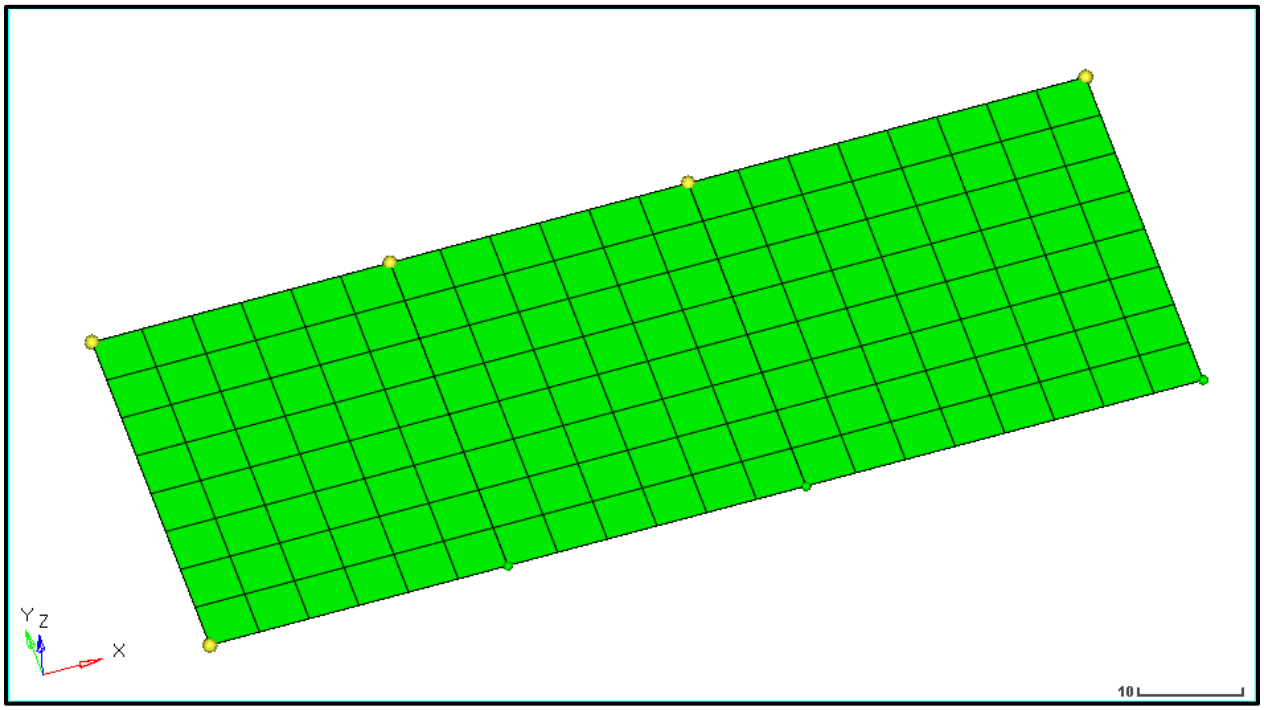
Exercise 3B: Creating Ply-based Laminates Using Geometry

This exercise introduces the user to creating a simple but complete and running model from geometric data and material properties.



Problem Setup

There are no files for this exercise.

Step 1: Open HyperMesh Desktop with the OptiStruct user profile

Step 2: Create nodes to mark the edges of the composite plate mesh geometry

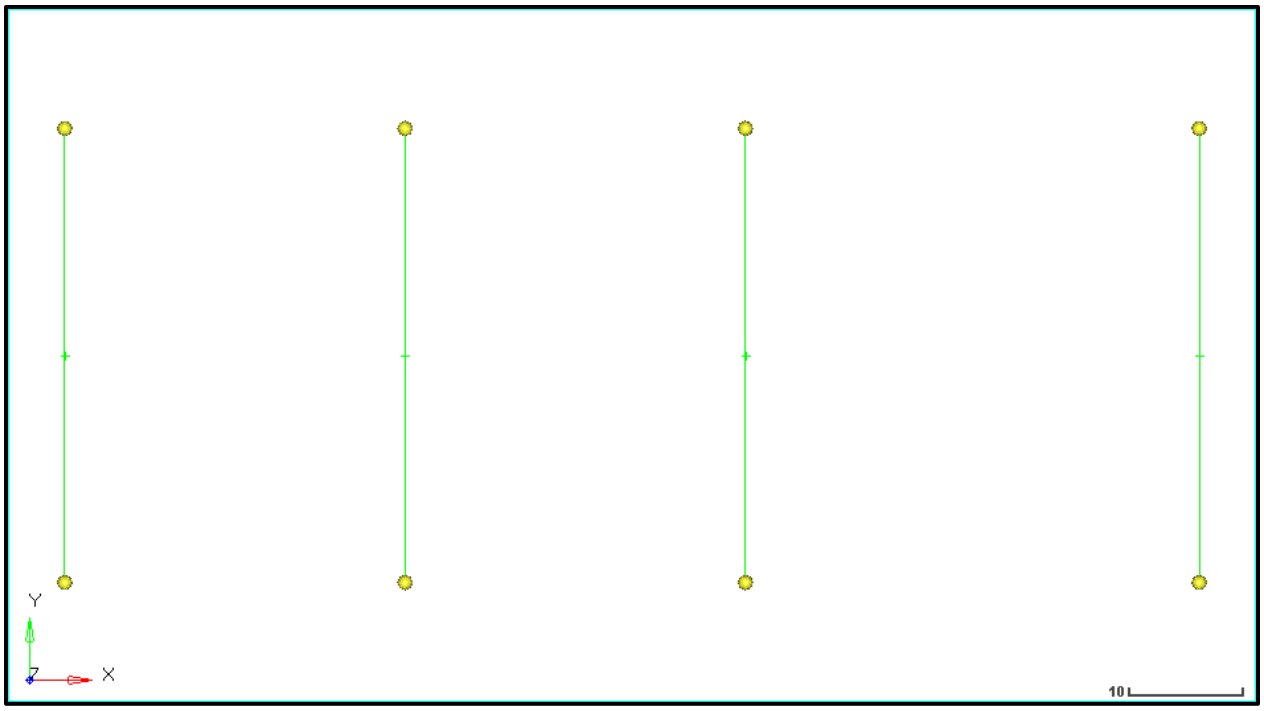
1. On the **Geom** page, enter the **nodes** panel.
2. With the ***XYZ***  option selected, enter the node coordinates `{-50,20,0}`. Click ***create*** to create the node.

x	-50.000
y	20.000
z	0.000
system	0

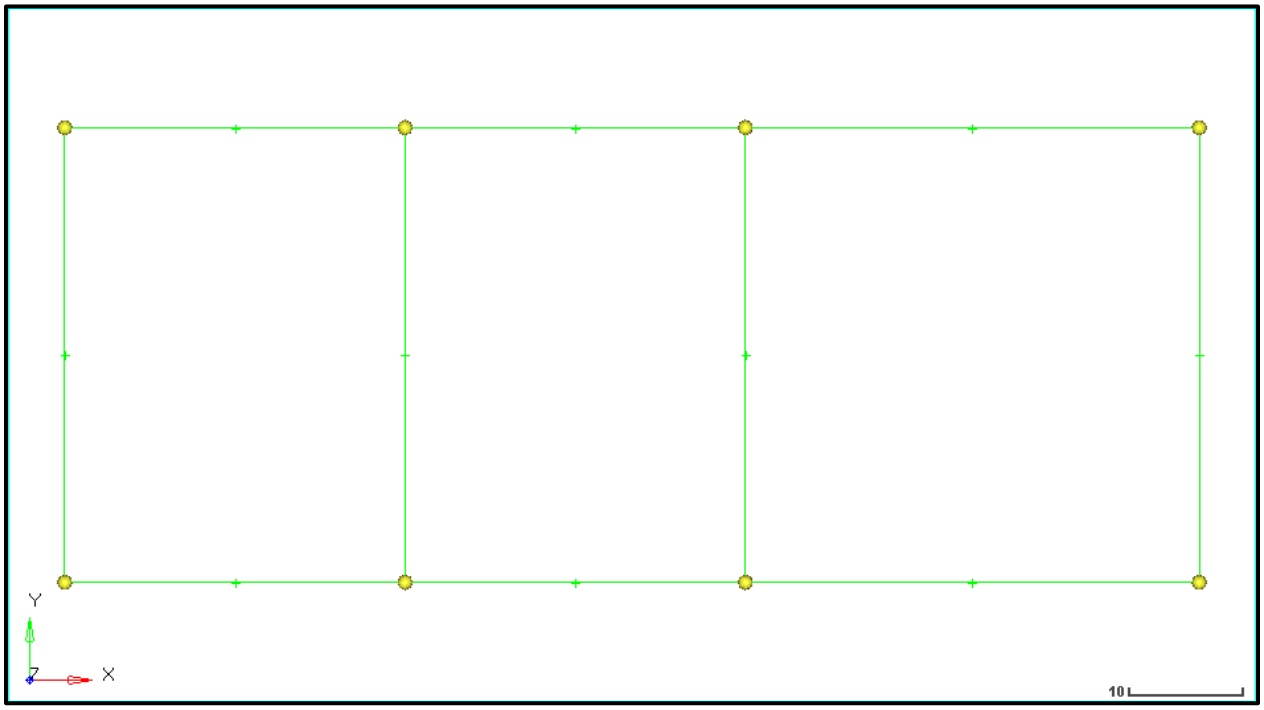
   Tip: Type “f” or use the fit to screen button  to show the newly created node in the graphics area.
3. Create seven more nodes at coordinates: `{-20,20,0}`, `{10,20,0}`, `{50,20,0}`, `{-20,-20,0}`, `{10,-20,0}`, `{50,-20,0}`, and `{-50,-20,0}`.

Step 3: Create lines to serve as ply boundaries

1. From the **Geom** page, select the ***lines*** panel.
2. With the ***Linear Nodes***  option selected and the ***node list*** entity selector active, select the top left and bottom left nodes in that order and click ***create*** to create a line between those two nodes.
3. Repeat step 2 to create 3 more vertical lines parallel to the first at each of the created node locations.



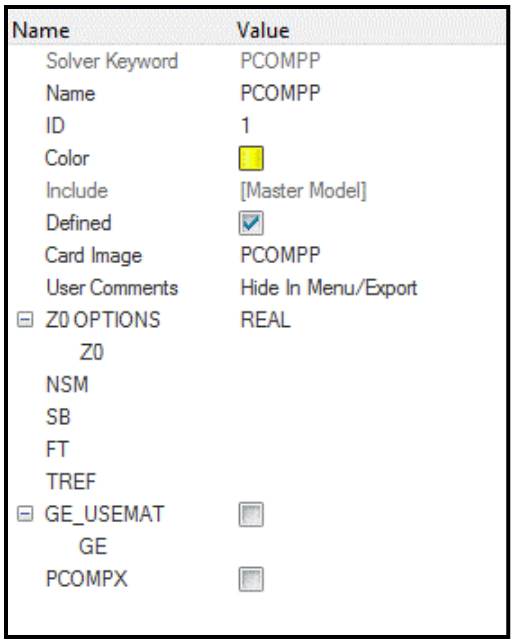
4. Similarly, create lines between each pair of adjacent nodes on the ‘top’ and ‘bottom’ of the rectangle to enclose each rectangle.



Step 4: Create a new MAT8 material with the following parameters

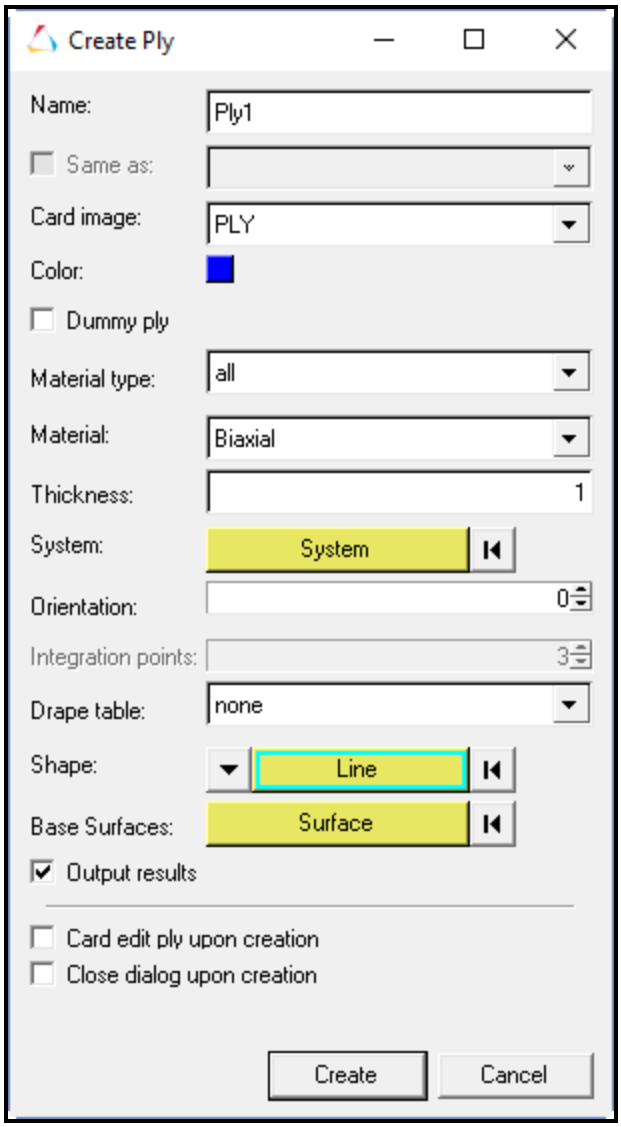
Name	Value
Solver Keyword	MAT8
Name	Biaxial
ID	1
Color	
Include	[Master Model]
Defined	☑
Card Image	MAT8
User Comments	Hide In Menu/Export
E1	75000.0
E2	6500.0
NU12	0.1
G12	2300.0
G1Z	2300.0
G2Z	2300.0
RHO	0.0015
A1	
A2	
TREF	
Xt	650.0
Xc	650.0
Yt	650.0
Yc	650.0
S	25.0
GE	
F12	
STRN	
MATT8	☐
MAT4	☐
MAT5	☐
MATF8	☐
MATX...	☐

Step 5: Create a new PCOMPP property with default parameters and assign to the component



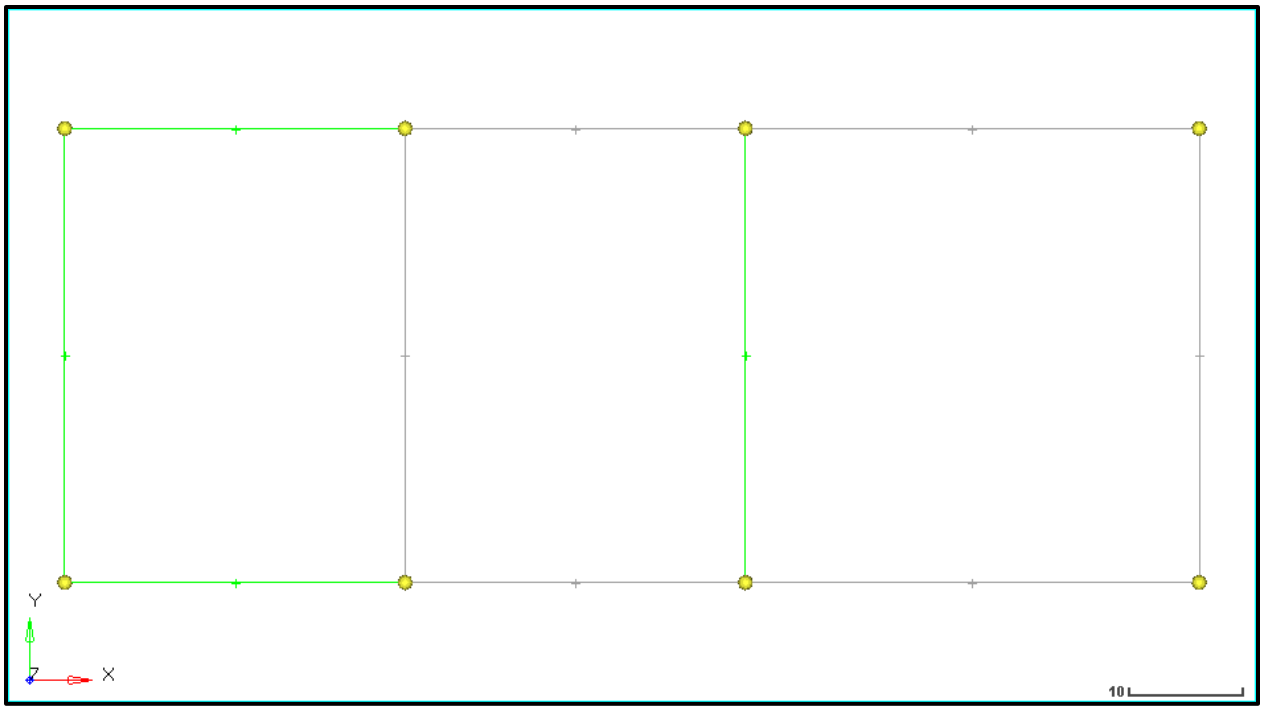
Step 6: Create the plies using the geometric lines as boundaries

1. In the **Model Browser**, right-click to select ***Create > Ply***.
2. For the first ply, set the ***Name:*** to `Ply1` with a ***Material*** of `Biaxial`, a ***Thickness*** of `1`, and an ***Orientation*** angle of `0` degrees. Change the ***Shape:*** drop-down entity selector to `Line` and use the line entity selector to select the outermost lines, signifying the ply is to represent the complete plate.

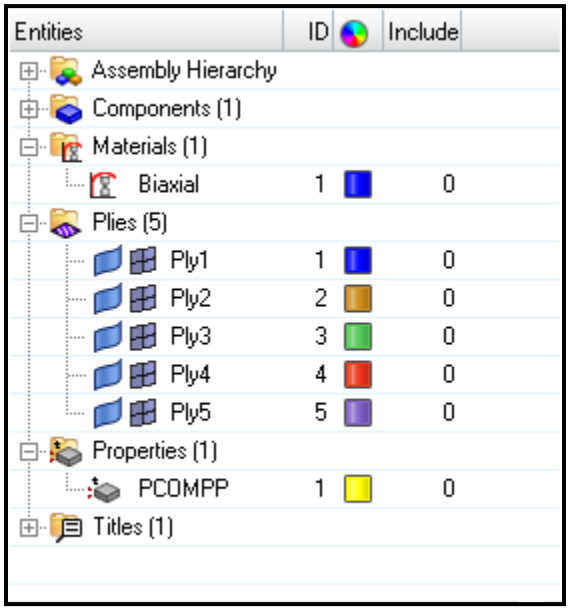


3. For the second ply, change the ply color and set the ***Name:*** to `Ply2` with a ***Thickness*** of `1`, an ***Orientation*** angle of `45` degrees, a ***Material*** of `Biaxial` and a shape using lines of the outline of the two rightmost sections.

   Tip: Do not select the dividing line between the sections.



4. For the third ply, change the ply color and set the ***Name:*** to `Ply3` with a ***Thickness*** of `1`, an ***Orientation*** angle of `90` degrees, a ***Material*** of `Biaxial` and shape of only the outline of the rightmost rectangle.
5. For the fourth ply, change the ply color and set the ***Name:*** to `Ply4` with a ***Thickness*** of `1`, an ***Orientation*** angle of `-45` degrees, a ***Material*** of `Biaxial` and shape of the outline of the two rightmost rectangles.
6. For the final ply, change the ply color and set the ***Name:*** to `Ply5` with a ***Thickness*** of `1`, an ***Orientation*** angle of `0` degrees, a ***Material*** of `Biaxial` and shape comprising the total plate.



Step 7: Create the laminate

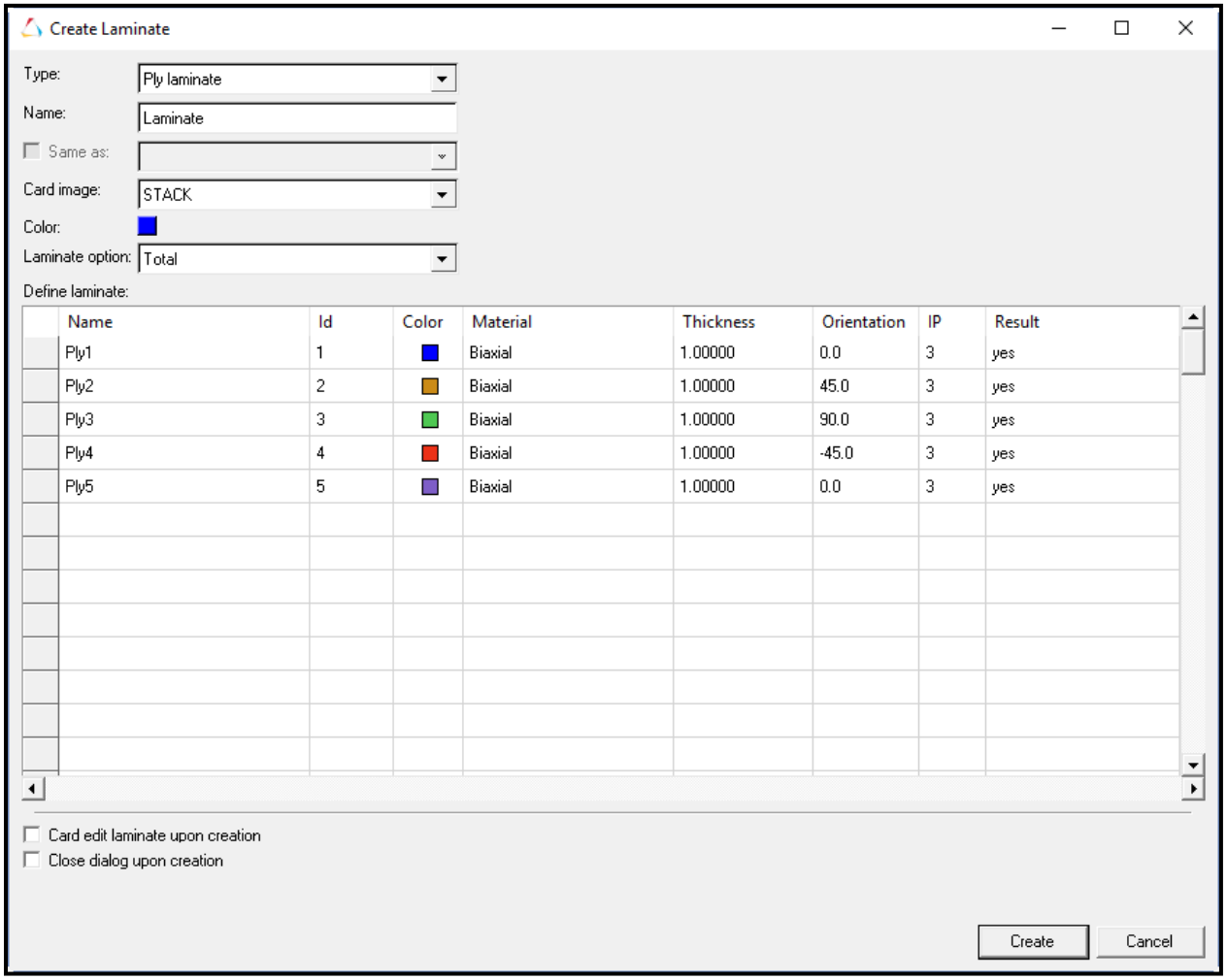
1. Right-click in the **Model Browser** and select ***Create > Laminate***. Name the new laminate `Laminate`.
2. Click in the first cell of the first row to select `Ply1` in the drop-down selector.

Tip: Information about the ply is populated on the rest of the row.

3. Repeat this process to create the other plies within the laminate in the same order they were created in HM.

Tip: Plies may not be used multiple times within a single laminate.

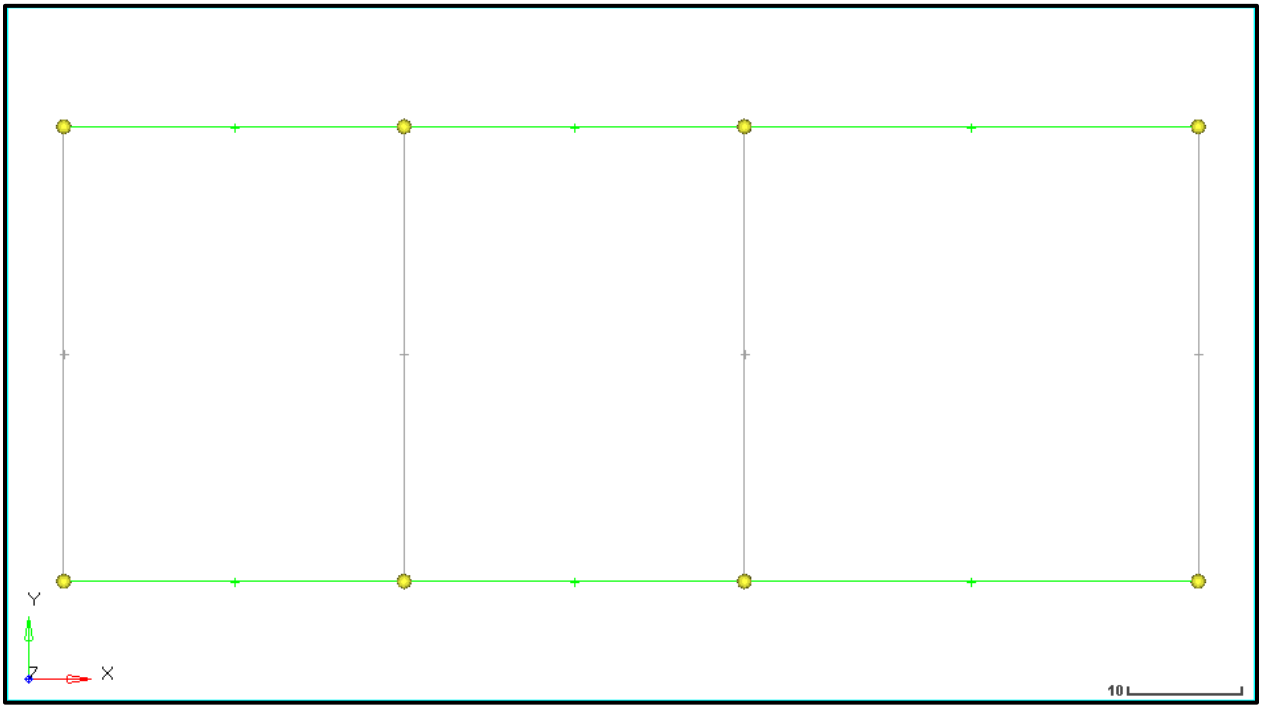
4. Click ***Create*** to create the laminate. Close the **Create Laminate** dialog box.



Step 8: Set composite visualization options for element color `by prop`, `2d detailed element representation`, and `composite layers`

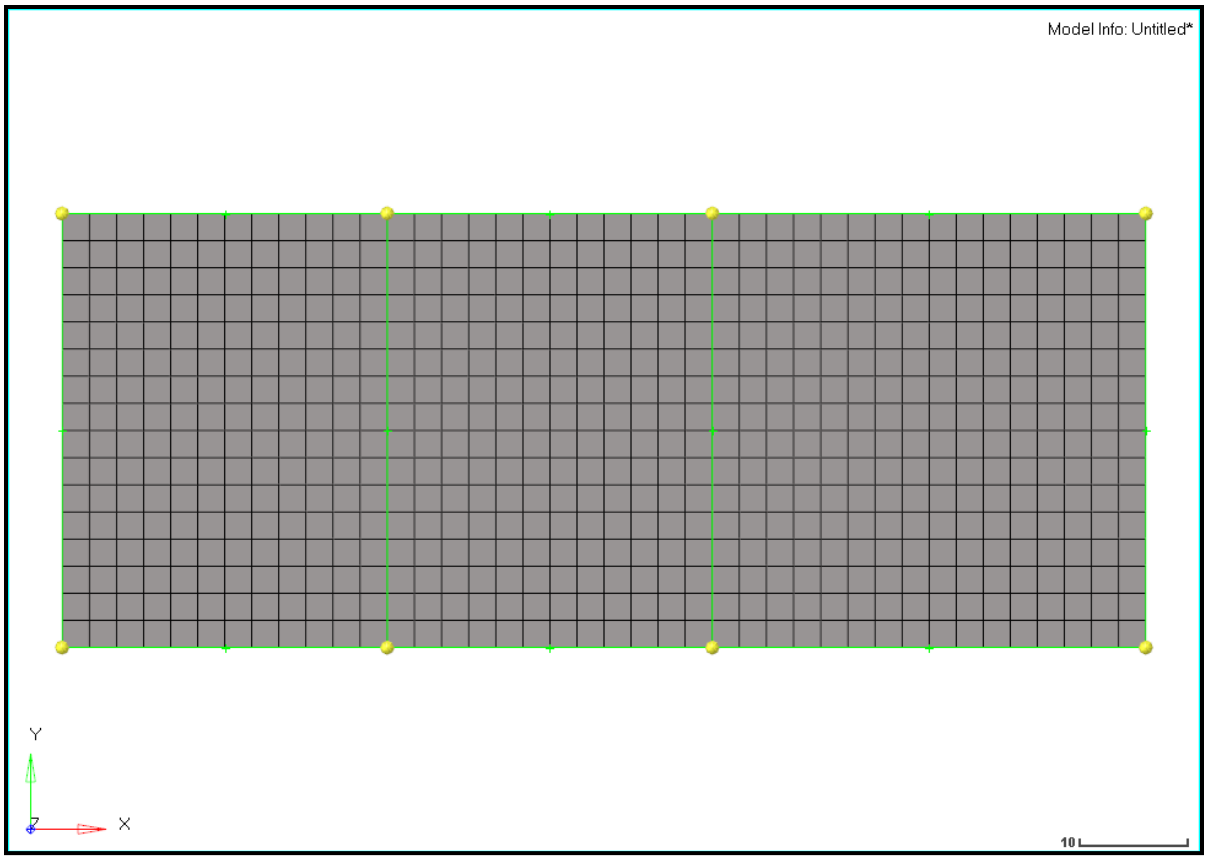
Step 9: Create the 2d mesh

1. On the **2D** page, select the ***skin*** panel. Set the ***mesh type*** to `mesh, w/o surf`.
2. With the ***line list*** entity selector active, select the four 'vertical' lines in the model from left to right.



3. Click ***create*** to accept the selection and go to the mesh parameters panel. Set the ***element size*** to `2.5`, click ***recalc all***, and click ***mesh***.

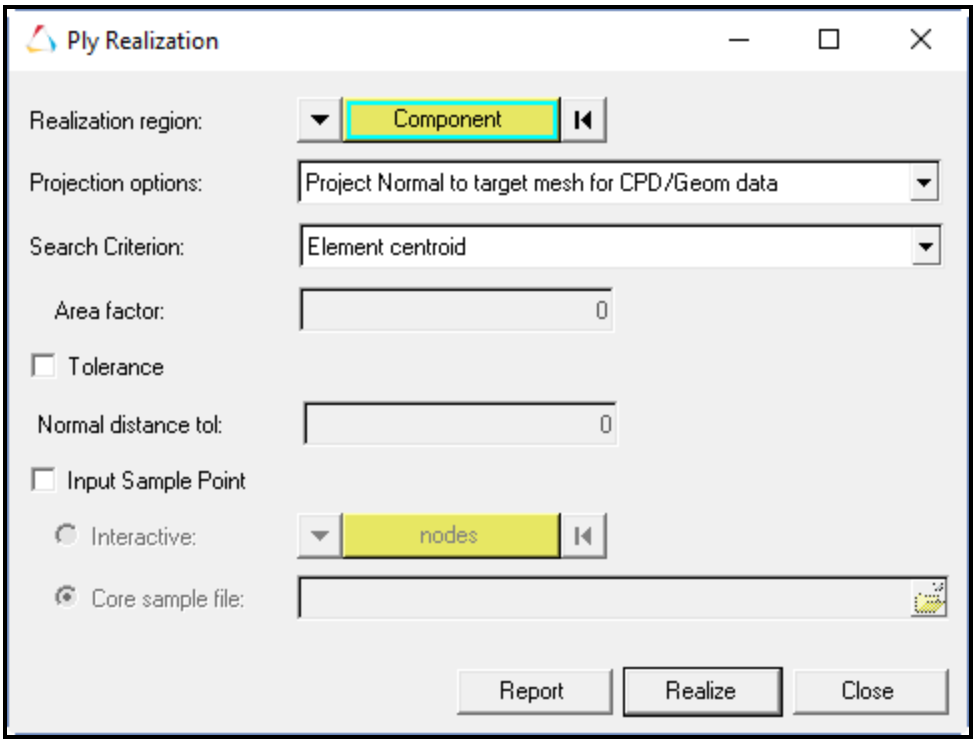
Tip: HyperMesh creates 640 elements from the line selections.



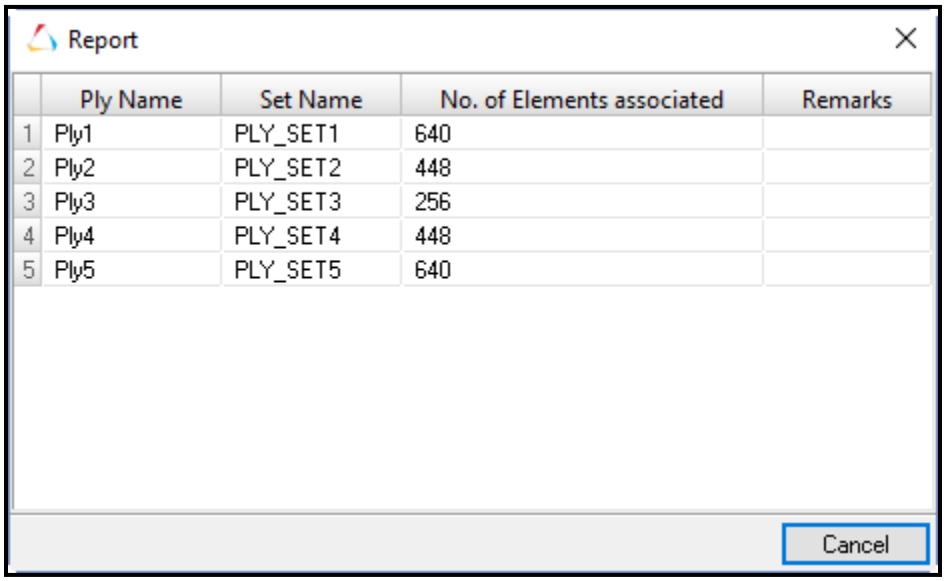
4. Click ***return*** twice to exit the skin meshing panel.

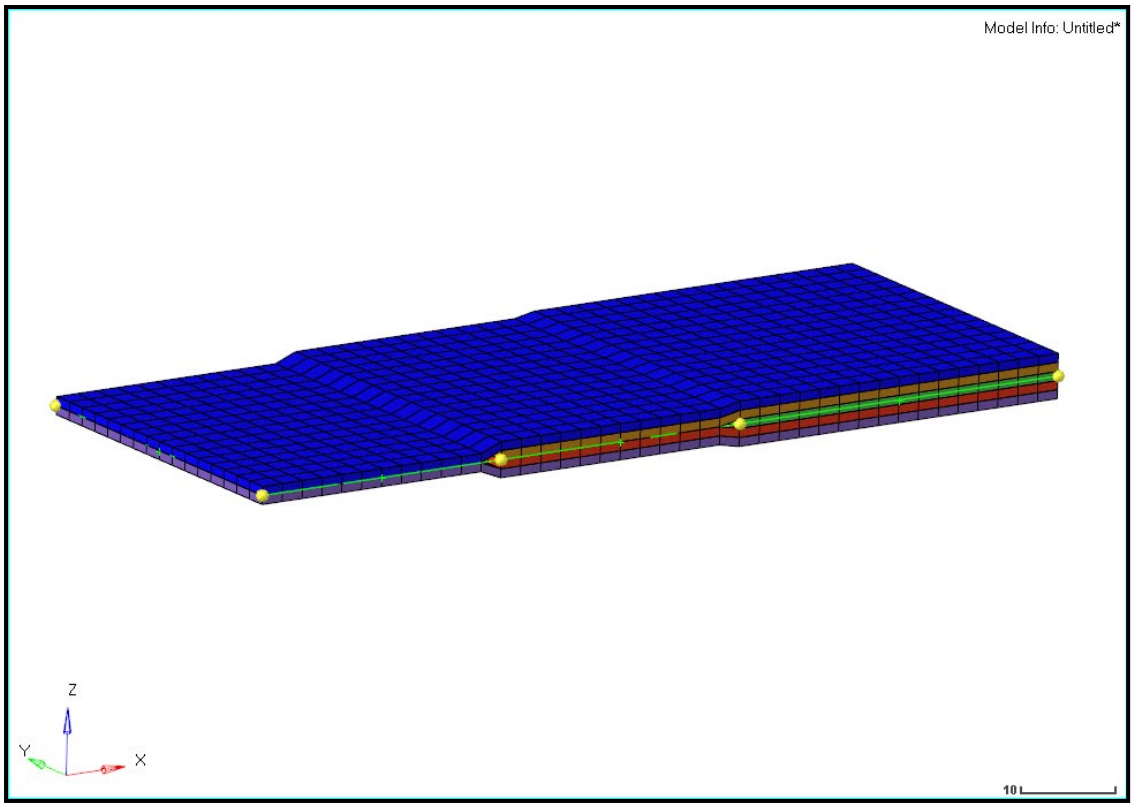
Step 10: Realize the plies to convert the geometric boundaries to element sets

1. In the **Model Browser**, right-click on the **Plies** container of the model tree and select ***Realize***.
2. Use the ***Component*** entity selector to select the `auto1` component as the ***Realization region:*** and set the ***Projection options:*** to `Project Normal to target mesh for CPD/Geom data`.



3. Click ***Realize*** to have **HyperMesh Desktop** project the line data onto the mesh to determine the mesh ply shapes.
4. Click on the ***Report*** button to review the plies and number of elements associated with those ply sets.

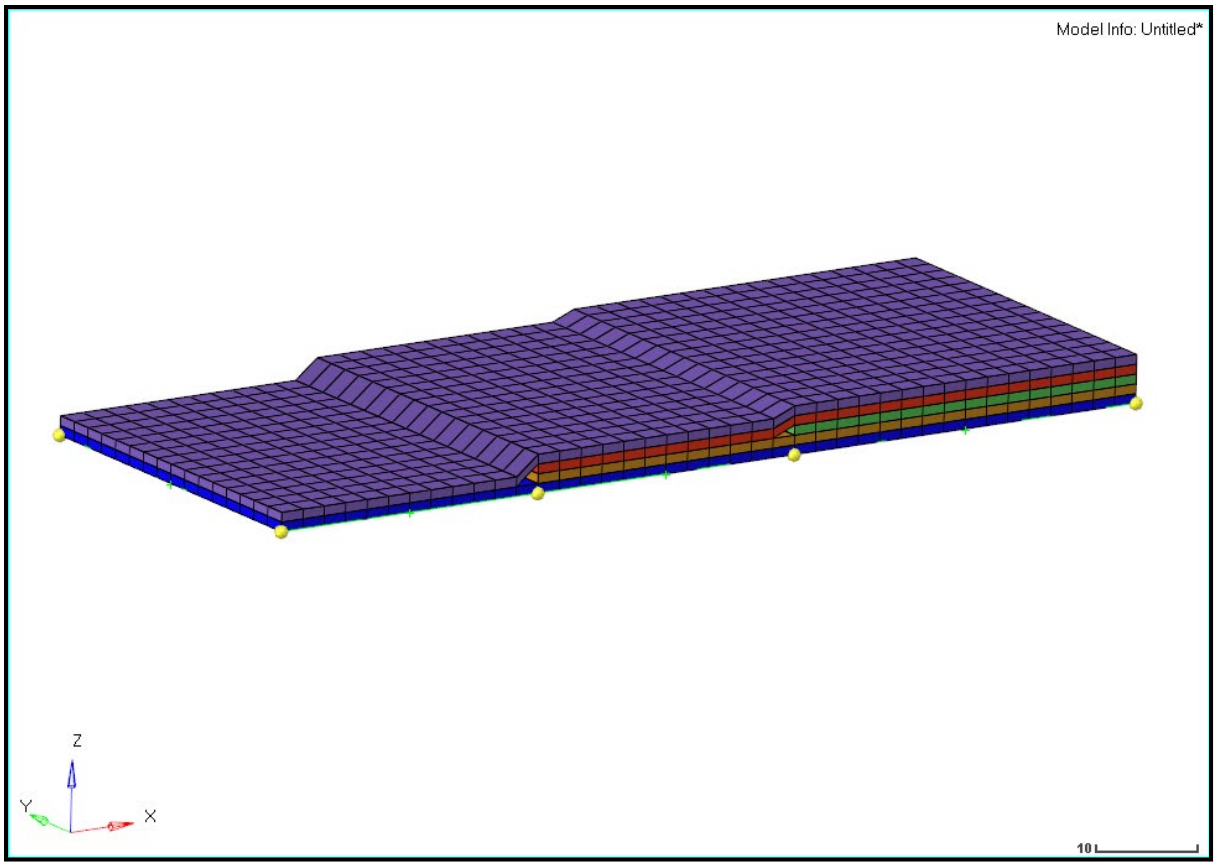




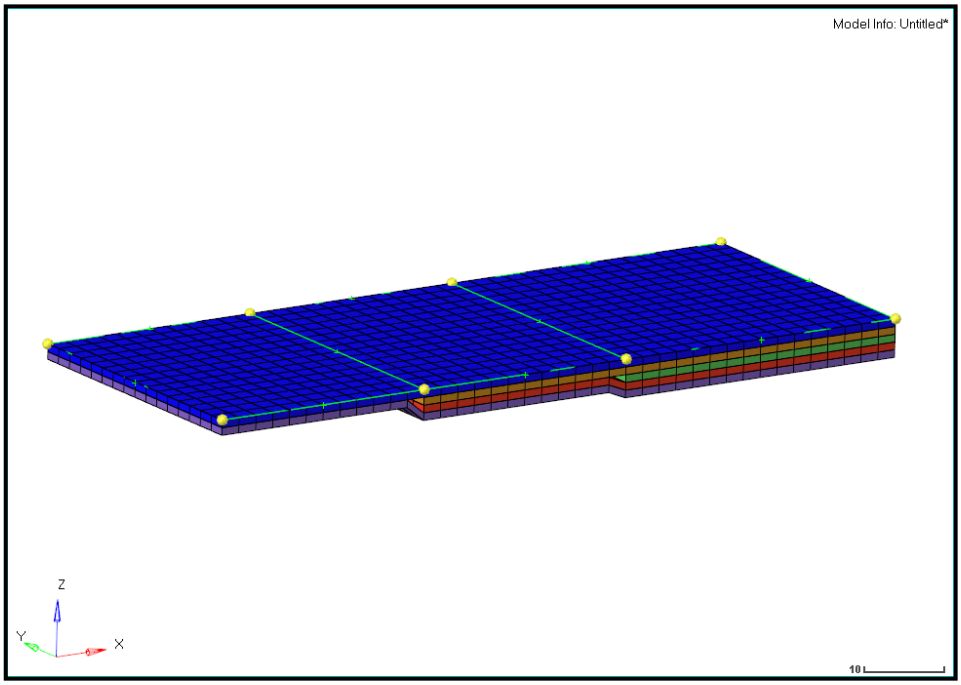
Step 10: Edit the PCOMPP card and set Z0 to 0 to set the bottom surface of the plate geometry coincident with the bottom surface of the base ply

Name	Value
Solver Keyword	PCOMPP
Name	PCOMPP
ID	1
Color	
Include	[Master Model]
Defined	☑
Card Image	PCOMPP
User Comments	Hide In Menu/Export
⊟ Z0 OPTIONS	REAL
Z0	0.0
NSM	
SB	
FT	
TREF	
⊟ GE_USEMAT	☐
GE	
PCOMPX	☐

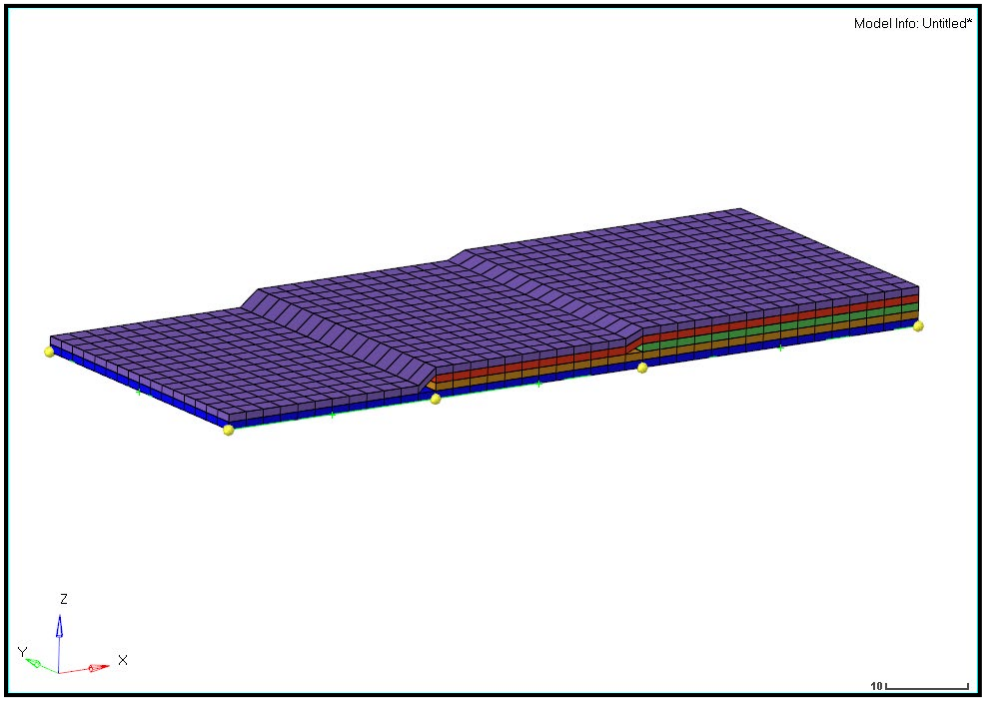
Tip: If your model does not look like the one shown below, continue to Step 11 to alter the model.



Step 11 [OPTIONAL]: If the model is shown in the graphics area with the laminate structure pushing the model thickness into the Z^- direction, use the menu to select *Mesh* > *Assign* > *Element Normals*.



Step 11 [CONTINUED]: Click *reverse* to invert all displayed element normals and make the laminate thickness rise into the Z^+ direction.



Step 12: Save the model as `plate.hm` and exit HyperMesh Desktop

EXERCISE RESULTS: `plate.hm`