Altair

HyperWorks

Altair HyperMesh 2019 Tutorials

HM-4460: Composite

HM-4460: Composite

In this tutorial, you will learn how to:

- Load the Ansys user profile and CAD data.
- Mesh all of the surfaces at once, specifying element sizes and an element type.
- Import an Equivalent Fibersim model.
- Define the dummy properties and assign them to the mesh.
- Define an orientation for the component
- Use the Ply Realization and distribution table option
- Laminate Realize
- Create/Edit a distribution table
- Use the Ply thickness visualization -3D representation option

Model Files

This exercise uses the pillar_w_ncf.CATPart file, which can be found in <hm.zip>/interfaces/ansys/. Copy the file(s) from this directory to your working directory.

Exercise

Step 1: Load the ANSYS User Profile

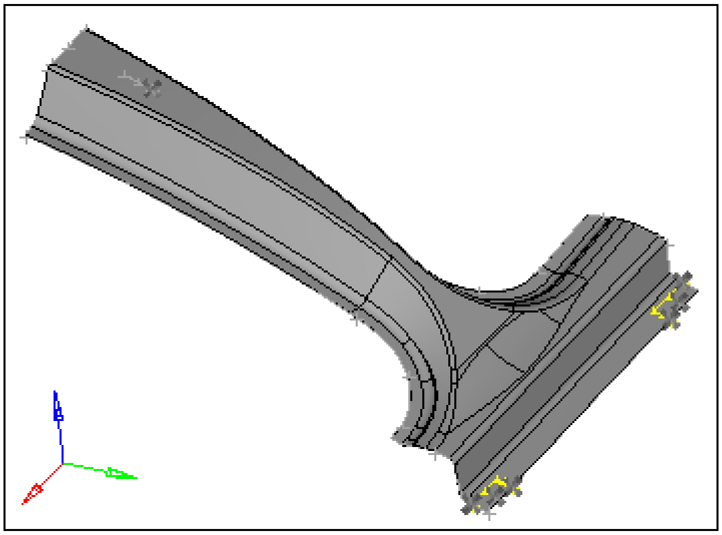
A set of standard user profiles is included in the HyperMesh installation. They include: OptiStruct, RADIOSS, Abaqus, Actran, ANSYS, LS-DYNA, MADYMO, Nastran, PAM-CRASH, PERMAS, and CFD. When a user profile is loaded, applicable utility menus are loaded, unused panels are removed, unneeded entities are disabled in the Find, Mask, Card and Reorder panels and specific adaptations related to the Ansys solver are made.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **Ansys**.

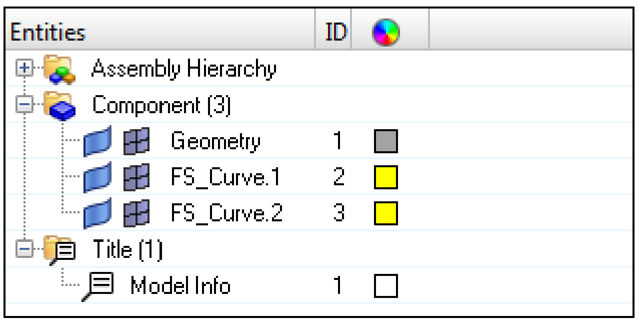
Step 2: Load the HyperMesh Model for this Tutorial

1. From the menu bar, click ***File*** > ***Import*** > ***Geometry***.
2. In the **Import** tab, set **File type** to ***CATIA***.
3. Click .
4. In the **Select CATIA file** dialog, open the pillar_w_ncf.CATPart file.
5. Click ***Import***. HyperMesh imports geometry data only.

   Note: You will import the Ply and Composite data later in this tutorial.

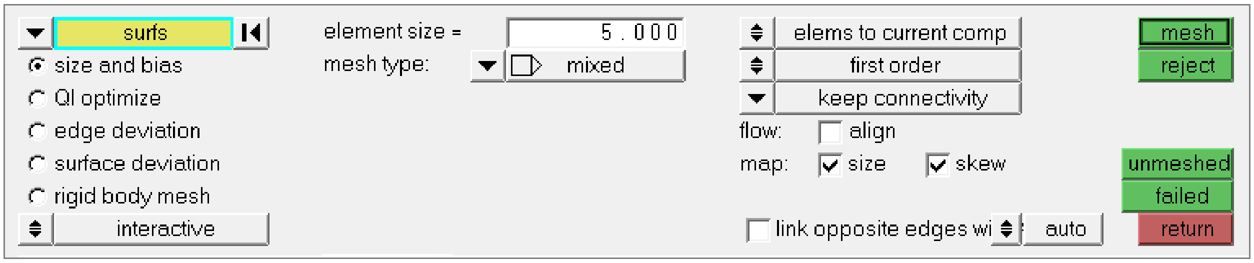


6. In the **Model** browser, review the model's contents.



Step 3: Mesh all of the Surfaces at Once, Specifying Element Sizes and Element Type

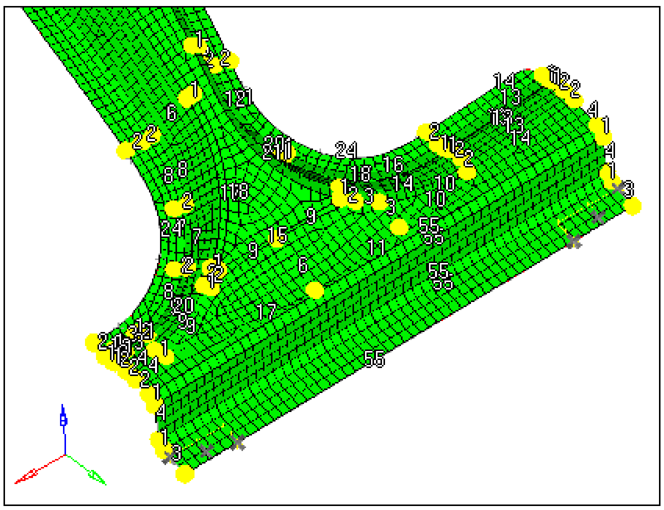
1. To open the **Automesh** panel, do one of the following:
   - From the menu bar, click ***Mesh*** > ***Create*** > ***2D AutoMesh***.
   - From the main menu, go to the ***2D*** page and click ***automesh***.
   - Press ***F12***.
2. Go to the **size and bias** subpanel.
3. Set the entity selector to ***surfs***.
4. Click ***surfs*** >> ***displayed***.
5. In the **element size=** field, enter `5`.
6. Set **mesh type** to ***mixed***.
7. Set the mesh mode to ***interactive***.
8. Set the **elements to surf comp/elements to current comp** toggle to ***elems to current comp***.



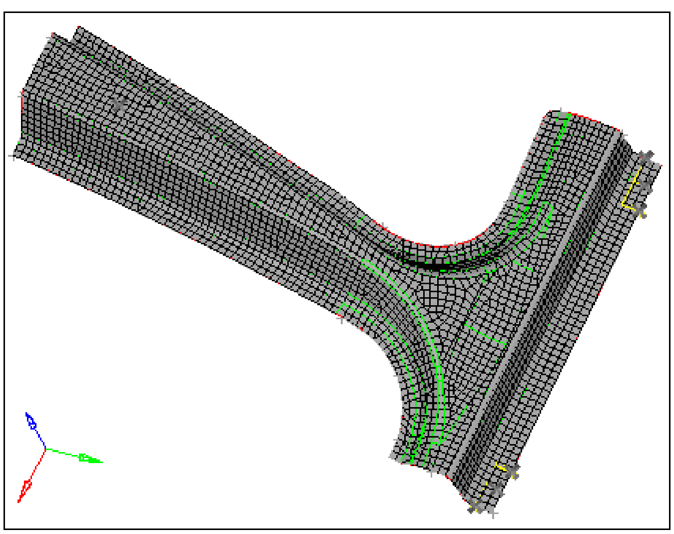
Size and bias subpanel settings for steps 2.3 through 2.8.

9. Click ***mesh***. The meshing module opens.

   Note: You should now be in the **density** subpanel of the meshing module. There is node seeding and a number on each surface edge. The number indicates the number of elements that were created along the edge.



10. Accept the mesh by clicking ***return***.

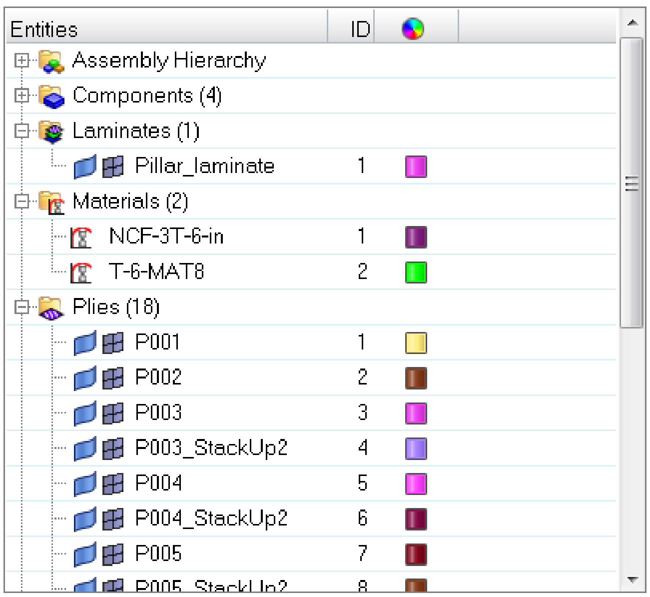


11. From the menu bar, click ***File*** > ***Save As*** > ***Model***.

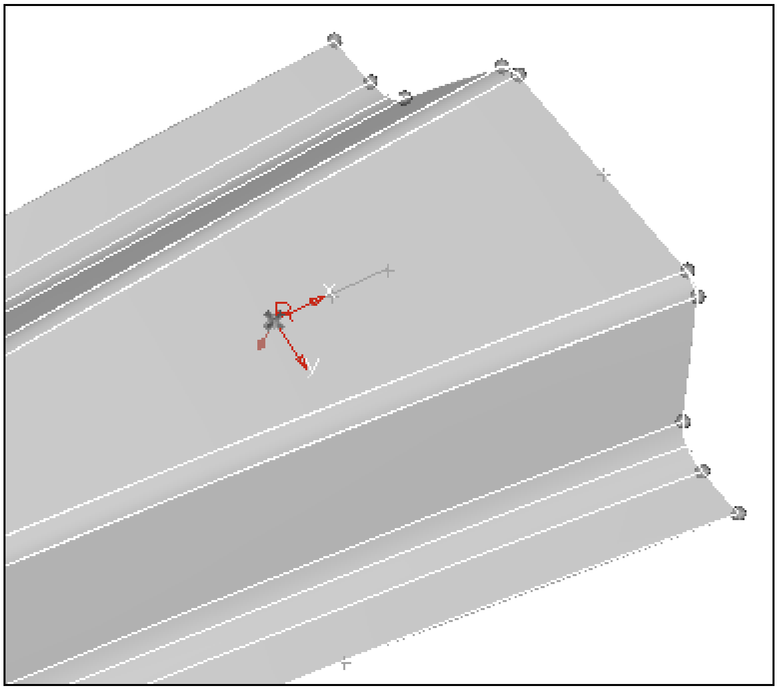
12. In the **Save Model As** dialog, navigate to your working directory and save the HyperMesh database with the name `pillar_w_ncf_FINAL.hm`.

Step 4: Load the Ply Information from FiberSim

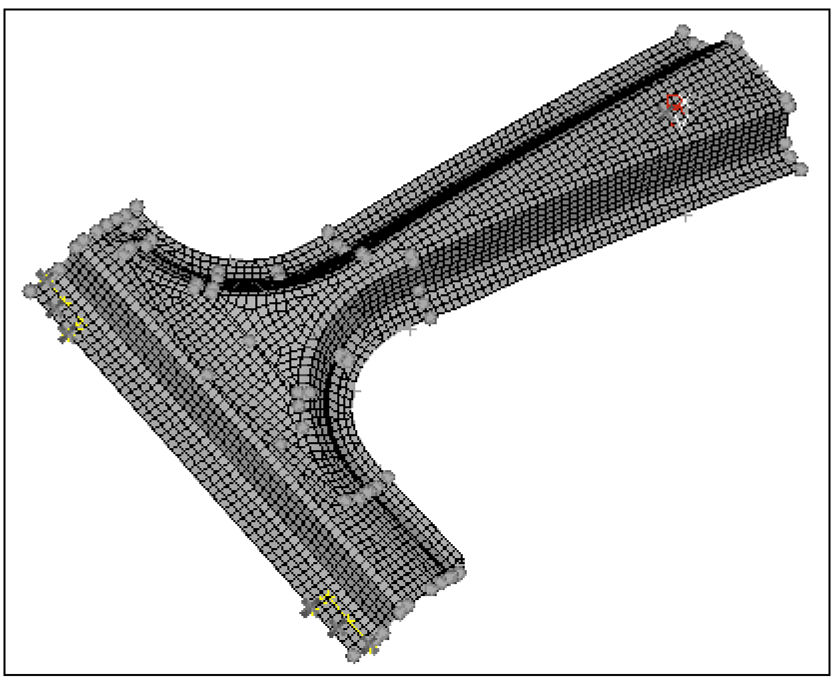
1. From the menu bar, click ***File*** > ***Import*** > ***Geometry***.
2. In the **Import** tab, set **File type** to ***FiberSim***.
3. Click .
4. In the **Select FiberSim file** dialog, open the `pillar.h5` file.
5. Click ***Import***. HyperMesh imports and populates the database with laminate data (ply book and ply stacking data), composite material information, each ply data (triangular elements spanning a single ply), and a coordinate system.
6. In the **Model** browser, review the model's contents.



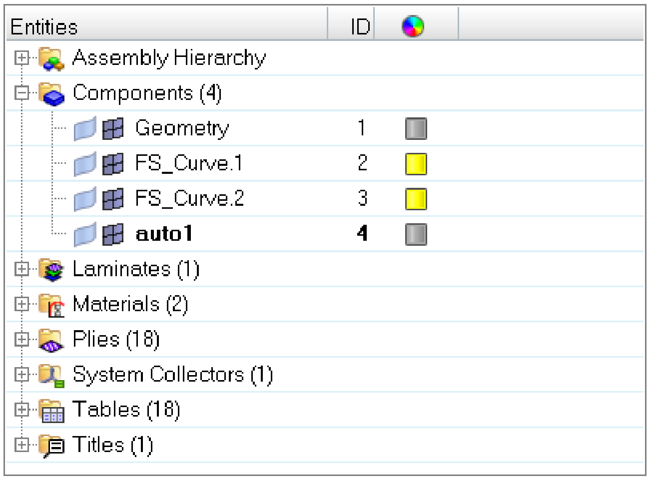
7. Make elements and feature lines transparent by clicking on the **Visualization** toolbar.
8. In the panel area, click ***comps***.
9. Select the components, ***Geometry*** and ***auto1***.
10. Click ***select***.
11. Use the **transparency** slider to review the system collector imported by the FiberSim model.



12. Click ***return***.
13. On the **Visualization** toolbar, click to shade the elements and mesh lines, and click to shade the geometry and surface edges.



14. In the **Model** browser, turn off the display of geometry for all of the components.

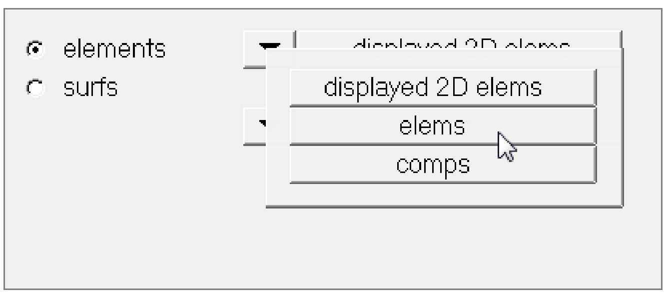


Step 5: Review and Edit Element Normals

1. Open the **Normals** panel by clicking ***Mesh*** > ***Check*** > ***Elements*** > ***Normals*** from the menu bar.

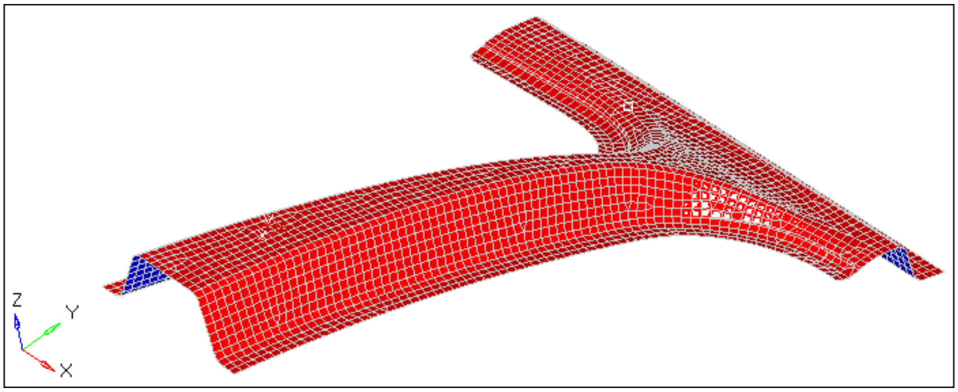
   Note: Element normals need to be changed to match the Z direction (red color).

2. Set the first switch to ***elems***.

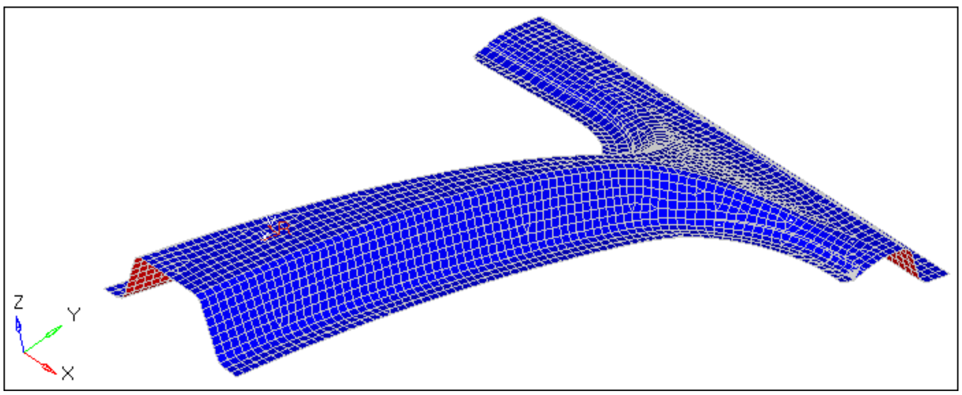


3. Click ***elems*** >> ***displayed***.
4. Set the **vector display**/**color display** toggle to ***color display***.
5. Click ***display***. HyperMesh displays, on each side of the part, the element normals using the colors red and blue.

   Note: The red side of the elements is the positive normal direction Z, while the blue side is the negative normal direction.



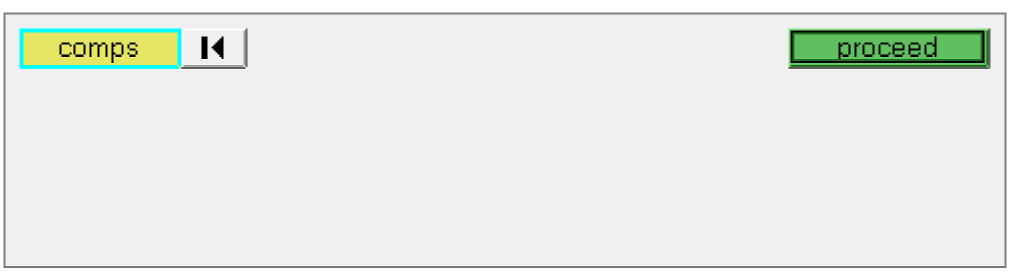
6. Set **orientation** to ***elem***.
7. Select one element that has the right direction. This element will be used as the model for all of the other elements.
8. Click ***adjust***. All of the elements are set in the same normal direction.
9. Optional. If the blue color is in the Z direction, click ***elems*** >> ***displayed*** and then click ***reverse***. All of the elements are set in the right normal direction (red).



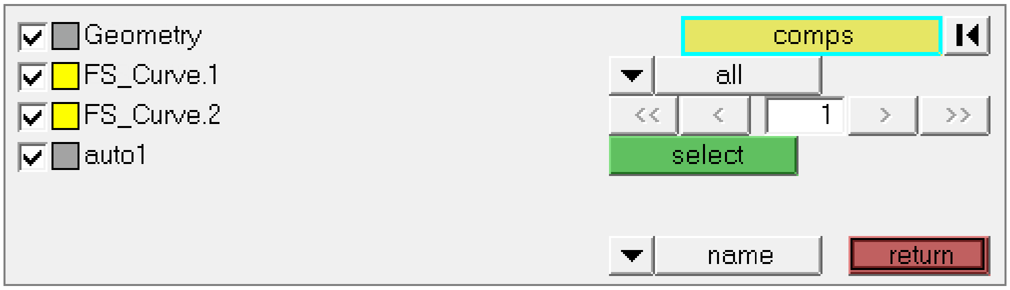
10. Click ***return*** to exit the panel.

Step 6: Realize Ply Geometry Shape

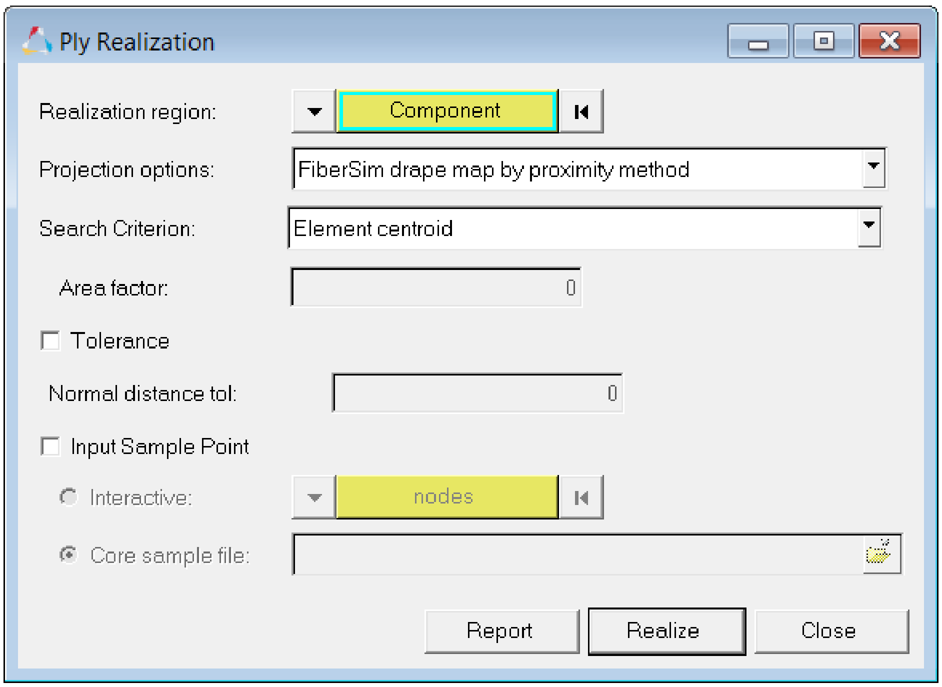
1. In the **Model** browser, right-click on the ***Plies*** folder and select ***Realize*** from the context menu.
2. In the **Ply Realization** dialog, click ***Component***.
3. In the panel area, click ***comps***.



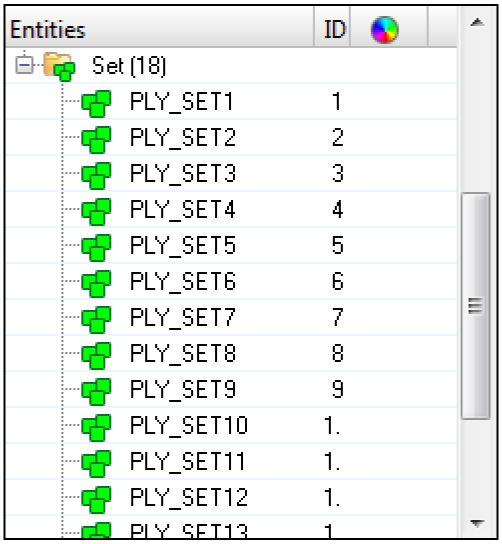
4. Select all of the components.



5. Click ***select***.
6. Click ***proceed***.
7. Set **Projection options** to ***FiberSim drape map by proximity method***.
8. Set **Search Criterion** to ***Element centroid***.



9. Click ***Realize***. This process takes each FiberSim Ply data and finds the FE elements which are bounded by the ply boundaries, and transfers the ply directions, draping data, and ply orientation into FE elements. This process also converts geometry plies into FE plies. At the end of realization, HyperMesh creates sets containing FE elements for each ply.

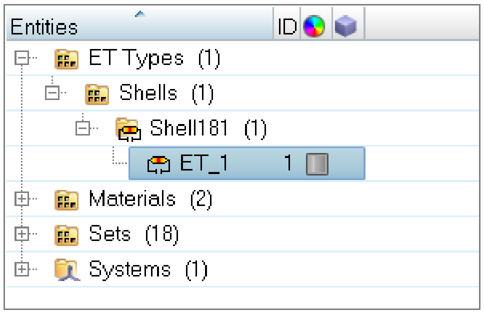


Step 7: Add an Element Type

1. In the **Model** browser, right-click and select ***Create*** > ***Sensor*** from the context menu. HyperMesh creates and opens a sensor (Et Type) in the **Entity Editor**.
2. In the **Entity Editor**, enter a name and ID, and select a color.
3. By default, **Element Type** is set to **SHELL181**.

   SHELL181

   - Suitable for analyzing thin to moderately-thick shell structures. It is a 4-node element with six degrees of freedom at each node: translations in the x, y, and z directions, and rotations about the x, y, and z-axes. (If the membrane option is used, the element has translational degrees of freedom only). The degenerate triangular option should only be used as filler elements in mesh generation.
   - Well-suited for linear, large rotation, and/or large strain nonlinear applications. Change in shell thickness is accounted for in a nonlinear analysis. In the element domain, both full and reduced integration schemes are supported. SHELL181 accounts for follower (load stiffness) effects of distributed pressures.
   - May be used for layered applications for modeling laminated composite shells or sandwich construction. The accuracy in modeling composite shells is governed by the first order shear deformation theory.
4. To simulate the element stiffness, set the stress stiffening option, extra displacement shapes, extra stress output, pressure loading, mass matrix, stress stiffness matrix, define the element coordinate system and specify the data storage using the respective keyopts, click ***Create/Edit***.
5. Open the **Solver** browser by clicking ***View*** > ***Browsers*** > ***HyperMesh*** > ***Solver*** from the menu bar.
6. Review the new element type you just created.



Step 8: Update the Component with Element Type

All elements in the model are in the Geometry component. In order for all elements in the Geometry component to be exported with the element type SHELL181, you must attach the element type defined in the previous step to the Geometry component. The desired element type is SHELL181, which is why you were instructed to create SHELL181 elements in the previous step. In this step, you will attach the sensor entity, with SHELL181 elements attached, to the Geometry component.

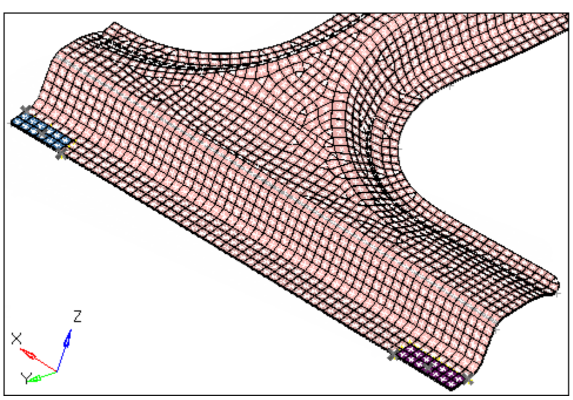
1. In the **Model** browser, **Component** folder, select ***Geometry***. The **Entity Editor** opens and displays the component's corresponding data.
2. In the **Entity Editor**, set **Card Image** to ***HM_COMP***.
3. For **Type**, click ***Unspecified*** >> ***Sensor***.
4. In the **Select Sensor** dialog, select ***sensor1*** and click ***OK***.

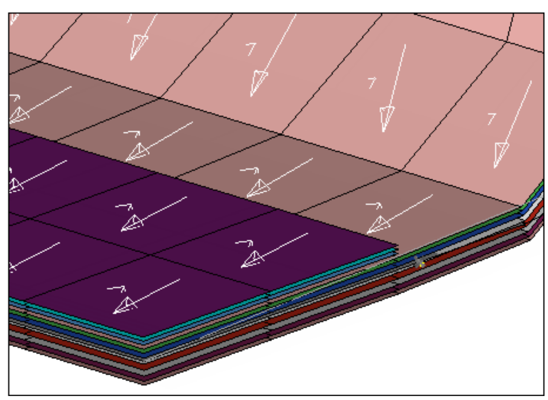
Step 9: Ply 2D Visualization

In this step you will verify FE plies thickness and orientation in HyperMesh.

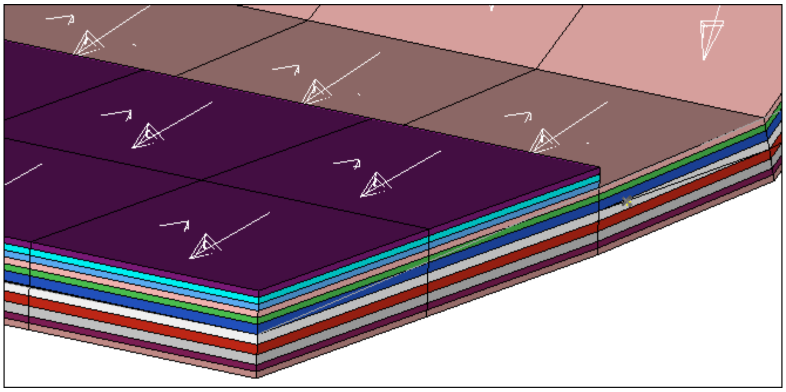
1. On the **Visualization** toolbar, set the element color mode to 

(visualize elements by property).
2. On the **Visualization** toolbar, set the layer representation mode to  (Composite Layers with Fiber Direction).
3. In the **Model** browser, ***Hide*** and ***Show*** each Ply.



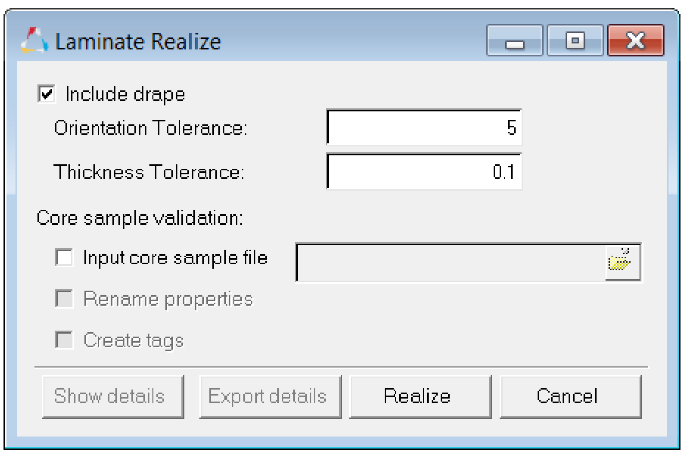


4. On the **Visualization** toolbar, set the element representation mode to  (2D Detailed Element Representation).



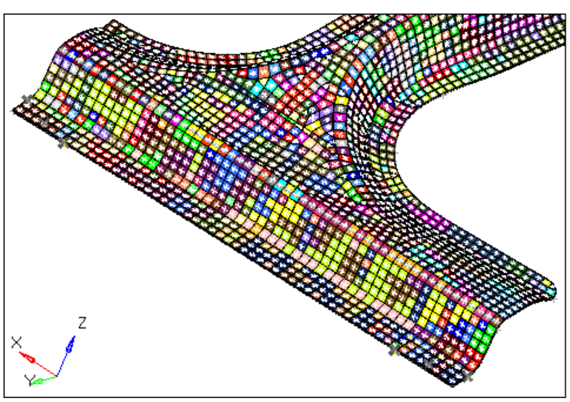
Step 10: Laminate Realize - Ply Based Model

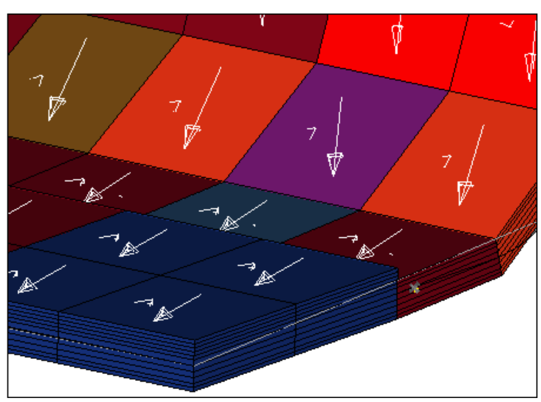
1. In the **Model** browser, right-click on the ***Laminate*** folder and select ***Realize*** from the context menu.
2. In the **Laminate Realize** dialog, accept the default settings and click ***Realize***. HyperMesh creates a property for each stack, and assigns it to a component.



3. On the **Visualization** toolbar, set the element color mode to 

.





4. In the **Model** browser, **Sections** folder, you will see the shell sections that were created when you realized the laminate.
5. Click on a section to display its corresponding details in the **Entity Editor**.
6. In the **Entity Editor**, under **PLIES**, next to **Data: TK**, click .
7. In the **PLIES** dialog, review the number of plies, thickness, orientation, and material data.
8. Click ***Close***.

Step 11: Export the Eeck to ANSYS `*.cdb` Format.

1. Open the **Export** tab by click ***File*** > ***Export*** > ***Solver Deck*** from the menu bar.
2. Set **File type** to **Ansys**.

   Note: If you are in the **ANSYS** user profile, HyperMesh automatically sets the **File type** to Ansys and loads ANSYS as the default **Template**.
3. In the **File** field, navigate to your working directory and save the file as `pillar_w_ncf-FINALhm.cdb`.
4. Click ***Export***.