Enter **FUSELAGE** for **Assembly name** in the **Create Assembly** DEZ.


b. Click  (Select part(s)).

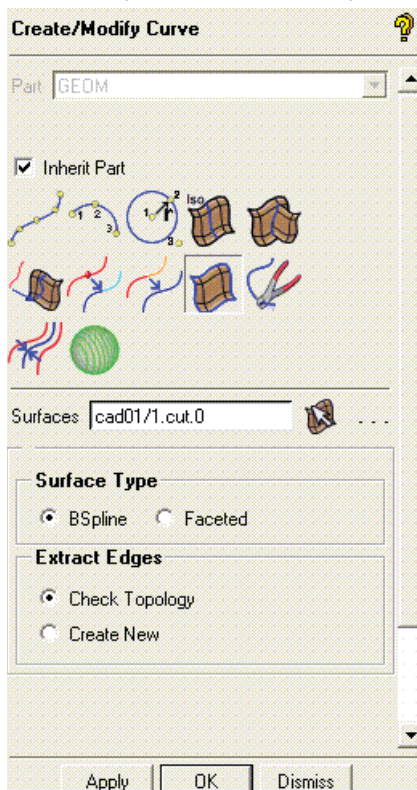
*The **Select parts** dialog will appear.*

c. Select all the parts except **FF** and **BIGSYM** in the **Select parts** dialog box and click **Accept**.



d. Click **Apply** in the **Create Assembly** DEZ.

4. Extract the feature curve from the symmetry plane.

Geometry > Create/Modify Curve  **> Extract Curves from Surfaces** 



a. Disable all parts except the symmetry, **BIGSYM** in the display control tree.

- b. Click  (**Select surface(s)**) and click  (**Select all appropriate visible objects**) in the selection toolbar.

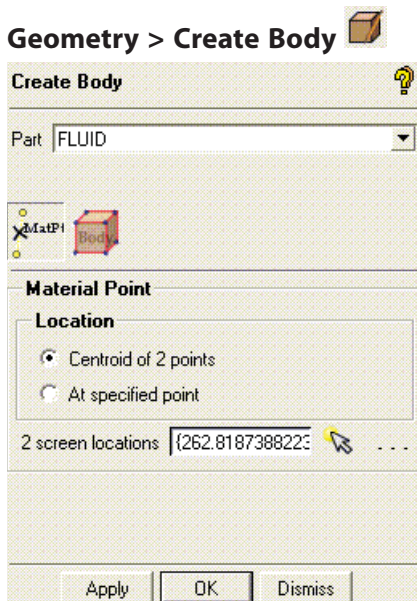
*You need not click the middle-mouse button when using the **Select all appropriate visible objects** option. You can also type **v** to select visible objects.*

Note

In this example, it may be easier to select the symmetry plane surface using a single left-button click. For complex models which have more surfaces, it will be easier to use **Select all appropriate visible objects** or **Select items in a part**, or even the box selection.

- c. Click **Apply**.

Step 2: Creating a Material Point




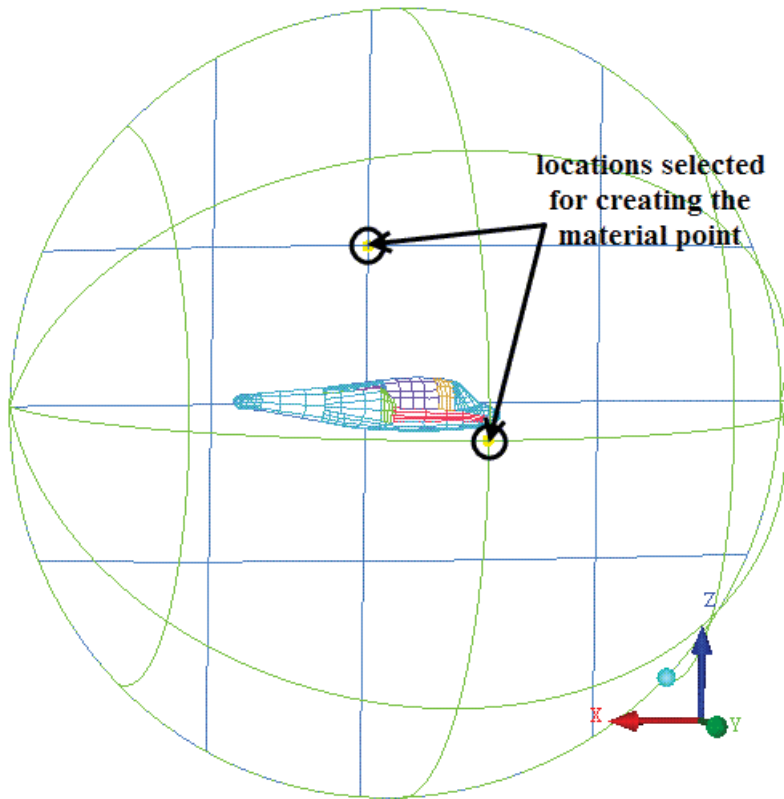
1. Enter **FLUID** for **Part**.
2. Ensure that **Points** is enabled in the display control tree.
3. Retain the selection of **Centroid of 2 points** for **Location**.
4. Click  (**Select location(s)**) and select two locations such that the midpoint lies within the volume (one on the symmetry surface and other on the far-field surface, see [Figure: Selection of Points for Creating Material Point](#) (p. 111)). Click the middle-mouse button to accept the selection of the points.

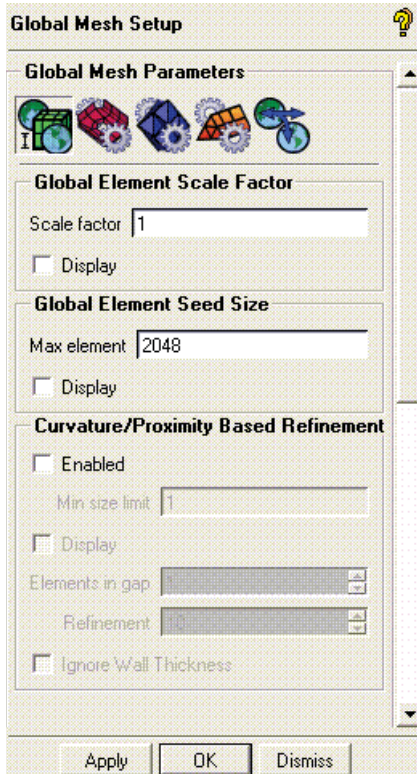
Figure: Selection of Points for Creating Material Point

5. Click **Apply** so that **FLUID** appears under **Parts** in the display control tree.

Step 3: Generating the Octree Mesh

1. Assign the mesh sizes.

Mesh > Global Mesh Setup  > Global Mesh Size 



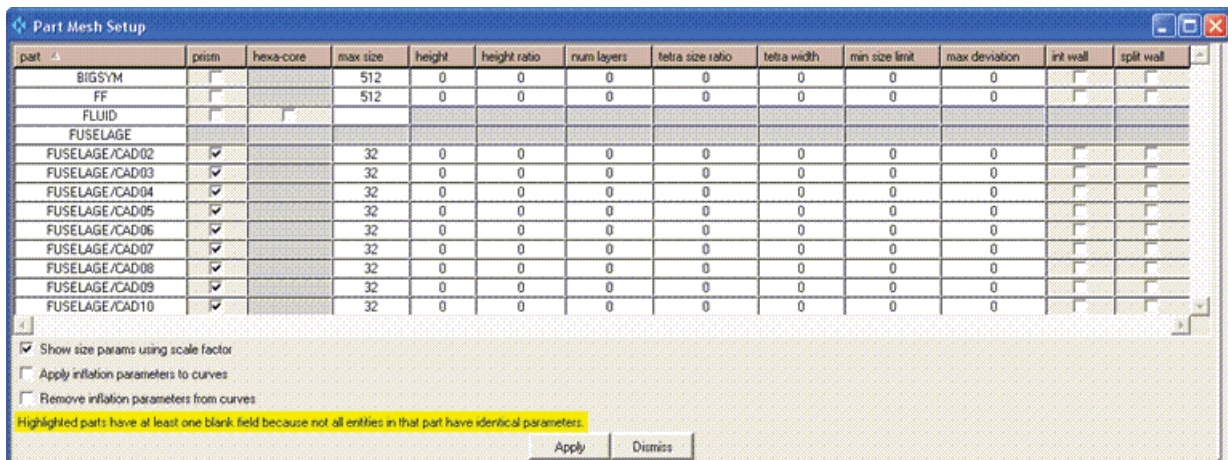
- a. Retain the value of 1 for **Scale factor**.
- b. Enter 2048 for **Max element**.

*The value for **Max element** is chosen to be 2048 because it is a power of two which is important for Octree mesh generation.*

- c. Click **Apply**.

2. Specify the parts for prism creation.

Mesh > Part Mesh Setup

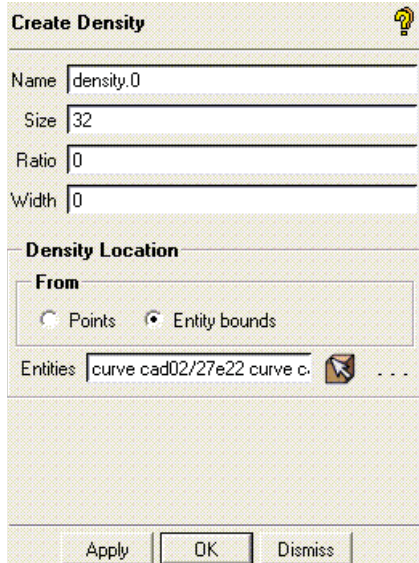


- a. Enable **prism** for the fuselage parts.
- b. Ensure that prism **height** is set to 0.

Setting the prism height to zero allows it to "float".

- c. Retain the default settings for other parameters.
 - d. Click **Apply** and then **Dismiss**.
3. Create a density box of size 32.

Mesh > Create Mesh Density



Create Density

Name: density.0

Size: 32


Ratio: 0

Width: 0



Density Location

From:

☐ Points ☒ Entity bounds

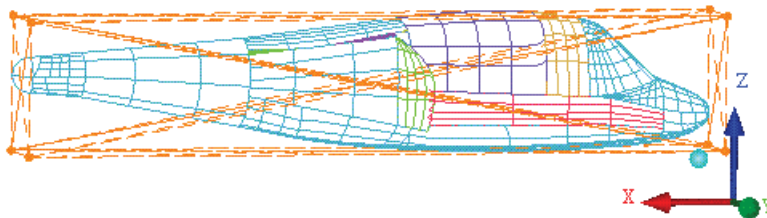
Entities: curve cad02/27e22 curve c...  ...

Apply OK Dismiss

- a. Disable **FF** and **BIGSYM** in the display control tree.
- b. Enter 32 for **Size**.
- c. Select **Entity bounds** for **Density Location**.
- d. Click  (**Select geometry**) and then  (**Select all appropriate visible objects**) in the selection toolbar.
- e. Click **Apply** in the **Create Density** DEZ.

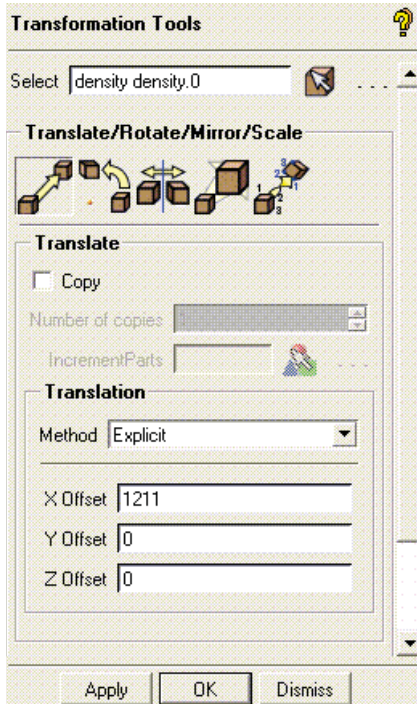
A density box will be created around the fuselage (Figure: Density Box Around the Fuselage (p. 113)).


Figure: Density Box Around the Fuselage




4. Move the density box to the rear end of the fuselage.

Geometry > Transform Geometry > Translate Geometry



- a. Select  (**Measure Distance**) from the utilities and measure the length of the aircraft.

The length of the aircraft is around 2422.

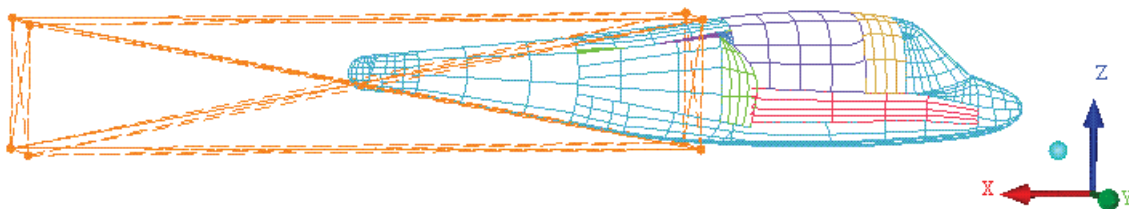
- b. Click  (**Select geometry**) and select the density box in the graphics window.
- c. Enter 1211 for **X Offset**.

*The **X Offset** value is chosen approximately half the length of the aircraft.*

- d. Click **Apply**.

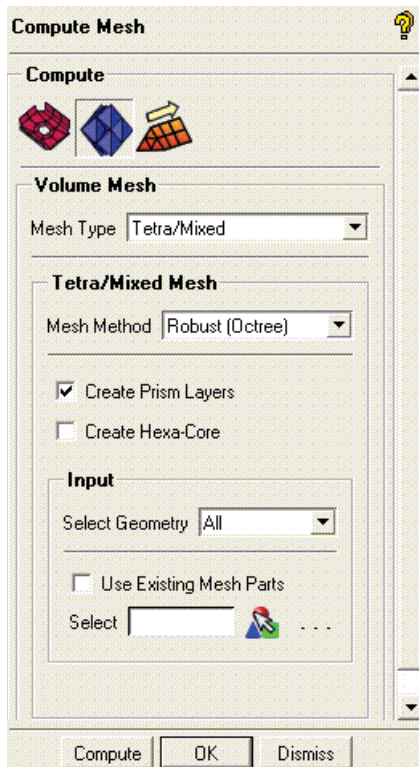
The density box is translated half the length of the fuselage (Figure: Density Box Translated to the Wake Region (p. 114). You will use this to refine the wake region.

Figure: Density Box Translated to the Wake Region



5. Compute the mesh.

Mesh > Compute Mesh  **> Volume Mesh** 





- a. Ensure that the **Mesh Method** is set to **Robust (Octree)**.
- b. Enable **Create Prism Layers**.
- c. Click **Compute**.

The progress of meshing will be reported in the message window.

6. Examine the mesh (*Figure: Octree Mesh for Helicopter* (p. 116)).

- a. Disable the display of surfaces.

 **Geometry**  **Surfaces**

- b. Disable **FF** in the display control tree.

 **Parts**  **FF**

- c. Select **Solid & Wire**.

 **Mesh**  **Shells**  **Solid & Wire**

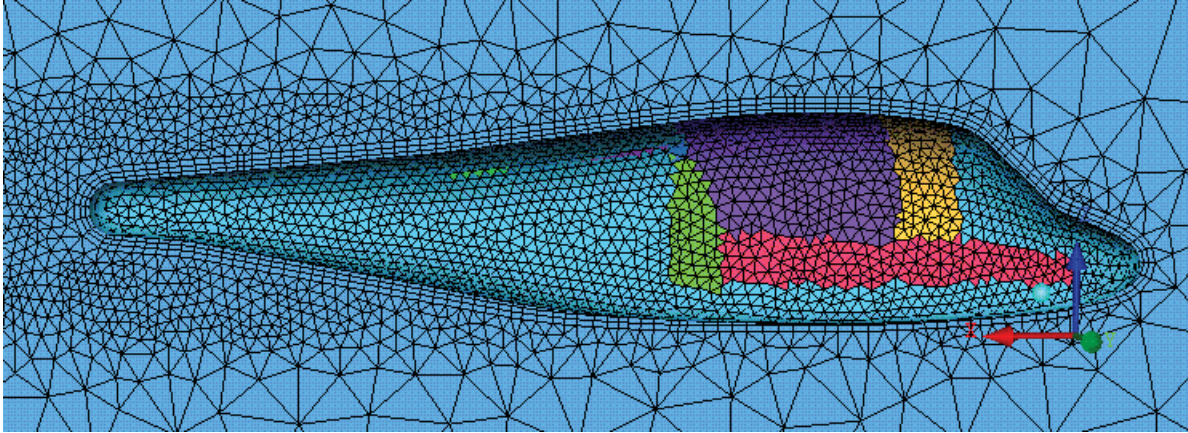
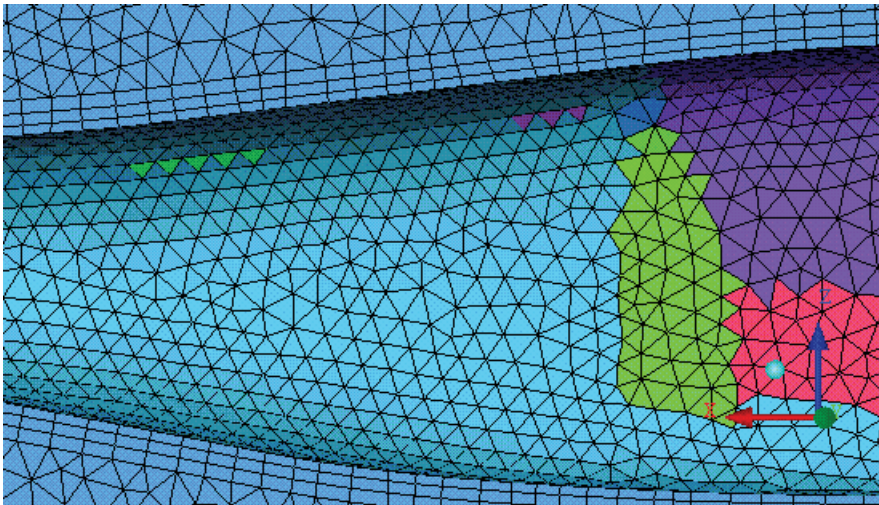
Figure: Octree Mesh for Helicopter

Figure: Octree Mesh for Helicopter (p. 116) shows the Octree mesh. The various colors help illustrate the patch independence. View the wake region and examine the prism layers. The prism height “floats” as the height was initially set to zero. The variation in layer thickness (float) is not significant for this model because the surface mesh size is relatively uniform. You may try with different mesh sizes, or with curvature based refinement for greater effect. Figure: Zoomed-in Mesh—Slivers Meshed with Equilateral Triangles (p. 116) shows the zoomed in slivers meshed with equilateral triangles.

Figure: Zoomed-in Mesh—Slivers Meshed with Equilateral Triangles**Note**

Some solvers may not like the volume transitions in the Octree mesh. Step 4 explains how you can replace the Octree volume mesh with a Delaunay volume mesh for smoother volume transition.

7. Use a cut plane to examine the mesh.
 - a. Select **Wire Frame**.

Mesh RMB → Shells LMB → Wire Frame

- b. Select **Manage Cut Plane**.

Mesh RMB → Cut Plane... LMB → Manage Cut Plane

- c. Set the following parameters:
- i. Select **by Coefficients** from the **Method** drop-down list.
 - ii. Enter 0.5 for **Bz**.
 - iii. Click **Apply**.
- d. Enable the display of volumes in the display control tree.

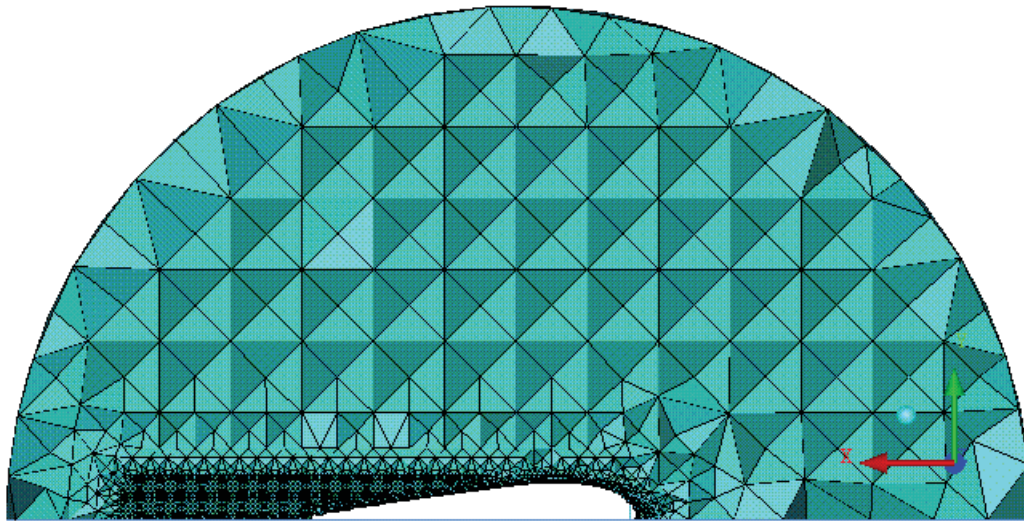
Mesh LMB → Volumes

- e. Select **Solid & Wire**.

Mesh RMB → Volumes LMB → Solid & Wire

The cut plane appears as shown in *Figure: Cut Plane in Z Direction for Octree Mesh (p. 117)*.

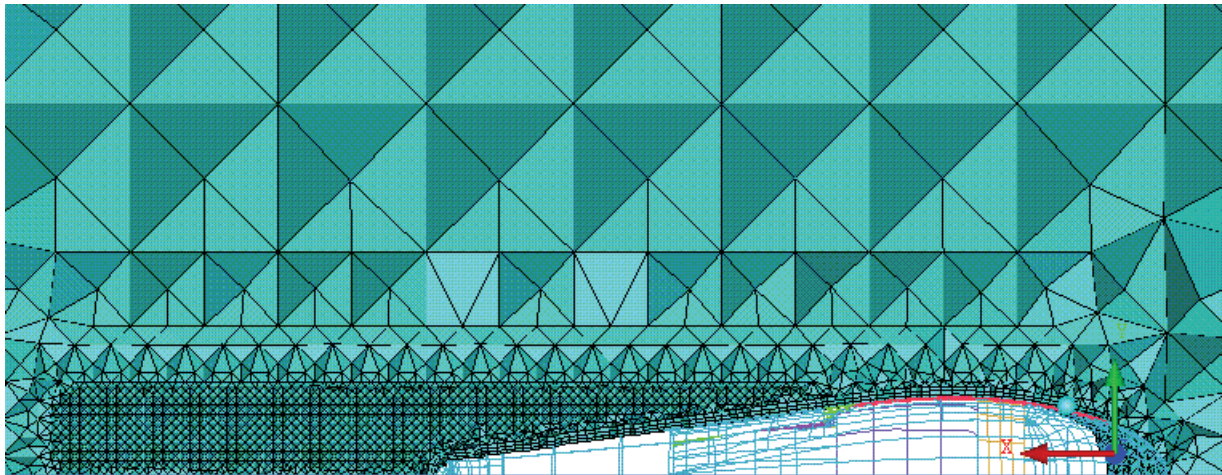
Figure: Cut Plane in Z Direction for Octree Mesh





- f. Enable **Surfaces**.



Geometry LMB → Surfaces

- g. Zoom in and examine the effect of the density box on the mesh (*Figure: Zoomed-in Cut Plane in Wake Region (p. 118)*).

Figure: Zoomed-in Cut Plane in Wake Region

- h. Disable the display of volumes and surfaces.

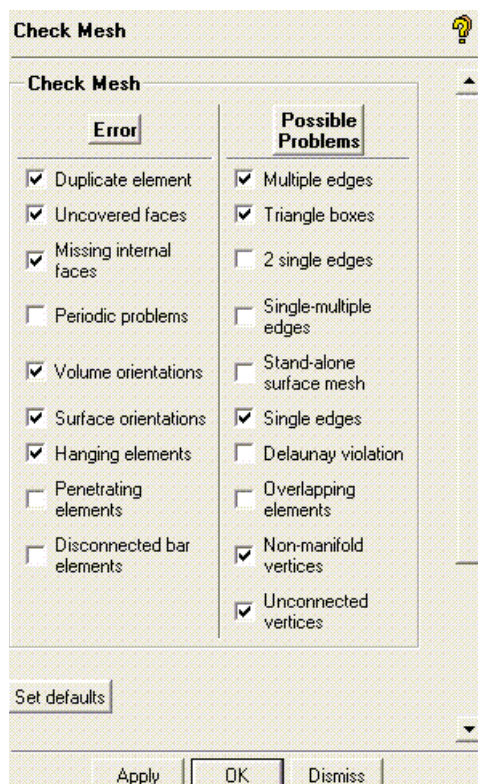
 **Mesh**  **Volumes**

 **Geometry**  **Surfaces**

- i. Disable **Show Cut Plane** in the **Manage Cut Plane** DEZ.

8. Check the mesh for any errors that may cause problems during the analysis.

Edit Mesh > Check Mesh



- a. Retain the default set of checks.

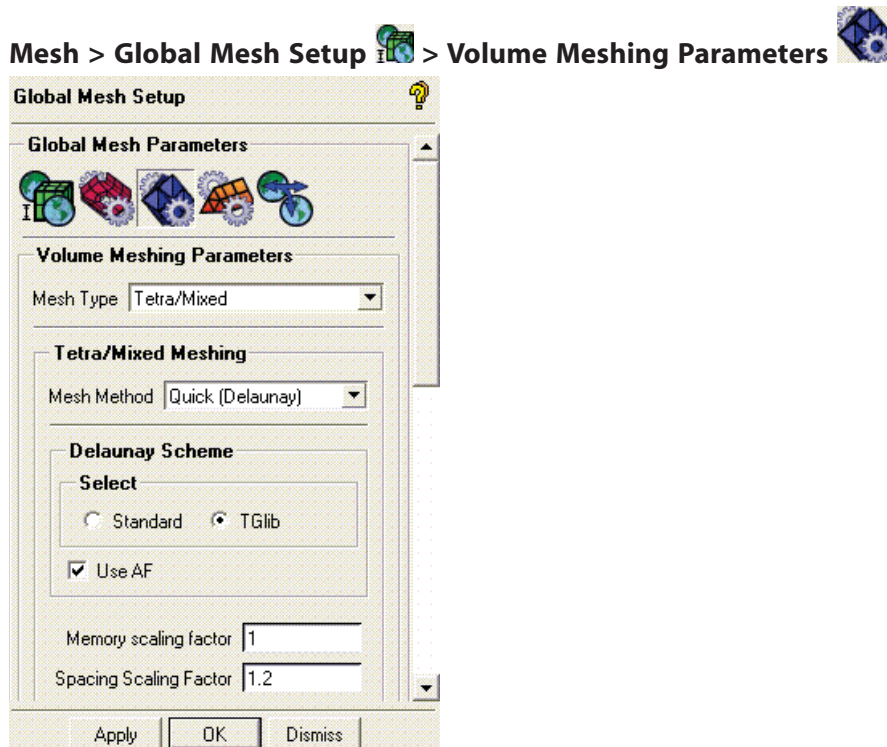
- b. Click **Apply** to check for errors and possible problems in the mesh.

Make sure no errors/problems are reported during the check.

Step 4: Generating the Delaunay Mesh

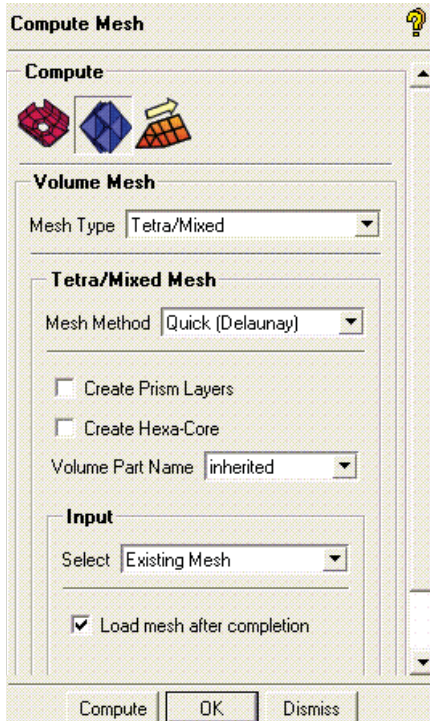
In this step, you will replace the Octree mesh with the Delaunay mesh because it fills the volume more efficiently and has smoother volume transition.

1. Set the volume mesh parameters.



- a. Select **Quick (Delaunay)** from the **Mesh Method** drop-down list.
 - b. Enter **1.2** for **Spacing Scaling Factor**.
 - c. Click **Apply**.
2. Compute the mesh.

Mesh > Compute Mesh  **> Volume Mesh** 



- a. Select **Quick (Delaunay)** from the **Mesh Method** drop-down list.
- b. Disable **Create Prism Layers**.
- c. Ensure that **Existing Mesh** is selected in the **Select** drop-down list.
- d. Enable **Load mesh after completion**.
- e. Click **Compute**.

The progress will be reported in the message window.

3. Examine the mesh (*Figure: Cut Plane in Z Direction for Delaunay Mesh* (p. 121)).
 - a. Select **Solid & Wire**.

 **Mesh**  **Shells**  **Solid & Wire**

- b. Examine the mesh using a cut plane.

Figure: Cut Plane in Z Direction for Delaunay Mesh (p. 121) and *Figure: Zoomed-in Cut Plane in Wake Region for Delaunay Mesh* (p. 121) show the cut planes for Delaunay mesh.

Figure: Cut Plane in Z Direction for Delaunay Mesh

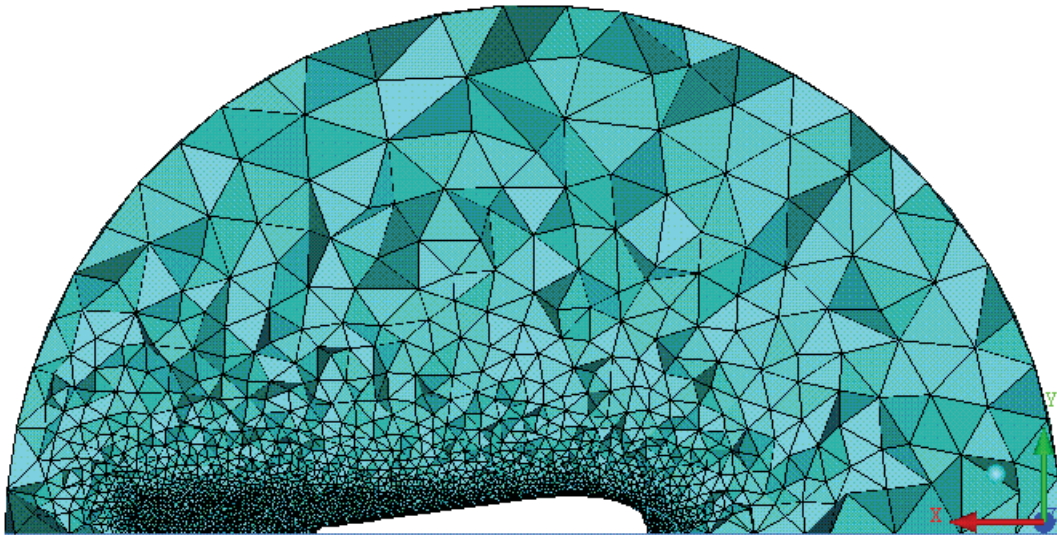
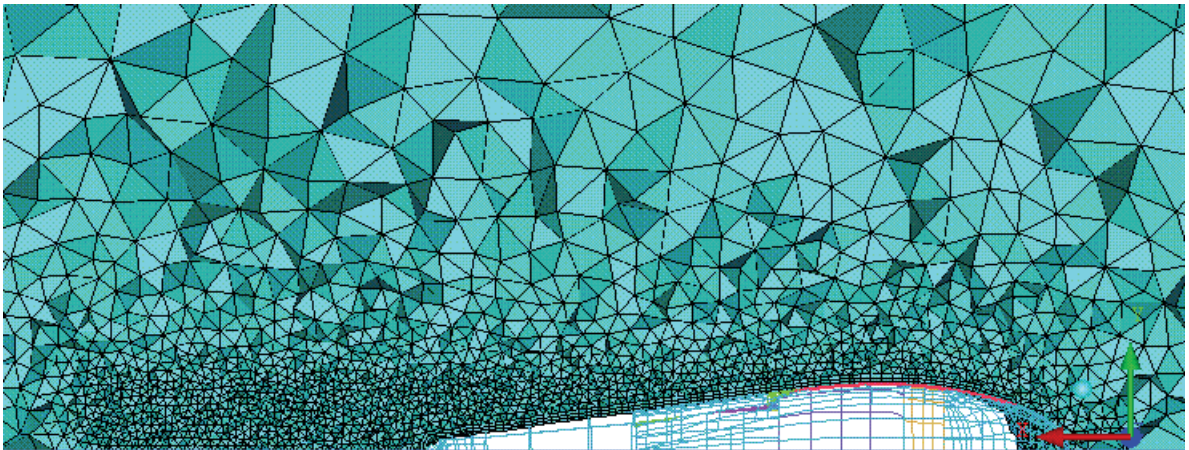


Figure: Zoomed-in Cut Plane in Wake Region for Delaunay Mesh



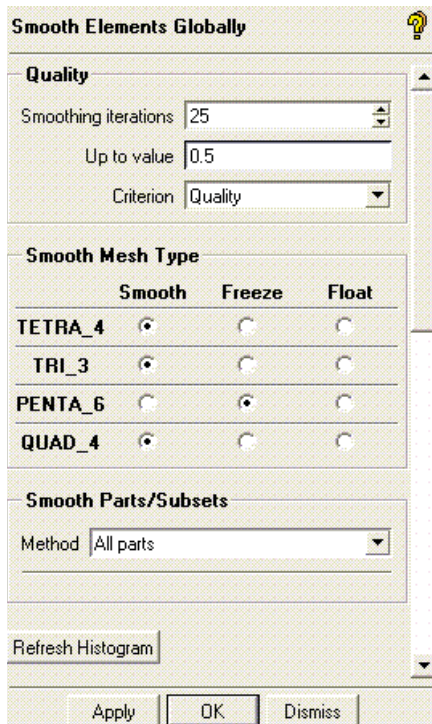
4. Check the mesh for any errors that may cause problems during the analysis.

Edit Mesh > Check Mesh 

Step 5: Smoothing the Mesh

In this step, you will smooth the mesh to improve the quality. The smoothing approach involves initial smoothing of the interior elements without adjusting the prisms. After initial smoothing, you will smooth the prisms as well.

Edit Mesh > Smooth Mesh Globally 



The quality histogram appears in the right hand corner.

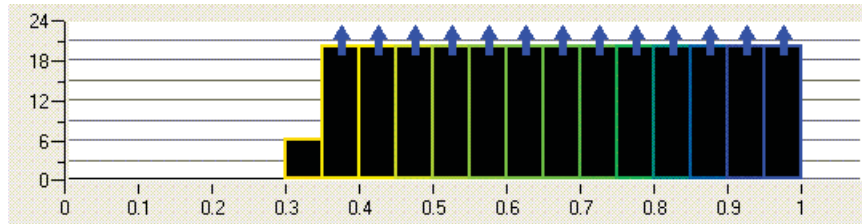
1. Smooth the interior elements without adjusting the prisms.
 - a. Enter 25 for **Smoothing iterations**.
 - b. Enter 0.5 for **Up to value**.
 - c. Retain the selection of **Quality** for **Criterion**.
 - d. Select **Freeze** for **PENTA_6**.

Note

The **PENTA_6** elements are five sided elements with six nodes (such as prism elements). These elements are usually ideal, but may be damaged by the smoother as it adjusts to optimize the adjacent tetra nodes. Freezing these prism elements (**PENTA_6**) protects them. If you smooth some prism elements, use a subset or reduce the number of smoothing iterations and the **Up to value** down to 0.01 so that only the worst elements are adjusted.

- e. Click **Apply**.

The quality histogram will be updated as shown in [Figure: Updated Quality Histogram](#) (p. 123).

Figure: Updated Quality Histogram

2. Smooth the interior elements including the prisms.
 - a. Enter 2 for **Smoothing iterations**.
 - b. Enter 0.01 for **Up to value**.
 - c. Select **Smooth** for **PENTA_6**.
 - d. Click **Apply**.

Step 6: Saving the Project

1. Save the project file (`helicopter-final.prj`).

File > Save Project As...

2. Save the output file for ANSYS FLUENT.

Output > Select solver



- a. Select **ANSYS Fluent** from the **Output Solver** drop-down list.
- b. Click **Apply**.

3. Set the appropriate boundary conditions.

Output > Boundary conditions



- Click **Accept** to set the boundary conditions.

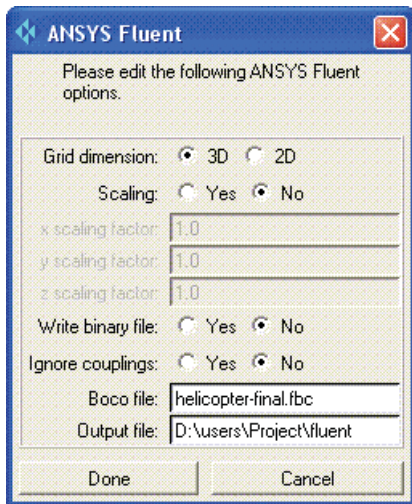
4. Write the input file for ANSYS FLUENT.

Output > Write input



- a. Select the appropriate **.uns** file.

The **ANSYS Fluent** dialog will appear.



- b. Enter `fluent` for **Output file**.
 - c. Click **Done**.
5. Exit the current session.

File > Exit