



Altair

---

**HyperWorks**

## HM-4370: FiberSim Support in Abaqus Interface

In this tutorial, you will learn how to:

- Load the Abaqus user profile and CAD data
- Mesh all of the surfaces at once, specifying element sizes and 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 distribution table
- View Ply thickness visualization 3D representation

## Model Files

---

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

## Exercise

---

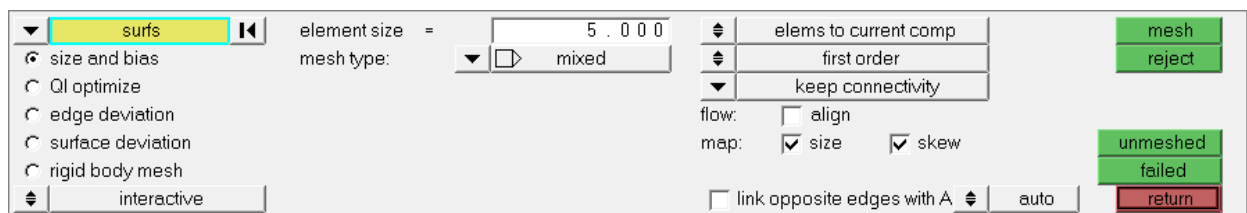
### Step 1: Load the Abaqus User Profile and Model

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 you load the user profile, HyperMesh displays the applicable utility menus, removes unused panels, disables unneeded entities in the Find, Mask, Card and Reorder panels, and makes specific adaptations related to the Abaqus solver.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **Abaqus, Standard3D**.
3. From the menu bar, click **File > Import > Geometry**. The **Import - Geometry** tab opens.
4. Set **File type** to **CATIA**.
5. In the **File** field, open the `B-Pillar.CATPart` file.
6. Click **Import**. HyperMesh imports geometry data only.  
**Note:** You will import the Ply and Composite data later.
7. Click **Close**.

## Step 2: Mesh All the Surfaces at Once, Specifying Element Sizes and Element Type

1. Open the **Mesh** panel by clicking **Mesh > Create > 2D AutoMesh** from the menu bar, or pressing **F12**.
2. Go to the **Size and Bias** subpanel.
3. Click **surfs >> displayed**.
4. In the **element size =** field, enter 5.
5. Set **mesh type** to **mixed**.
6. Set the mesh mode to **interactive** (it may currently be on **automatic**).
7. Set the **elements to surf comp/elements to current comp** toggle to **elems to current comp**.



Size and Bias subpanel: Steps 2.3 through 2.7

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

**Note:** You are now 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.
9. Accept the mesh as the final mesh by clicking **return**.
 

**Note:** At this point, you are done using the **Mesh** panel to mesh the part. The mesh quality is very good. However, you will remain in the meshing module to perform the next steps, which demonstrate how to use various subpanels to interactively control the creation of the mesh.
10. Click **return** to go back to the main menu.

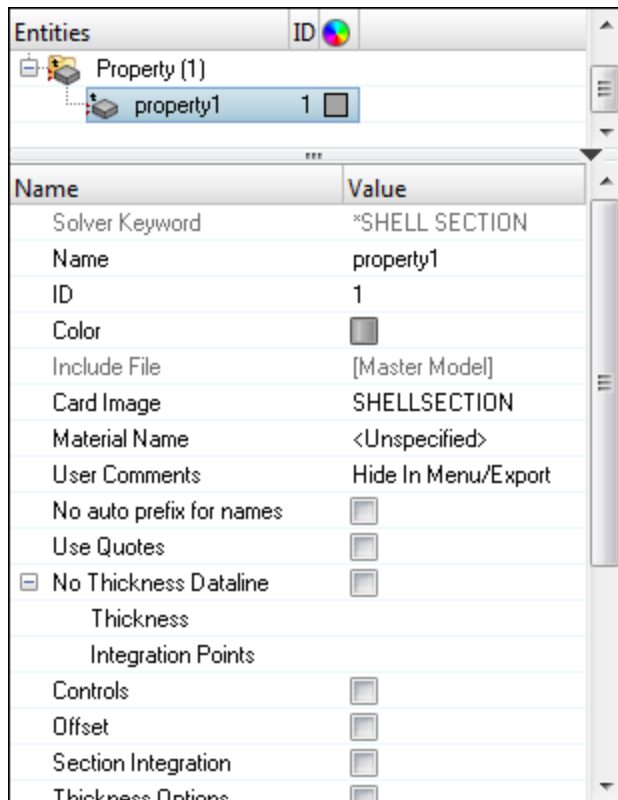
## Step 3: Load the Ply Information from FiberSim.

1. From the menu bar, click **File > Import > Geometry**. The **Import - Geometry** tab opens.
2. Set **File type** to **FiberSim**.
3. In the **File** field, open the `bpillar.h5` file.
4. Click **Import**. HyperMesh imports and populates the HyperMesh 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.

Entities	ID	
Assembly Hierarchy		
Component (36)		
Laminate (1)		
bpillar_laminate	1	
Material (2)		
DRY-5H-3K	1	
NCF-3T-6-in	2	
Ply (16)		
P001	1	
P001_2	2	
P002	3	
P002_2	4	
P003	5	
P003_2	6	
P004_0	7	
P004_1	8	
P006-A_0	9	
P006-A_1	10	
P006-B_0	11	
P006-B_1	12	
P006-C_0	13	
P006-C_1	14	
P006-D_0	15	
P006-D_1	16	
System Collector (1)		
<b>ROS001</b>	1	
Table (16)		
P001-Table	1	
P001_2-Table	2	
P002-Table	3	
P002_2-Table	4	
P003-Table	5	
P003_2-Table	6	
P004_0-Table	7	
P004_1-Table	8	
P006-A_0-Table	9	
P006-A_1-Table	10	
P006-B_0-Table	11	
P006-B_1-Table	12	
P006-C_0-Table	13	
P006-C_1-Table	14	
P006-D_0-Table	15	
P006-D_1-Table	16	
Title (1)		

## Step 4: Define the Dummy Properties and Assign them to the Mesh

1. In the **Model** browser, right-click and select **Create** > **Property** from the context menu. HyperMesh creates and opens a property in the **Entity Editor**.



2. For **Name**, enter `Dummy_Prop`.
3. Click the **Color** icon, and select a color to display the property.
4. Set **Card Image** to **SHELLSECTION\_COMPOSITE**.

## Step 5: Define Material Orientation

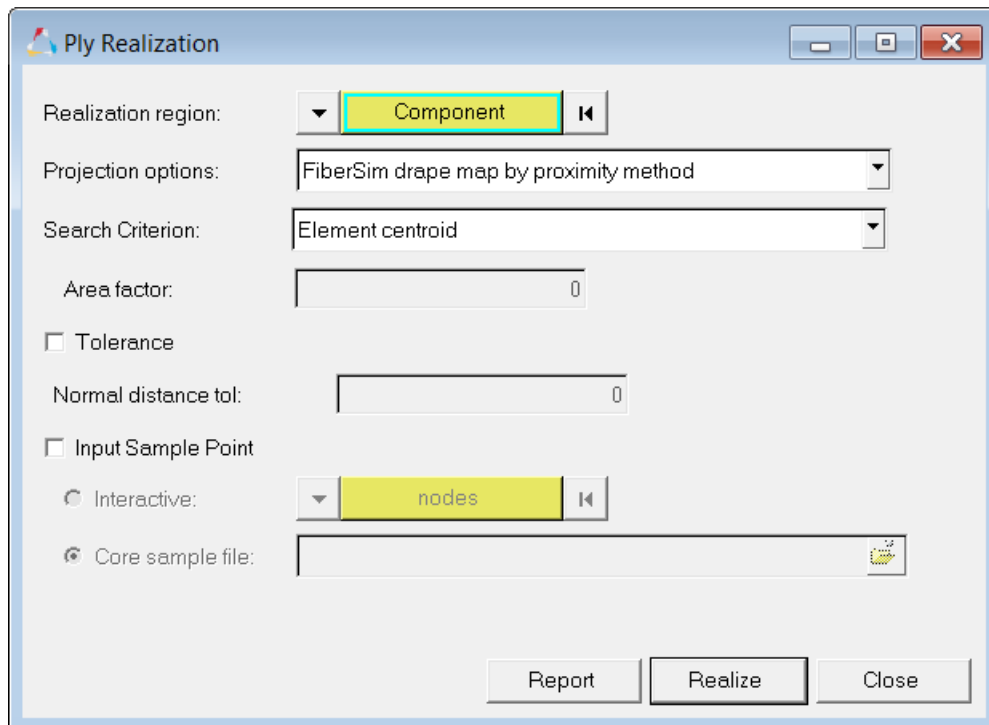
1. Open the **Composites** panel by clicking **composites** in the **2D** page.
2. Go to the **material orientation** subpanel.
3. Set the entity selector to **props**.
4. Click **props**.
5. Select the property, **Dummy\_prop**.
6. Click **select**.
7. Double-click **system**, and then enter the ID of the system which was created while importing the FiberSim model in the **id=** field.
8. Set the local axis switch to **local 1-axis**.

**Note:** By default, a local axis will automatically be selected when you enter a system ID.

9. Click **project**.

## Step 6: Ply Realization with FiberSim Drape

1. In the **Model** browser, right-click on the **Ply** folder and select **Realize** from the context menu.
2. In the **Ply Realization** dialog, set the **Realization region** selector to **Component**.
3. Click **Component**.
4. In the panel area, click **comps**.
5. Select all of the components.
6. Click **select**.
7. Click **proceed**.
8. Set **Projection options** to **FiberSim drape map by proximity method**.
9. Set **Search Criterion** to **Element centroid**.

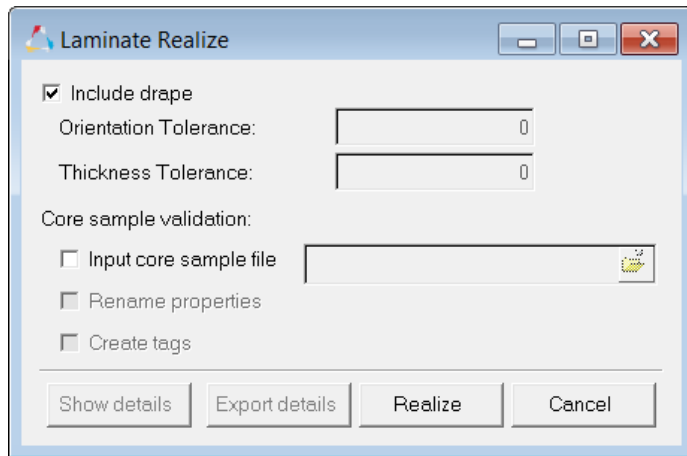


10. Click **Realize**. HyperMesh takes each FiberSim's ply data and finds the FE elements which are bounded by the ply boundaries, and then transfers the ply directions, draping data, and ply orientation into FE elements. Also, HyperMesh converts geometry plies into FE plies, and creates sets containing FE elements for each ply.

## Step 7: Laminate Realize

1. In the **Model** browser, right-click on the **Laminate** folder and select **Realize** from the context menu.

- 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. On export the dummy property is ignored.






## Step8: Create/Edit Distribution Table

The variation of element thickness, offset, and drape information is preserved in a distribution table.

- In the **Solver** browser, right-click and select **Create > Table > \*Distribution Drape** from the context menu.
- In the **Create Table** dialog, **Name** field, enter `DrapeTable`.
- Double-click **Element**.
- Select the elements for which you are creating a distribution table for.
- Click **proceed**.
- In the **Confirm** dialog, click **Yes** to append the selected elements to the table.
- In the **Thick** and **Angle** columns, enter the corresponding thicknesses and angles for each element.
- Click **Create**. HyperMesh adds the drape table to the **Solver** browser, **Table** folder.
 

**Note:** Drape data from the FiberSim model is collected in the **Table** folder.
- Optional: Access or edit a table entity by expanding the **Table** folder, right-clicking on the table entity, and selecting **Edit** from the context menu. The **Edit Table** dialog opens with all of the drape information.

## Step 9: Ply Thickness Visualization - 2D Representation

- On the **Visualization** toolbar:
  - Set the **Element Color Mode** to **By Prop** (  By Prop ).
  - Set the **Composite Visualization** to **Composite Layers** (  ).
  - Set the **Element Visualization** to **2D Detailed Element Representation** (  ).