Exercise 10a - Creation of an Aircraft Door Surround Composite Skin Model for OptiStruct and NASTRAN

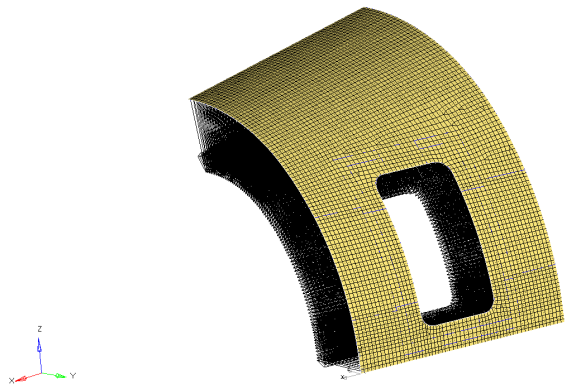
Step 1: Start HyperMesh in the OptiStruct user profile and open the HyperMesh model file

1. Start HyperMesh
2. Select the OptiStruct user profile
3. Open the HyperMesh model file, `10a_Composite_Skin.hm`

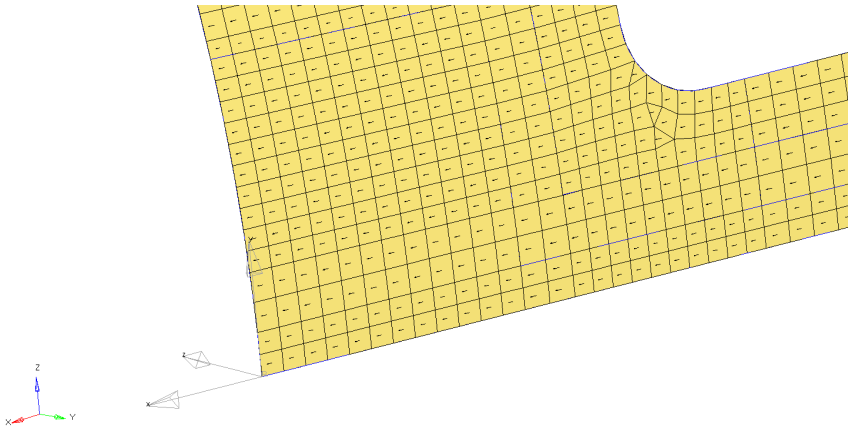
   This model contains a mesh of the aircraft door surround composite skin within the component Skin. A .CATPart file containing CPD data for 140 plies which make up the composite skin of the door surround was also previously imported. This import process produced 140 plies with related ply data and ply shape (contained within the component Ply_shapes), 1 laminate containing the stacking sequence, 1 material reference, and 1 coordinate system defining the ply fiber direction.

Step 2: Assign element normals and element material orientations

1. Review the current element normals. From the menu bar select ***Mesh > Assign > Element Normals.***
2. With the component collector active, select the **Skin** component by clicking any element of the skin component in the graphics window, and then click ***display normals.***
3. Notice that the element normals are pointing “outward” from the fuselage skin. Typically we desire the element normals to point “inward”. Since the staking direction of the plies is in the direction of the element normals, and if this points “inward”, then we can define the laminate in such a way that the outer mould line (OML) of the fuselage skin remains “smooth” and that all ply drops will occur in the inner mould line (IML).
4. Reverse the element normals so that element normals point “inward”. With the component collector active, select the **Skin** component again to ensure it is selected, and then click ***reverse normals***. Now the element normals are pointing “inward”.
5. Click ***return*** to exit the panel.

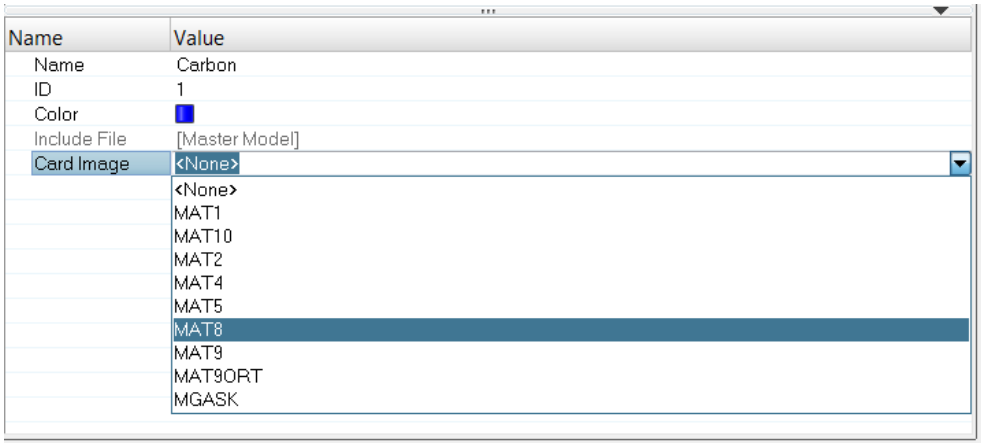


6. Review the current element material orientations. From the menu bar select ***Mesh > Assign > Element Material Orientation***.
7. With the element collector active, select ***elems >> all***, and then select ***review***.
8. Zoom in on the mesh and notice that the element material orientations are in all different directions. (You may need to change the element view to wireframe to view the orientations). The element material orientation defines the 0° direction for a ply. In this case, we desire the 0° direction for a ply to be along the length of the fuselage. Therefore, we need to reassign the element material orientations to be consistent and in this direction.
9. With the element collector active, select ***elems >> all*** again.
10. Select the ***by system axis*** material orientation method.
11. With the system collector active, select ***system 1*** located at the lower left hand corner of the door surround skin.
12. Select the ***local 1-axis*** option, and then click ***project***.
13. Zoom in on the mesh and notice that all element material orientations are now defined along the length of the fuselage as we desired, thus defining the 0° ply direction.
14. Click ***return*** to exit the panel.



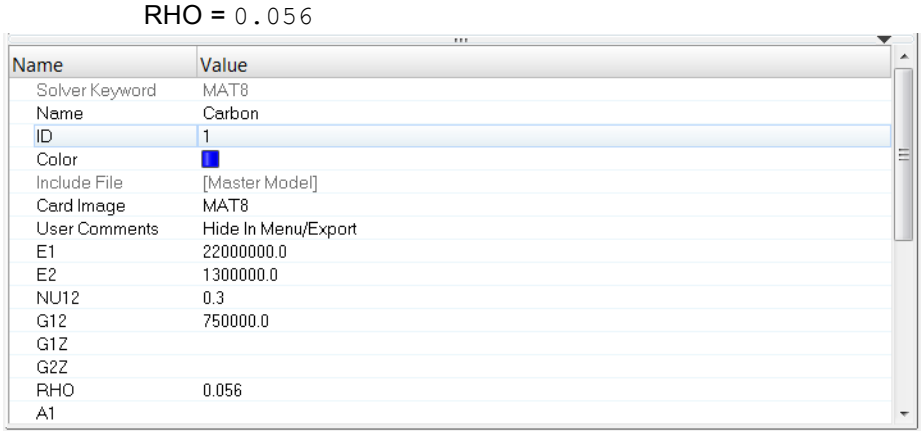
Step 3: Edit Carbon material and create a new FiberGlass material

1. From the **Model Browser**; expand the ***Material*** folder and select ***Carbon***.
2. Click on ***Carbon*** and update the **Card image** to ***MAT8*** in the entity editor.



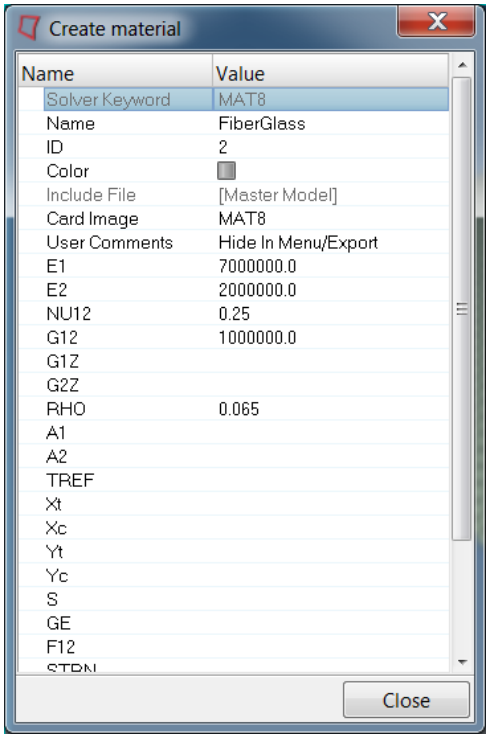
3. Next we will define orthotropic material properties for the Carbon material. Enter the following orthotropic properties for the Carbon material in the entity editor.

   E1 = `2.2e7`
   E2 = `1.3e6`
   NU12 = `0.3`
   G12 = `7.5e5`
   RHO = `0.056`



4. Create a new FiberGlass material with MAT8 orthotropic material properties by right clicking in the menu bar and selecting ***Materials > Create***.
5. On the **Create Material dialog**, enter the **Name** as `FiberGlass`, set the **Card image** as ***MAT8***, then enter the following orthotropic properties for the **FiberGlass** material and then click ***close***

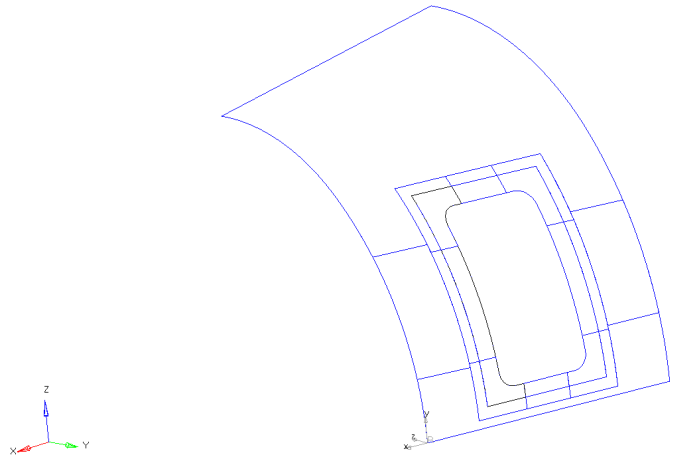
   E1 = `7e6`
   E2 = `2e6`
   NU12 = `0.25`
   G12 = `1e6`
   RHO = `0.065`



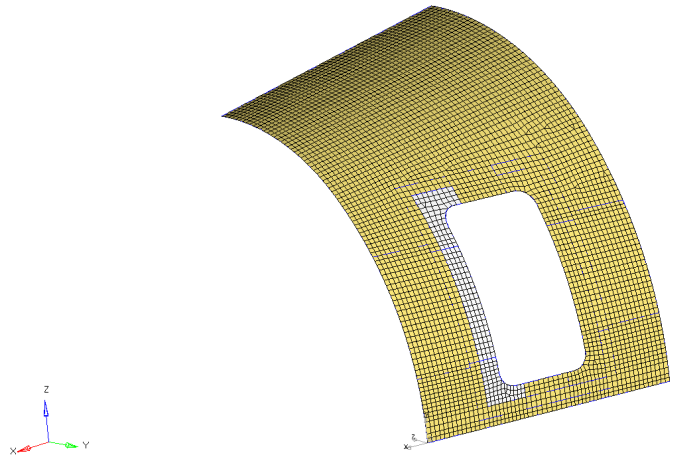
Step 4: Realize geometric ply shapes to FE ply shapes

On import of ply information from a CAD file, the ply shape definitions are defined with geometric lines. In order to create an FE model suitable for analysis, we must convert the geometry ply shapes to FE ply shapes. This process is called ply realization. First let's review the geometry ply shape definition for a ply.

1. First we will review the geometric ply shapes. Using the **Model Browser**, turn off the ***FE display*** of the component ***Skin*** and turn on the ***Geometry display*** of the component ***Ply_shapes***.
2. Within the **Model Browser**, expand the ***Ply folder*** and select ***Plies Group.1_PLY-30***.
3. Right click on the selection and select ***Edit***. This will bring up the **Edit Ply** dialog and highlight the geometric shape (lines) of the ply in the graphics area for review.
4. Select ***Cancel*** to close the dialog.



5. Next we will realize the geometric ply shapes to FE ply shapes. Using the **Model Browser**, turn on the ***FE display*** of the component ***Skin***.
6. From the **Model Browser**; select the ***Ply*** folder.
7. Selecting any folder implicitly selects all entities within that folder. Right click on the ***Ply*** folder and select ***Realize***.
8. Click on ***Component*** and then select the **Skin** component by clicking any element of the skin component in the graphics window. Select ***proceed***.
9. Select the ***Project Normal to target mesh for CPD/Geom data*** as the projection option, and then click ***Realize***.
10. All 140 plies will have their geometric shape definition converted to an equivalent FE shape definition by identifying all elements within the boundaries of the closed loop lines which define the geometric shape of each ply.
11. Next we will review the FE ply shapes. From the **Model Browser**, select ***Plies Group.1_PLY-30***. Right click on the selection and select ***Edit***. This will bring up the **Ply edit** dialog and highlight the FE shape (elements) of the ply in the graphics area for review.
12. Select ***Cancel*** to close the dialog.



Step 5: Create two OML FiberGlass protection plies

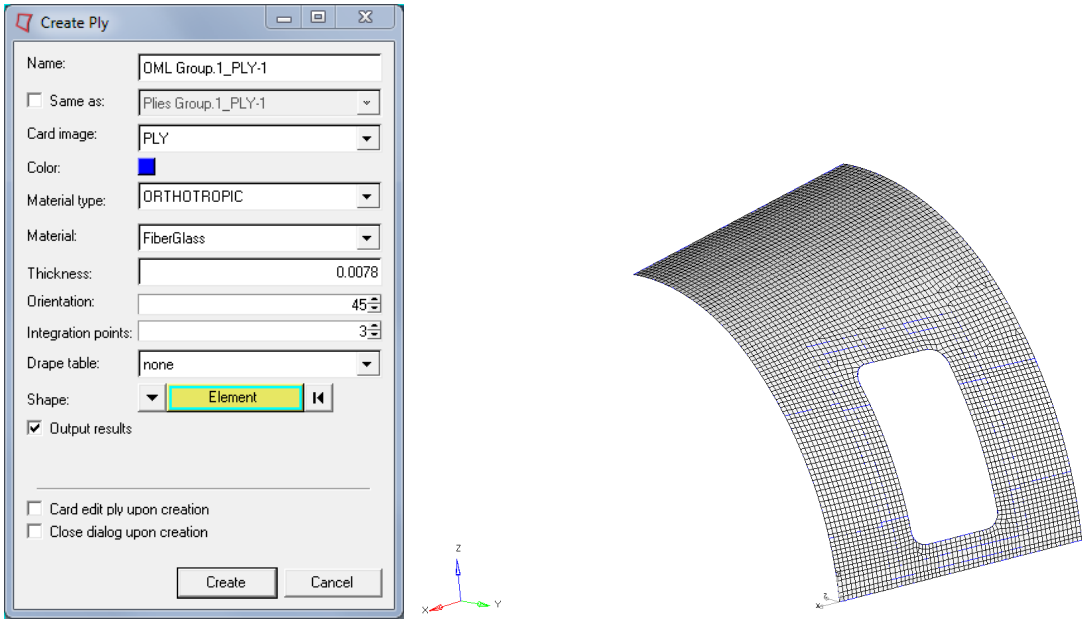
13. First we will create a 45° FiberGlass ply. From the menu bar select ***Properties > Create > Plies***.

14. On the **Create ply dialog**, enter the following ply data:

    Name: `OML Group.1_PLY-1`
    Material type: ***ORTHOTROPIC***
    Material: ***FiberGlass***
    Thickness: `0.0078`
    Orientation: `45`

15. Define the FE **Shape** of the 45° FiberGlass ply by clicking on ***Element*** and then selecting ***elems>>all***. Then select ***proceed***.

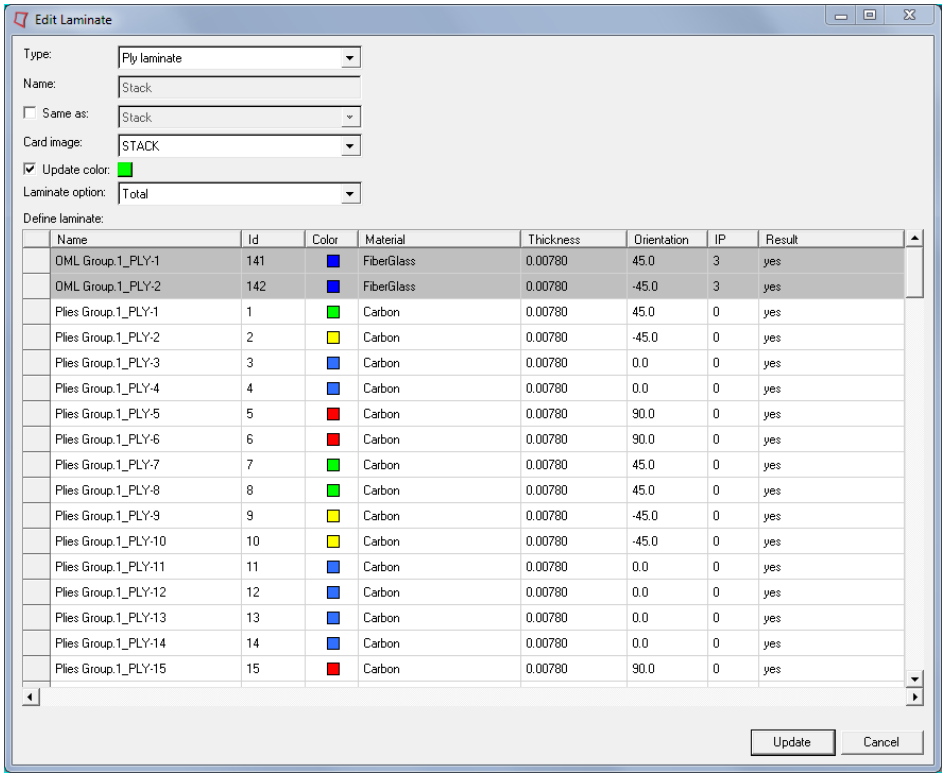
16. Click ***Create*** to create the ply.



17. Now create a -45° FiberGlass Ply. Repeat steps 5.1-5.4 above with **Name** as `OML Group.1_PLY-2` and an orientation of ***-45°***. Close out of the **Create Ply** dialog when complete.

Step 6: Edit the laminate stacking sequence definition to insert the OML FiberGlass protection plies to the OML.

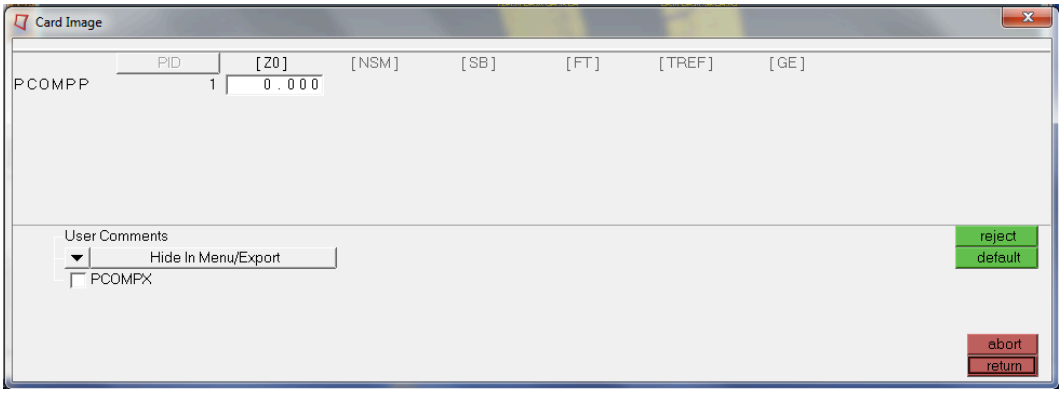
1. Review the current laminate stacking sequence. From the **Model Browser**, expand the ***Laminate*** folder and select ***Stack***.
2. Right click on the selection and select ***Edit***. This will bring up the **Edit Laminate** dialog showing the current stacking sequence of plies for the door surround skin.
3. For the **Card Image**, select ***STACK***.
4. For the **Laminate** option, select ***Total***.
5. Edit the laminate stacking sequence by inserting the OML FiberGlass protection plies to the "top" of the stack so that they are on the OML of the door surround skin. Click on the **1st column** of the **1st row** to highlight the first row.
6. Right click on the selection and select ***Insert***. This will insert a blank row at the "top" of the stack.
7. Do these steps again to insert a second black row at the "top" of the stack.
8. Click in the ***Name column*** (2nd column) of the **first row** and select ***OML Group.1_PLY-1*** to insert this ply into the stack at this location.
9. Click in the ***Name column*** (2nd column) of the **second row** and select ***OML Group.1_PLY-2*** to insert this ply into the stack at this location. The final stacking sequence is shown below.
10. Click ***Update*** to commit the changes to the HyperMesh database and exit the **Edit Laminate** dialog.



Step 7: Create and Assign a ply-based template property to all elements

1. Next we will create a ply-based template property (PCOMPP) with Z0 = 0.0. From the menu bar select ***Properties > Create > Properties***.
2. Define the **Name** as `template_property`.
3. For the **Card image** select ***PCOMPP***.
4. Make sure to check ***Card edit property upon creation*** and then click ***Create***.
5. On the **PCOMPP card image**, click on ***Z0*** and enter `0.0`.

   This defines the node plane of the mesh as the OML surface (i.e. bottom of the laminate) and since the element normals are pointing “inward” and the laminate plies stack in the direction of the laminate normal, all ply drops will be toward the IML surface and the OML surface will remain “smooth”.

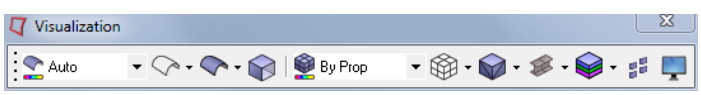


6. Select ***return***.
7. Now we will assign the ply-based template property to all the elements. From the **Model Browser**, expand the **Property** folder and select ***template_property***.
8. Right click on the selection and select ***Assign***.
9. This will bring up an elements collector panel. With the elements collector active, select ***elems>>all*** and then click ***proceed***.

   This assigns all elements the template_property and alerts OptiStruct to calculate each elements laminate from the PLY and STACK card image information defined within the model.

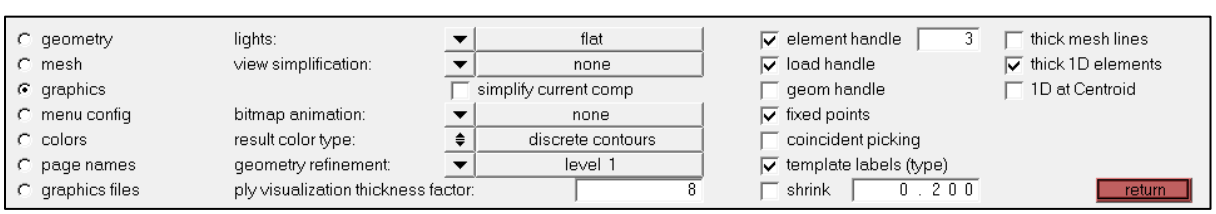
Step 8: Review the composite skin definition using 3D Representation with Composite Layers visualization

1. Set the visualization settings to review the model. From the **Visualization** toolbar select the ***By Prop*** color mode, set ***Shaded Elements and Feature Lines***, set ***3D Element Representation***, and set ***Composite Layers*** as shown below.

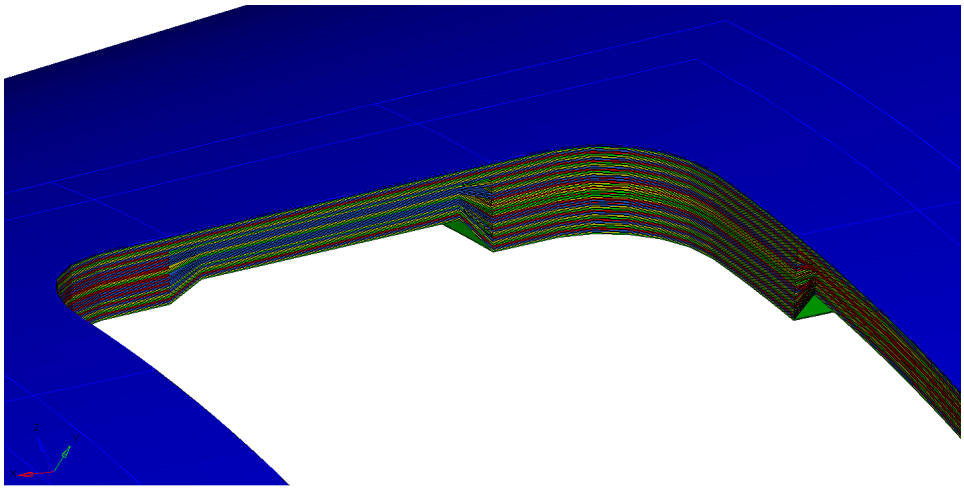


2. Now we will set the ply visualization thickness factor. From the menu bar select ***Preference > Graphics***.
3. Set the ***ply visualization thickness factor*** to 8.

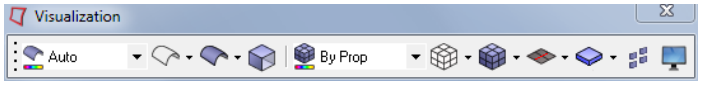
   This will artificially increase the thickness of the plies 8x to allow for visualization of the composite skin definition while also maintaining visualization capabilities of the extents of the part.



4. Review the composite skin definition by translating, rotating, and zooming the part as required/necessary.

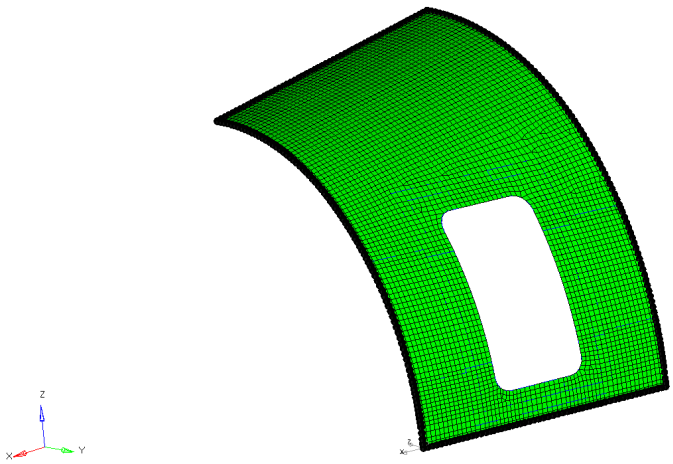


5. Set the visualization settings to work on the model again. From the **Visualization** toolbar, set ***Shaded Elements with Mesh Lines***, set ***Traditional Element Representation***, and set ***Layers Off*** as shown below.

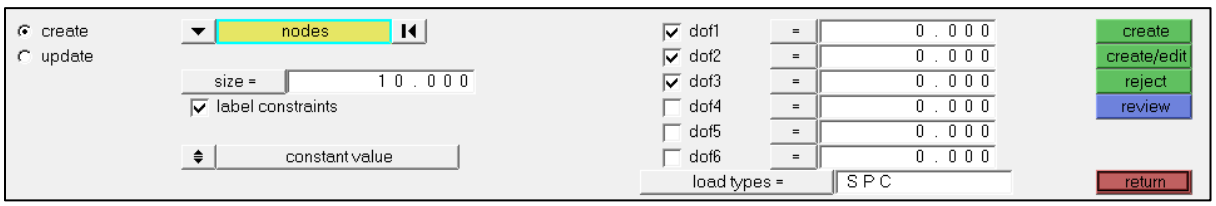


Step 9: Create boundary conditions; constraints and loads

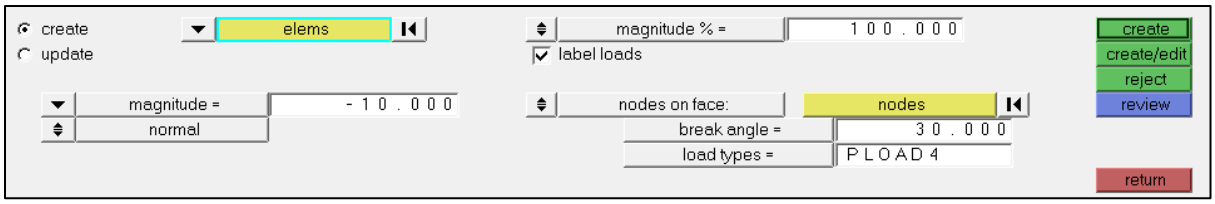
1. Create a load collector to "hold" the constraint definition. From the menu bar, select ***Collectors > Create > Load Collectors***.
2. Enter the **Name** as `Constraint`, set the **Card image** as ***none***, set the color as ***Blue***, and click ***Close***. Note that this operation makes **Constraint** the current collector. Any created boundary condition automatically gets organized into the current collector.
3. Now we will create simple support constraints (SPC) on all edges of the door surround composite skin. From the menu bar select ***BCs > Create > Constraints***.
4. With the nodes collector active, click on the ***nodes >> by path*** and then select ***each corner node*** of the door surround composite skin in ***clockwise fashion***.

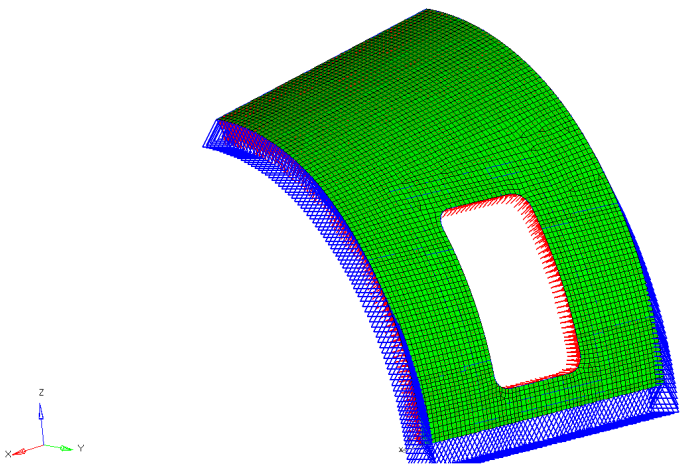


5. Check ***dof1*** (x-translation), ***dof2*** (y-translation), and ***dof3*** (z-translation); and uncheck ***dof4*** (x-rotation), ***dof5*** (y-rotation), and ***dof6*** (z-rotation).



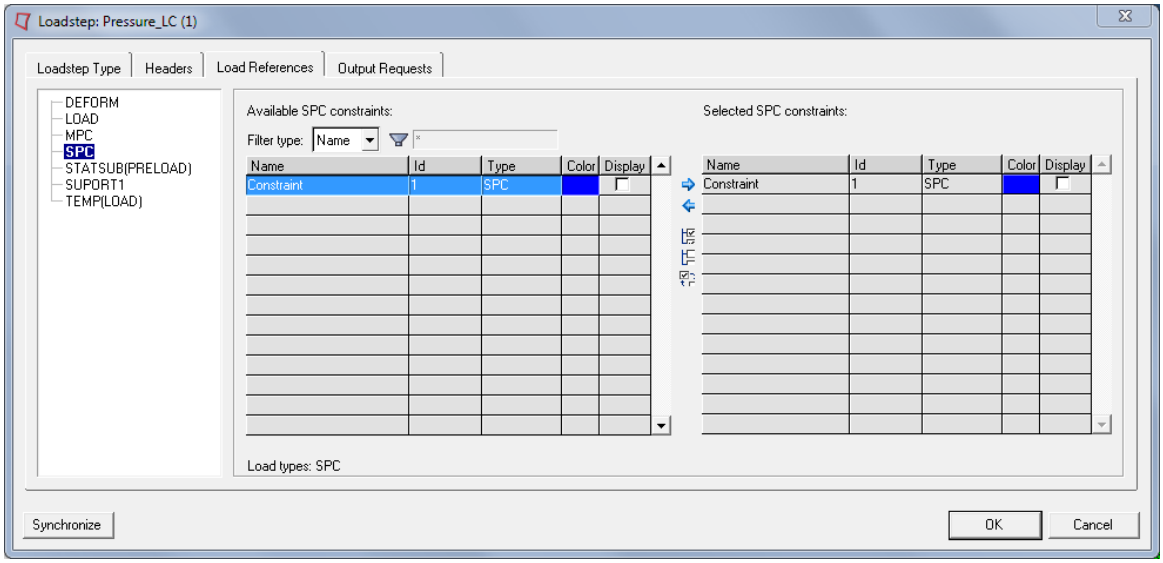
6. Set the load types as ***SPC***. Click ***create*** to create the simple support constraint.
7. Click ***return*** to exit the panel.
8. From the menu bar select ***Collectors > Create > Load Collector***.
9. Define the **Name** as `Pressure`, set the **Card image** as ***none***, set the color as ***Red***, and click ***Close***. Note that this operation makes **Pressure** the current collector. Any created boundary condition automatically gets organized into the current collector.
10. Create a constraint pressure (PLOAD4 = 10 psi) on the IML side of the door surround composite skin applied outward toward the OML. From the menu bar, select ***BCs > Create > Pressures***
11. With the element collector active, select ***elems>>all***.
12. Enter the **magnitude** as `-10.0`, since the element normals are pointing “inward” and we want the pressure to be applied “outward”.
13. Set the direction as ***normal***.
14. Set the **load types** as ***PLOAD4***.
15. Click ***create*** to create the pressure loads.
16. Click ***return*** to exit the panel.



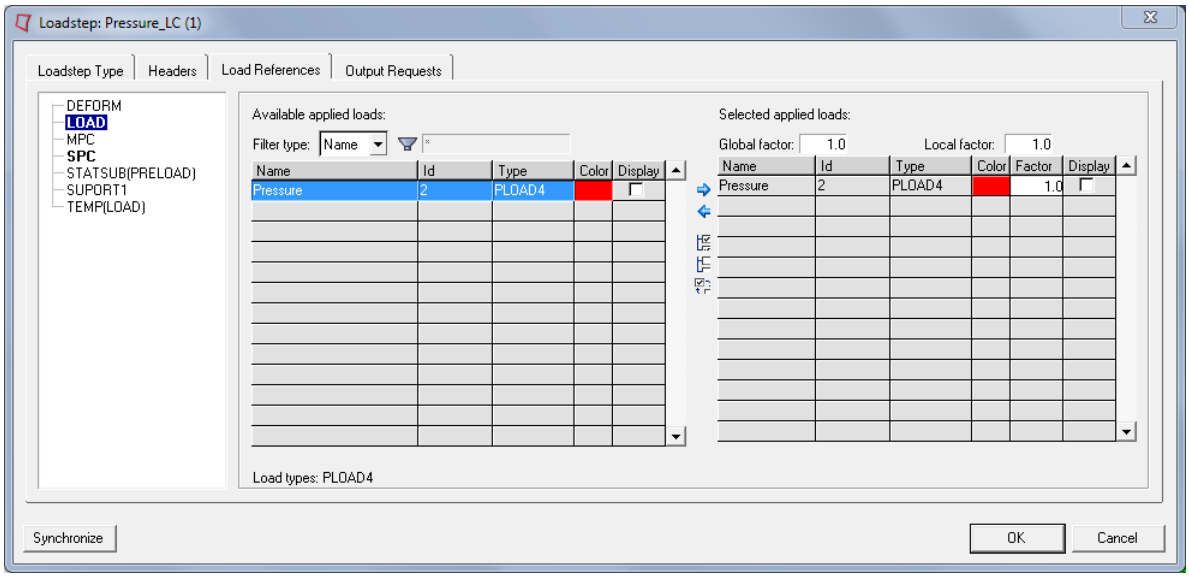


Step 10: Create load steps and control cards

1. Next we will create a pressure loadstep (subcase). From the menu bar select ***Tools > Load Step Browser***.
2. Within the **Load Step Browser** right click in the white space and select ***New Loadstep***.
3. Define the **Loadstep name** as `Pressure_LC` and click ***Create***.
4. Click on the **Loadstep Type** tab and select ***Linear static*** for the loadstep type.
5. Click on the ***Headers*** tab and do nothing. You can optionally define loadstep (subcase) title/subtitle/etc. information on this tab.
6. Click on the ***Load References*** tab and then click **SPC** to define the SPC definition for this linear static loadstep (subcase). The **Available SPC constraints table** lists the available SPC constraints that can be defined for this linear loadstep (subcase).
7. Select ***Constraints*** and then select the arrow pointing to the right to add it to the **Selected SPC constraints table**.

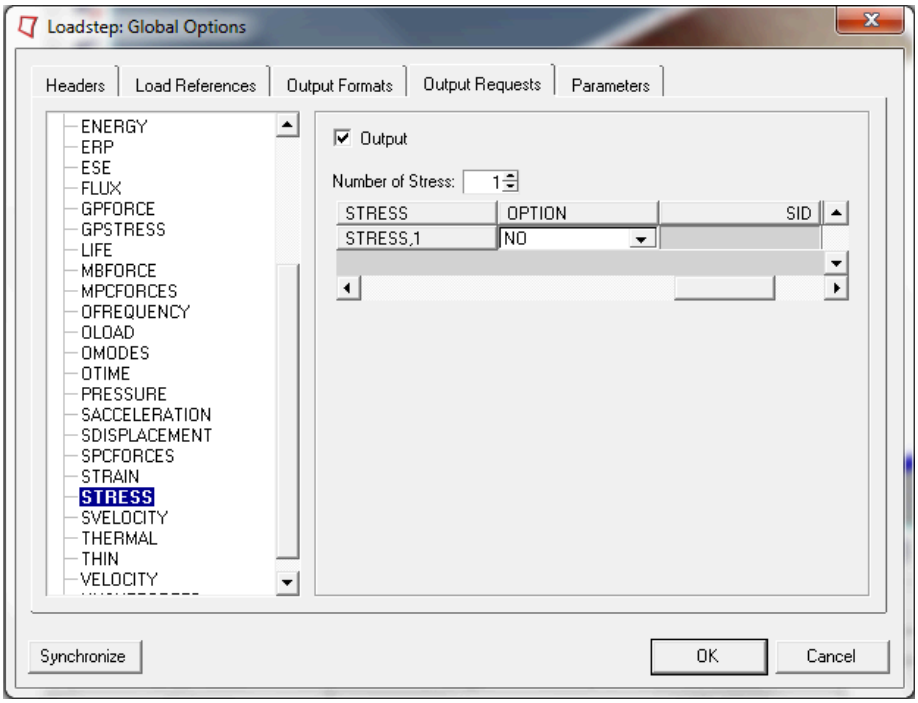


8. Remaining on the **Load References** tab, click ***LOAD*** to define the applied load definition for this linear static loadstep (subcase).
9. The **Available applied loads table** lists the available applied loads which can be defined for this linear static loadstep (subcase). Select ***Pressure*** and add it to the **Selected applied loads table** by click the right arrow button.

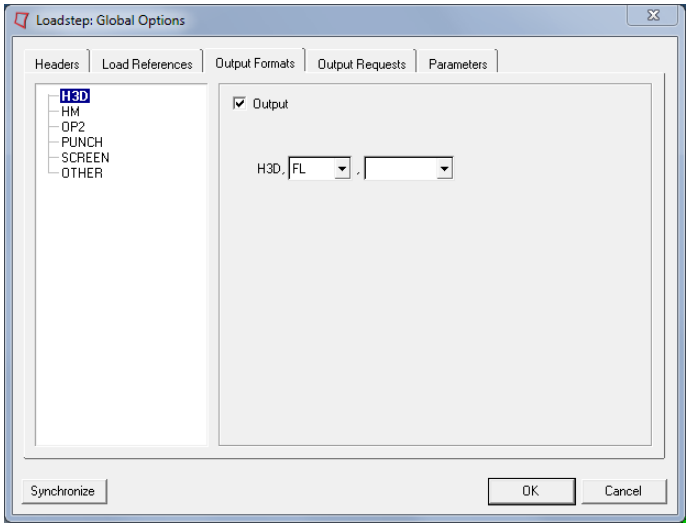


10. Click on the ***Output Requests* tab**, and do nothing. You can optionally define loadstep (subcase) dependent output requests on this tab. However, we will define global output requests in the next step for this particular model setup.
11. Click ***OK*** to finish defining the loadstep (subcase) and exit the wizard. You will notice that the loadstep definition now exists in the tree structure of the **Loadstep Browser**.
12. Now we are going to define the global output requests. From the **Loadstep Browser**, right click on ***Global Options*** and select ***Edit Options*** to display the **Loadsteps Global Options** dialog.

13. Select the ***Output Requests tab***, then select ***CSTRAIN*** and check ***Output*** to obtain ply-level strain for plies with results.
14. Select ***CSTRESS*** and check ***Output*** to obtain ply-level stress for plies with results.
15. Select ***DISPLACMENT*** and check ***Output*** to obtain nodal displacements.
16. Select ***STRESS*** and check ***Output***. In addition, scroll to the right and set **OPTION** to ***NO*** so that homogeneous shell stresses are not output.

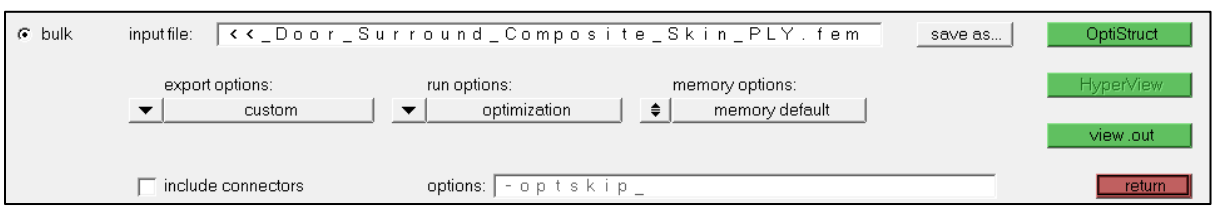


17. Next we will define the output file format. Remaining on the **Loadsteps Global Options** dialog, click the ***Output Formats tab***.
18. Select ***H3D*** and check ***Output*** to obtain results in the .h3d file format.
19. Click ***OK*** to finish defining the loadstep global options and exit the dialog. You will notice that the loadstep global options now exist in the tree structure of the **Loadstep Browser**.



Step 11: Save, Export, and Solve the ply-based model in OptiStruct

1. From the menu bar, select ***File > Save as > Model***.
2. In the Save Model as dialog, save the model as `9a_Door_Surround_Composite_Skin_PLY.hm`.
3. From the panels area, select the ***Analysis*** page and enter the ***OptiStruct*** panel.
4. Keep all entries as default, and enter `-optskip` in the **options** field as shown below.



5. Click ***OptiStruct*** to export the model deck and submit to the OptiStruct solver automatically from within HyperMesh.
6. Once the run is completed, click ***return*** to exit the panel.

Step 12: Post-process the ply-based model

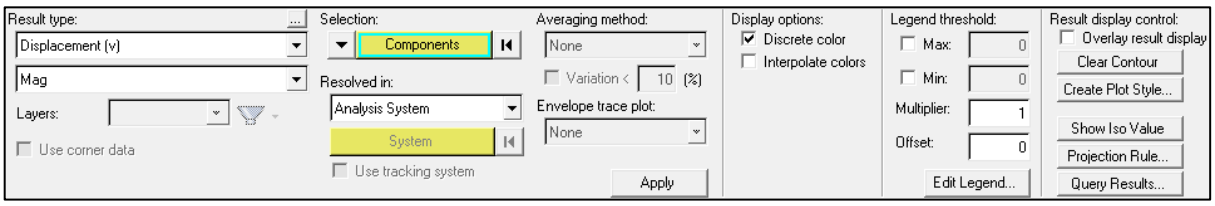
1. Add a page 2 to the session for post-processing the model. From the **Page Controls** toolbar; click the ***Add Page*** button to add a page 2 to the session. By default the new page single window client will be a HyperView client.

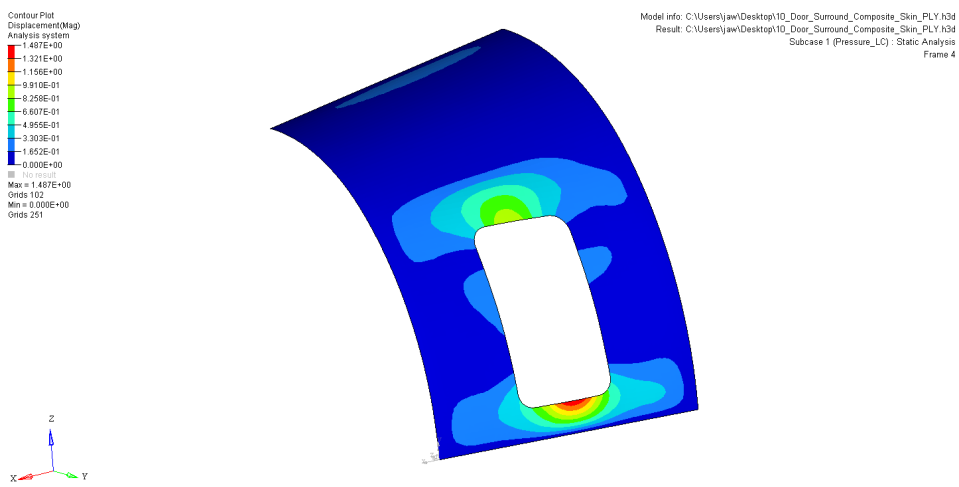
2. Open the model and results files for post-processing within HyperView. From the menu bar select ***File > Open > Model***.

3. Within the panel, select the `10_Door_Surround_Composite_Skin_PLY.h3d` file for both **Load model** and **Load results** and click ***Apply***. The model and results get loaded into HyperView for post-processing.

4. From the **Results** toolbar, click ***Contour*** to open the **Contour** panel.

5. Within the **Contour** panel set the **Result type** as ***Displacement (v)*** and the component to ***Mag*** and click ***Apply***. This contour plots the nodal displacements.

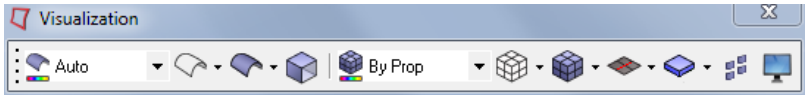




Step 13: Realize the ply-based model as a zone-based mode

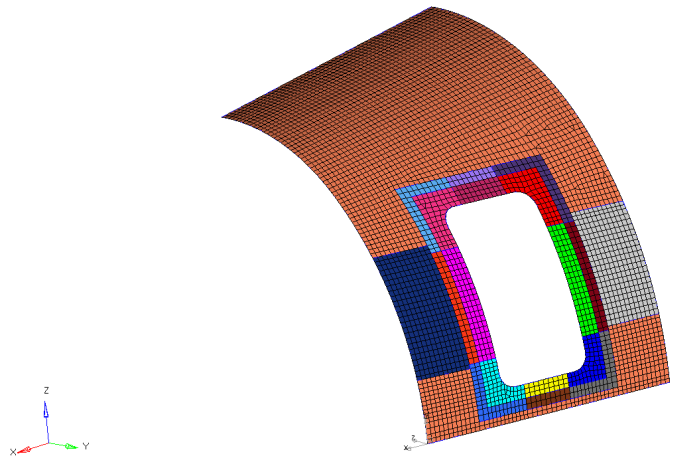
1. Switch back to page 1 where the HyperMesh client exists in window 1. From the page controls toolbar; click the ***Previous Page*** button to switch back to page 1.

2. Realize the ply-based model into a zone-based model. Make sure the ***By Prop*** color mode is activated on the **Visualization** toolbar.



3. From the **Model Browser**, expand both the ***Laminate*** and ***Property*** folders and select the ***Stack*** laminate.

4. Right click on the selection and select ***Realize***. This operation will convert the ply-based model defined by PLY/STACK into a zone-based model defined by PCOMP/G. You should notice 19 additional properties (PCOMP/G) get generated and assigned to the proper elements as you can visualize from the color coding of the model in By Property color mode shown below.



5. Edit the zone-based property to review the PCOMP/G definitions. From the **Model Browser**, select ***prop_lam1_13*** from the **Property** folder.
6. Right click on the selection and select ***Card Edit***. This will bring up the PCOMP/G card image for the **prop_lam1_13** property. Notice that this definition is equivalent to the ply-based form and that it retains the ply Ids for each ply.
7. Click ***return*** to exit the card image dialog.

Step 14: Save, Export, and Solve the zone-based model in OptiStruct

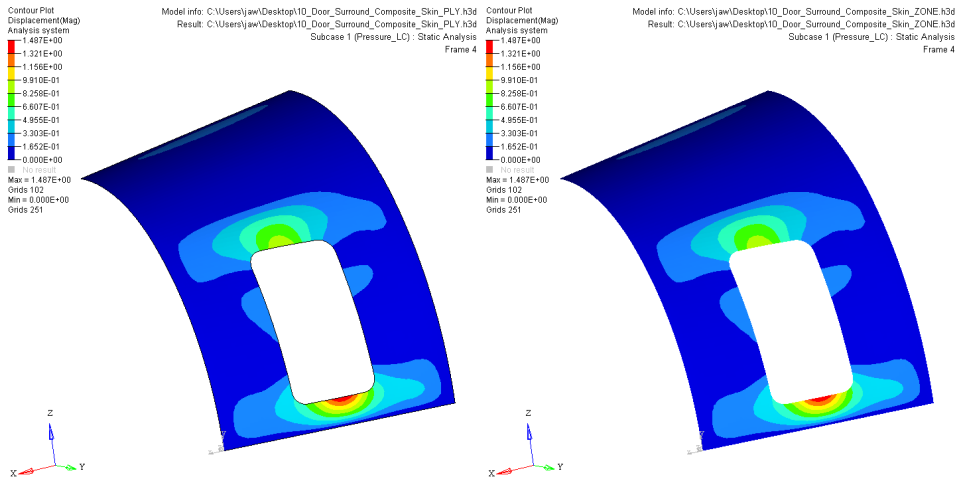
1. From the menu bar, select ***File > Save as > Model***.
2. In the **Save Model as** dialog, save the model as `10_Door_Surround_Composite_Skin_ZONE.hm`.
3. From the panels area, select the ***Analysis*** page and then the ***OptiStruct*** panel.
4. Keep all entries as default, and enter ***–optskip*** in the options field.
5. Click ***OptiStruct*** to export the model deck and submit to the OptiStruct solver automatically from within HyperMesh.
6. Once the run is completed, click ***return*** to exit the panel.

Step 15: Post-process the zone-based model

1. From the **Page Controls** toolbar, click the ***Next Page*** button to switch back to page 2.

2. From the **Page Controls** toolbar; click the ***Page Window Layout*** button and select a two window layout, .
3. Click inside the currently empty page 2 window 2 to make it the active client (blue halo around the window frame). From the Menu bar, select ***File > Open > Model***.
4. Within the panel, select the `10_Door_Surround_Composite_Skin_ZONE.h3d` file for both **Load model** and **Load results** and click ***Apply***. The model and results get loaded into HyperView for post-processing.
5. From the **Page Controls** toolbar, click the ***Synchronize Windows*** button, make sure ***both windows are "blue"*** and click ***OK***. This operation synchronizes the display for both widows. Whatever display manipulation you make in one window will automatically be applied to the other window. Try it out by translating, rotating, zooming the model in one of the windows. Both models move together!
6. Click inside page 2 window 2 to make it the active client (blue halo around the window frame). From the **Results** toolbar, click ***Contour*** to open the **Contour** panel.

   Within the **Contour** panel set the **Result type** as ***Displacement (v)*** and the component as ***Mag*** and click ***Apply***. This contour plots the nodal displacements.



Notice that the displacement results between the ply-based and zone-based models are identical. Thus we can conclude that the ply-based realization into a zone-based model was accurately performed!