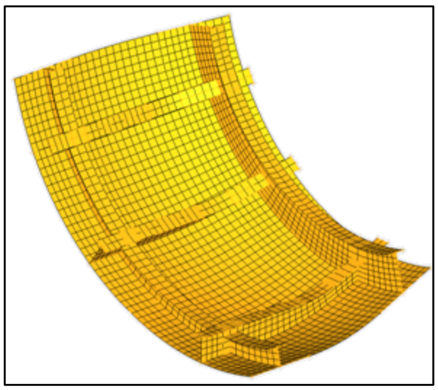
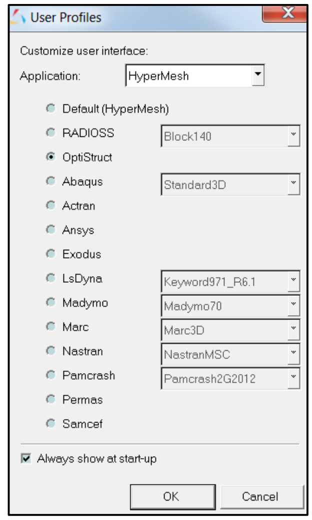
Exercise 9a - Analysis Setup and Loading

This exercise will focus on setting up a model for analysis. At the end of this exercise, you will run an analysis in OptiStruct. While this exercise is focused on an OptiStruct Analysis setup, the methods and techniques explored here are applicable to a setup in any solver.

Step 1: Load the file `9a-ANALYSIS-SETUP-OPTISTRUCT.hm` and the OptiStruct user profile.





Step 2: Studying the Model

The normal process for setting up an analysis would be the setup of materials, properties and components before the meshing of the model. As this exercise focuses only on analysis setup, the mesh has already been created for you.

This model is a quarter segment of a submarine pressure hull. The exercise will cover the steps required to analyze the stress on the hull of a decent to a depth of 300 meters and determine if the hull design can handle that pressure.

1. Take a few minutes to familiarize yourself with the model and get a concept of the size and scale of the parts.
2. Based upon measurements and knowledge of how large a submarine is, what would you assume to units of this model to be?

Now that the scale of the model has been determined, it is important to establish a unit scheme. These are often dictated by corporate standards, but in this case it will be established by the units that were used to create the model.

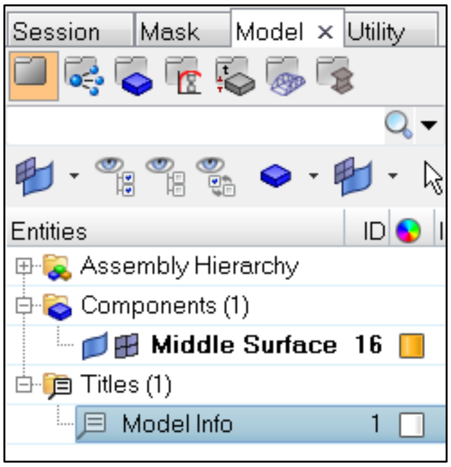
For this analysis, the **Millimeter-Ton-Second** scheme will be utilized.

The first step in any analysis should be model organization. This frequently occurs before the model is meshed but can be done post mesh as well.

To make sure each step has the information already available, the ideal order is to create materials first, then properties and then finally component collectors.

Step 3: Model Organization

In this step, we will take the elements that represent the Hull and place them into the Hull component. The collector that holds the remaining Rib elements will then be renamed Ribs.

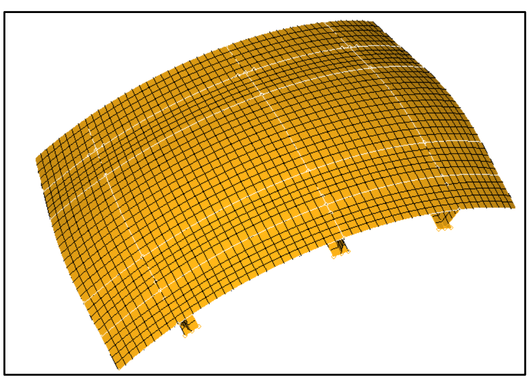


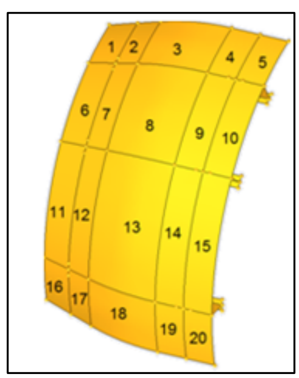
Component collectors are, as previously discussed, used for model organization. One of the most logical organization schemes for this model would be a component for the Hull elements and then another for the Ribs. This, of course, is only one method and could be altered for any number of organizational reasons.

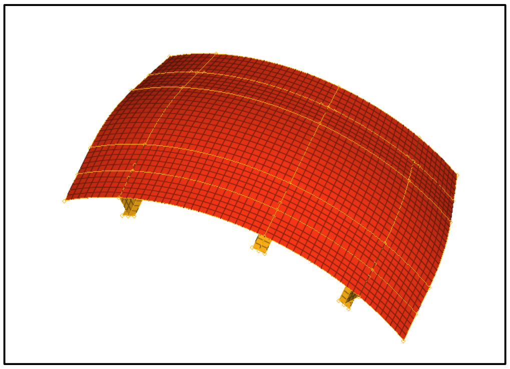
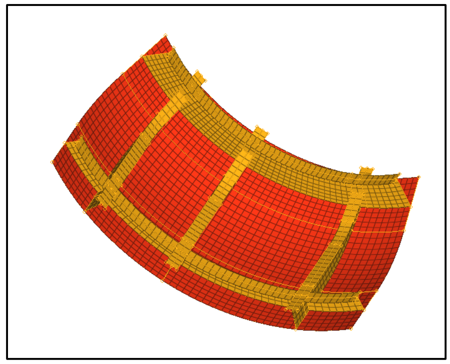
1. Right click mouse button in the **Model Browser** and select **Create >** ***Component***
2. A new component will be created and you can edit it using **Entity Editor**. Enter `Hull` in the **Name** field. Assign it a unique color.
3. **Organize** the **Hull** elements into the **Hull** component, by clicking  or from the menu bar, click **Mesh > Organize > Elements >** ***to Component***.

   Click the selector ***elems >> by geoms***, then active the **surfs** selector to select the surfaces and in the end, click the green button **add to selection.**

   <u>HINT:</u> Using the extended selection option of **by geoms** and picking the 20 surfaces that make up the hull is the easiest way to get all of the appropriate elements.





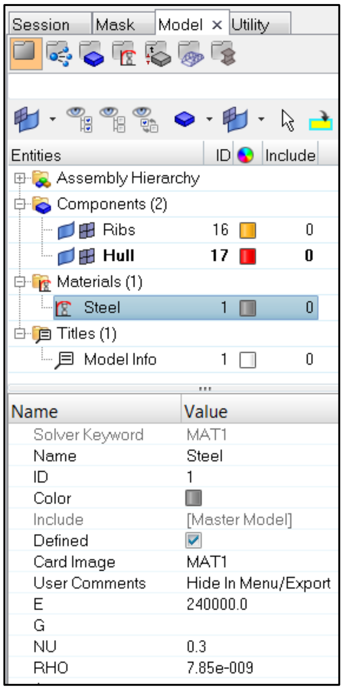
 

4. **Rename** the **Middle Surface** component to `Ribs.`

Step 4: Material and Property Creation.

1. Right click mouse button in the **Model Browser** and select **Create >** ***Material.***
2. A new material will be created and you can edit it using **Entity Editor**. Enter `Steel` in the **Name** field and pick a **color**.
3. For **Card Image** select ***MAT1*** (A Linear Elastic Isotropic Material).
4. Click **[E]**, **[NU]** and **[RHO]** to open the fields. These fields are the material properties for the material being created and are defined as follows:
   - [E] Young's Modulus (Modulus of Elasticity)
   - [NU] Poisson's Ratio
   - [RHO] Density

   As it has been established the Millimeter-Ton-Second unit scheme will be utilized, the Young's Modulus needs to be in terms of Newton/mm^2 (MPa) and the Density in Ton/mm^3. Poisson's ratio is unit-less and is the same no matter what the unit scheme.
   Enter the following values:
   - [E] `2.4e+5`
   - [NU] `0.3`
   - [RHO] `7.85e-9`



At this point you can see that a new folder has been created in the **Model Browser**, **Material**, and the new material, **Steel**, is included in it.

5. Right click mouse button in the **Model Browser** and select **Create >** ***Property.***

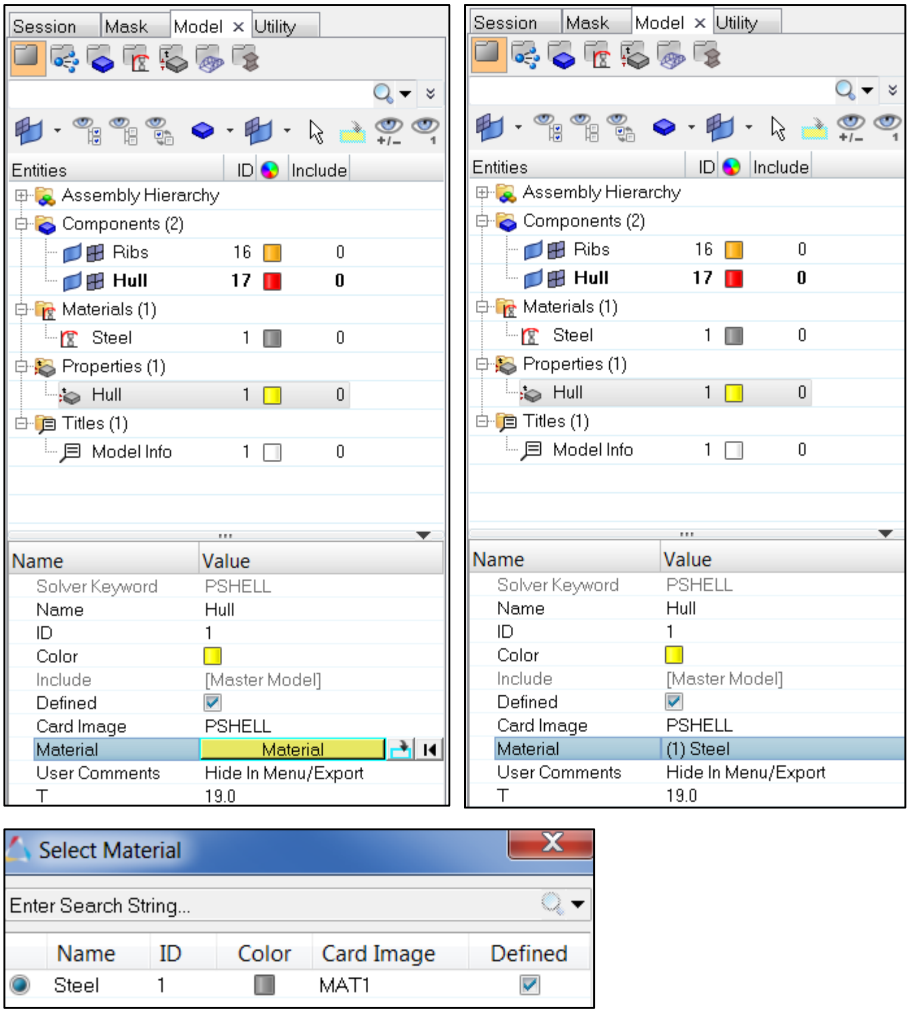
   While the elements (quads and trias) have been created, they need to be defined as an entity the solver can analyze. In the case of OptiStruct, these 2D elements are defined as PSHELL. Creating the PSHELL property will give these elements their definition (card Image) and will allow for the definition of the material thickness they have.

6. A new property will be created and you can edit it using **Entity Editor**. Enter `Hull` in the **Name** field and pick a color.

7. For **Card image** select ***PSHELL.***

8. Set the value for Thickness **T** field at `19`.

9. Go to the **Material** field; “**Select from list**” the material “***Steel***”, to assign this material to the property.



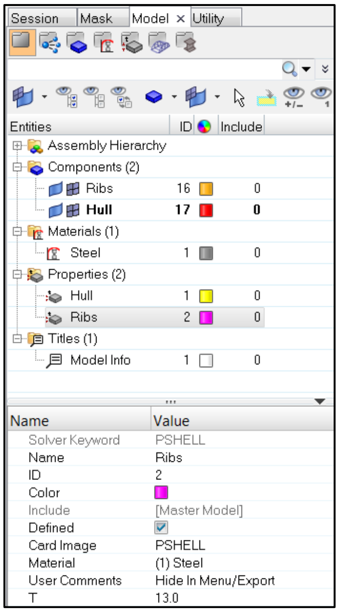
10. Right click mouse button in the **Model Browser** and select **Create >** ***Property.***
11. Using the techniques explored, create a property with the name **Ribs** with the following settings:

    Card image = *PSHELL*

    Material = *Steel*

    Thickness = *13*

    Set a **color**.

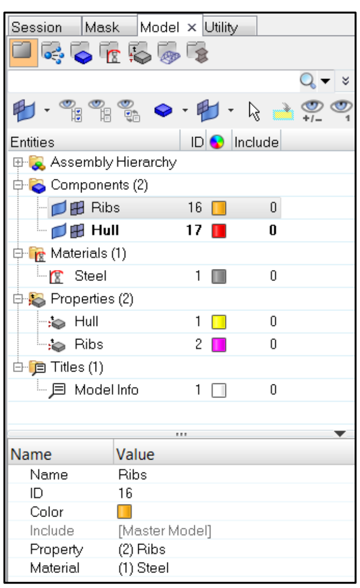


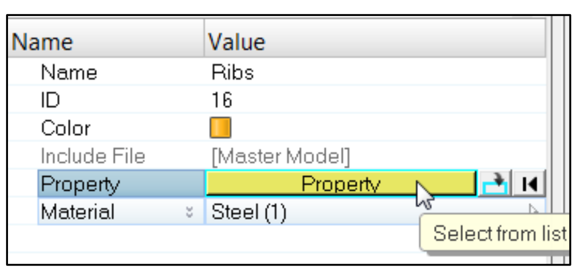
Step 5: Property and Material Assignment.

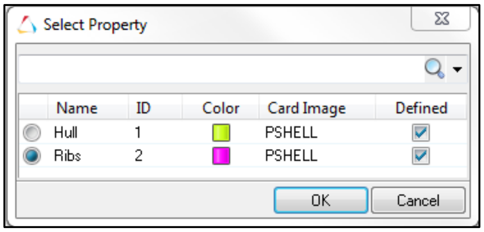
Property and material can be created without creating a component at the same time. This is usefull when the components already exists, you can assign property and material later.

As the component were created prior to the creation of the properties, it is now necessary to assign the property to them.

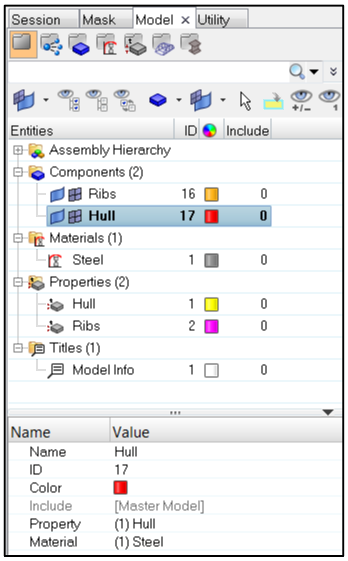
1. From the **Model Browser,** select component “**Ribs**”.
2. The **Entity Editor** will open, Go to the **Property** field; “**Select from list**” the property “***Ribs***”, to assign this property (and associated material “**Steel**”) to the “**Ribs**” component.

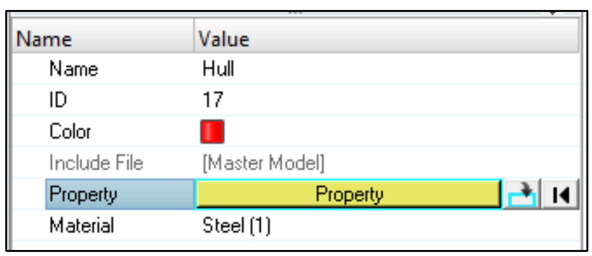


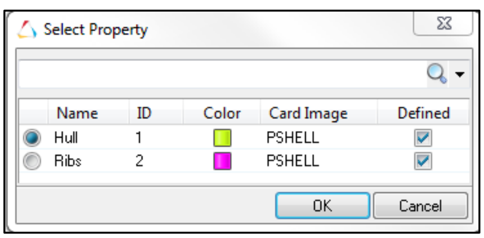




3. From the **Model Browser,** select component "**Hull**".
4. The **Entity Editor** will open, Go to the **Property** field; "**Select from list**" the property "***Hull***", to assign this property (and associated material "**Steel**") to the "**Hull**" component.

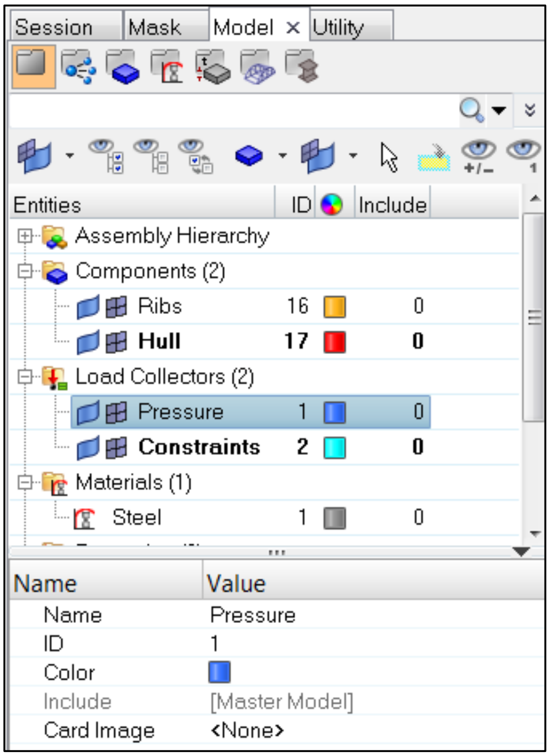


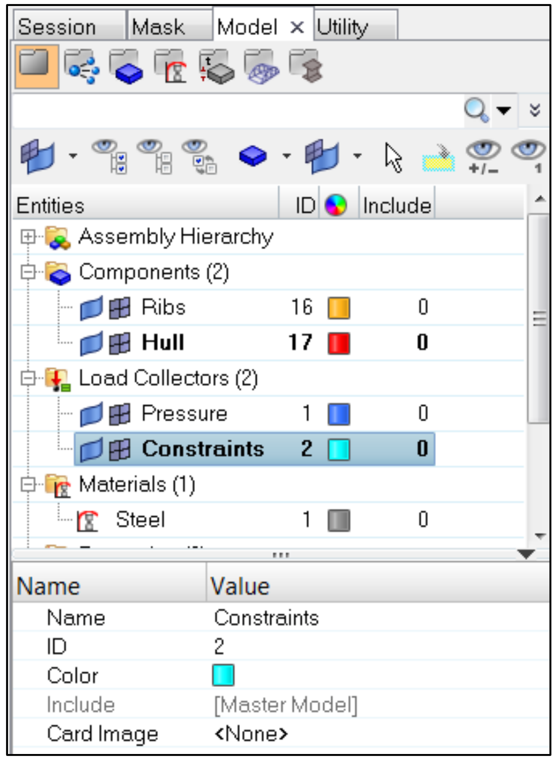




Step 6: Load Collector Creation

1. From the **Model** Browser, click right mouse button to create a ***LoadCollector***.
2. The **Entity Editor** will open and name it `Pressure`.
3. Assign it a unique color
4. Leave the **Card image** as ***<None>.***
5. Follow the previous steps to create another **LoadCollector** called **Constraints**.



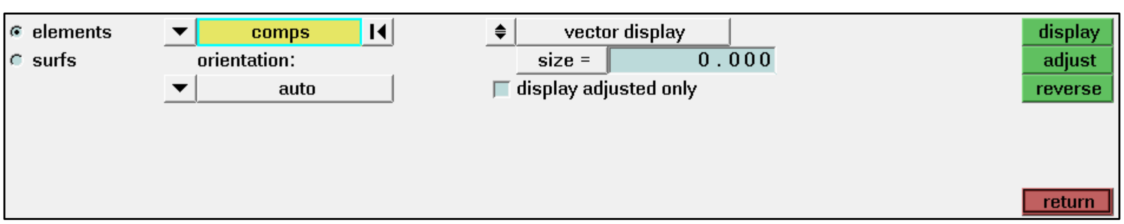


Step 7: Model Loading

With the elements properly assigned a card image (through the property) and a material, it is now necessary to create the loads on the model. As this is a submarine hull, a constant pressure will be applied to the exterior of the hull, directed inwards normal to the elements.

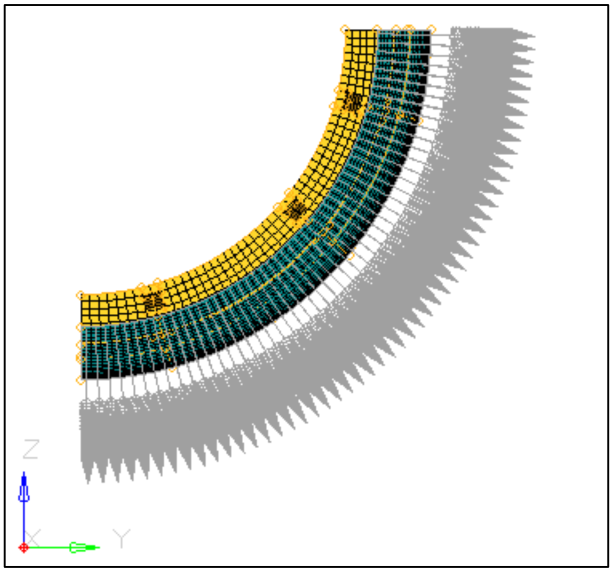
To establish the orientation of the pressure load, the element normals direction must first be discovered.

1. Go to **View > Toolbars > HyperMesh > *Checks*** toolbar, select the ***Normals*** icon .
2. In the **elements** sub-panel select all of the elements in the **Hull** collector.



3. Click ***display***.

   Arrows should now indicate the element normal direction.



The element normals should be pointing outward from the **hull**, so if they are not, click the green button ***reverse***.

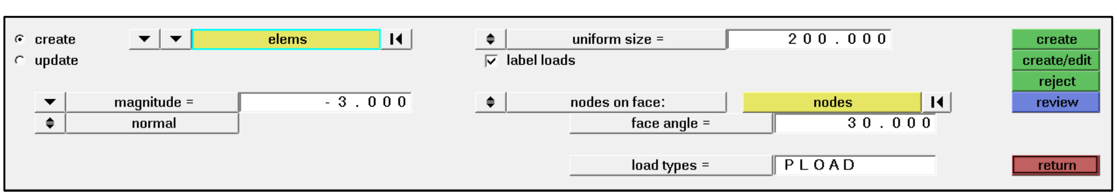
4. Make the **Pressure** Load Collector **current**.
5. From the menu bar, click **BCs > Create > *Pressures*** panel.
6. In the **create** sub-panel, select the **elements** in the **Hull** collector.
7. Set the ***magnitude* =** to `-3.0`. (This value is in MPa and corresponds to the approximate pressure at a depth of 300 meters)

   The direction switch under the magnitude field allows for the direction of the pressure to be set. If this value is NOT set then the default is to make the pressure normal to the element. The value previously entered was negative so that the pressure is opposite the element normal and thus directed inwards.

8. Change the **relative size=** toggle to **uniform size =** and set it to `200`.

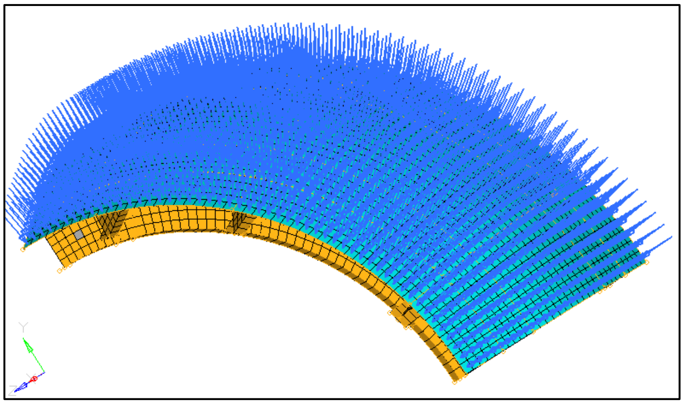
   This option establishes the size of the arrow that will graphically represent the load. "Relative size" will make the arrow length the set percentage of the value of the load in model units. For example in our case of a 3.0 magnitude load, a relative size= value of 200 would result in a load arrow of 6 units in length. Uniform size will set the length to the set number of model units regardless of the magnitude value.

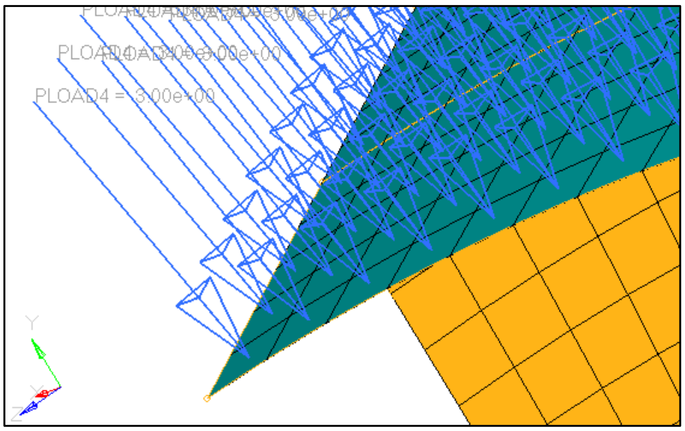
9. Click the **load types=** button and select ***PLOAD***.



   PLOAD is the standard pressure loading card in OptiStruct Analysis. For explanations of other types of pressures and loads you can refer to the online help.

10. **Create** the pressures by clicking the green button ***Create***. The model should now look similar to this picture.





Step 8: Save the Model

While this step is optional, it is good practice to frequently save your model.

Step 9: Constraints

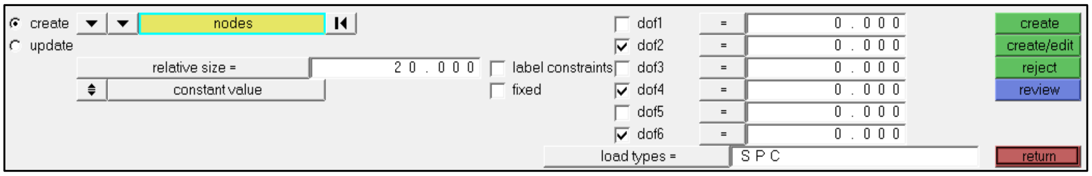
Constraints hold the model in place. Without them any force applied to the model would send it flying off. Constraints typically represent the physical restrictions on a part, some examples being welds, fasteners or other parts that constrain the part and allow it to resist the forces applied. These are represented through the use of an SPC (single point constraint) which restricts the movement of a single node in any of 6 degrees of freedom (X, Y, Z translational and X, Y, Z rotational).

In the case of this model, a special constraining system called Symmetric Constraining is used. This is a common practice when analyzing a part with some form of symmetry. In the case of this Submarine Hull model, it represents ¼ of the complete hull circle. Analyzing only part of a symmetric model saves time in both model setup and analysis. The results can be assumed to be identical across planes of symmetry, assuming the loading is also identical across the plane.

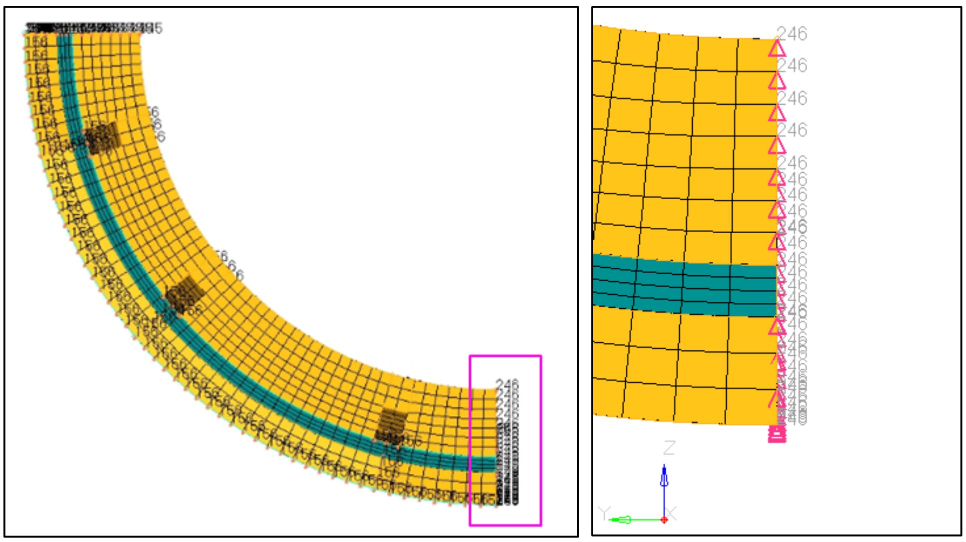
1. From the **Model** Browser, select Load Collector “**Pressure**” and right click mouse button on “***Hide***”
2. ***Make current*** the **Constraints** load collector.
3. From the menu bar, click **BCs > Create > *Constraints***.

4. Select the **YZ Front Plane View** .

5. Select or de-select the appropriate check boxes so that the only **DOFs** selected are **2**, **4** and **6**.

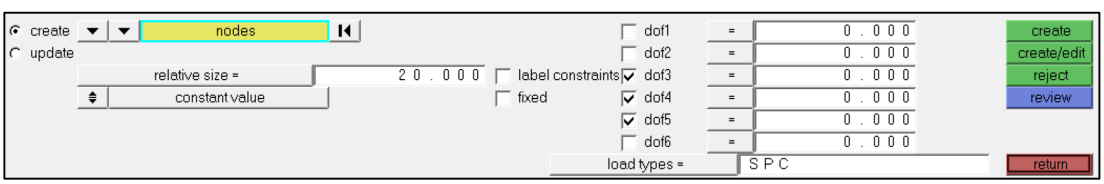


6. Using a box select (***HINT:*** **Shift+Left mouse button** Drag a box) to pick the nodes shown in the image below.

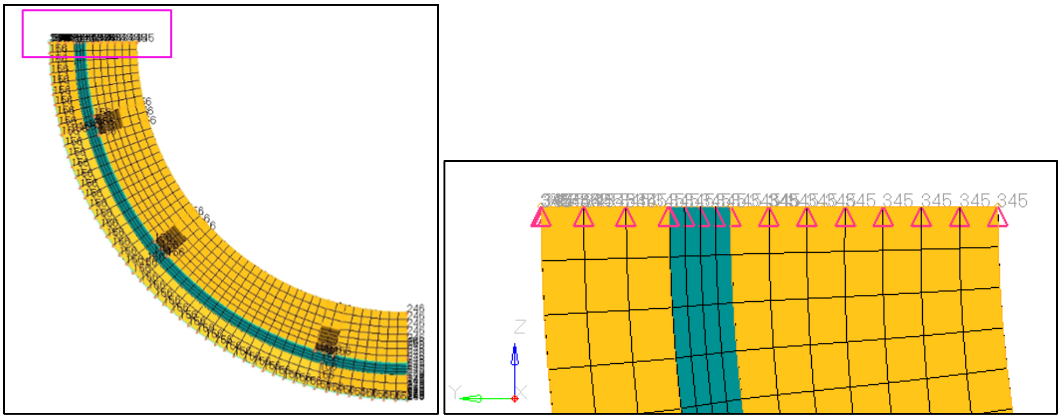


7. Click ***create***.

8. Select and de-select the appropriate check boxes so that the only **DOFs** selected are **3**, **4** and **5**.



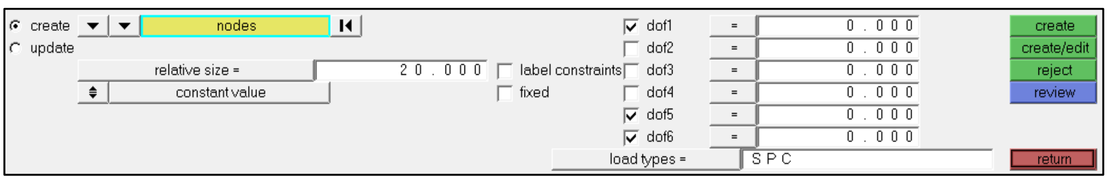
9. Using a box select pick the nodes shown in the image below.



10. Click ***create***.

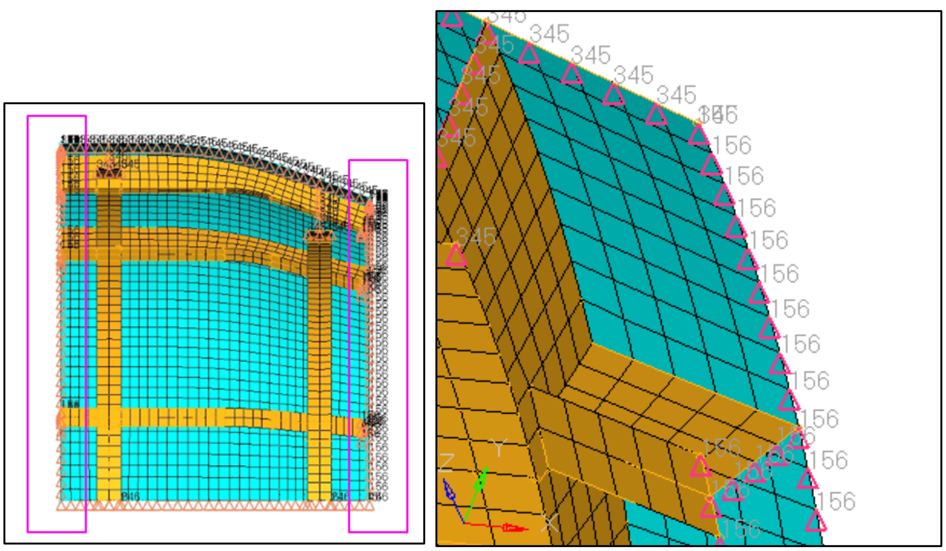
11. Select the **XY Top Plane View** .

12. Select and de-select the appropriate check boxes so that the only **DOFs** selected are **1**, **5** and **6**.



13. Using the standard views and model rotation tools, select all of the nodes on both remaining edges of the Hull elements.

    You will have to manually select the nodes at the end of the ribs, component "Ribs", zoom and rotate the model.



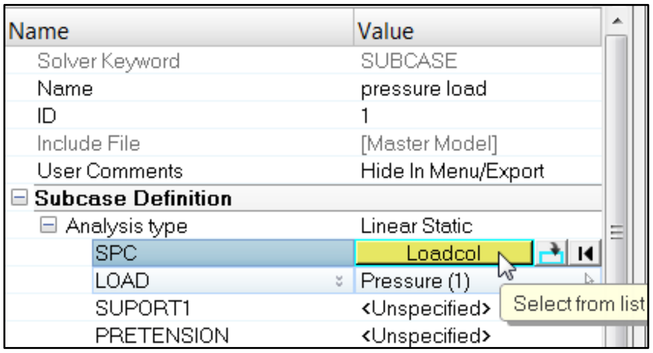
14. Click ***create***.

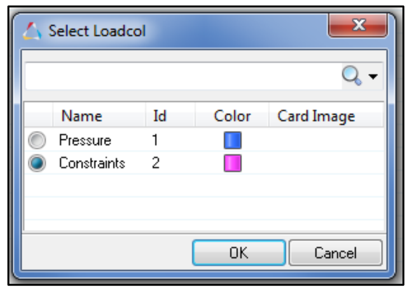
15. The model is now properly constrained for the analysis.

Step 10: Define the LoadStep

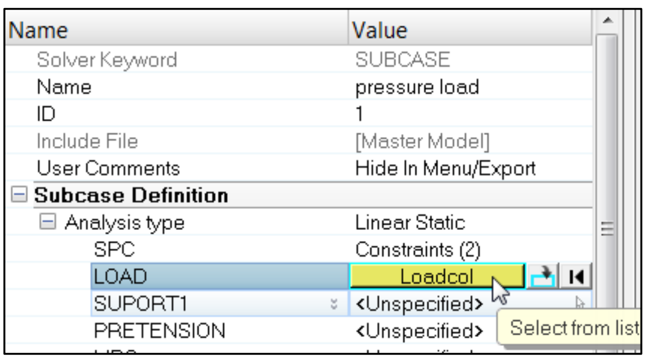
This step in the analysis setup is to establish a load step. A load step is combination of constraints and loads that will define a single analysis in the solver. Multiple load steps can be defined in a single model allowing for one run of the solver to conduct numerous studies.

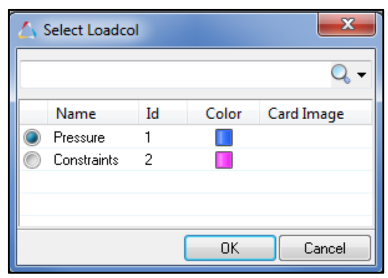
1. From the **Model** Browser, click right mouse button to create a ***Load Step***.
2. The **Entity Editor** will open and name it as `pressure load.`
3. Go to the **SPC** field; "**Select from list**" the Loadcol "**Constraints**".



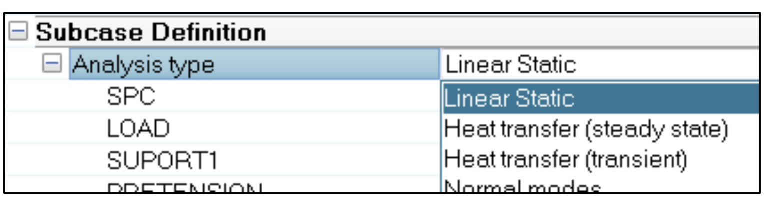


4. Go to the **LOAD** field; "**Select from list**" the Loadcol "**Pressure**".

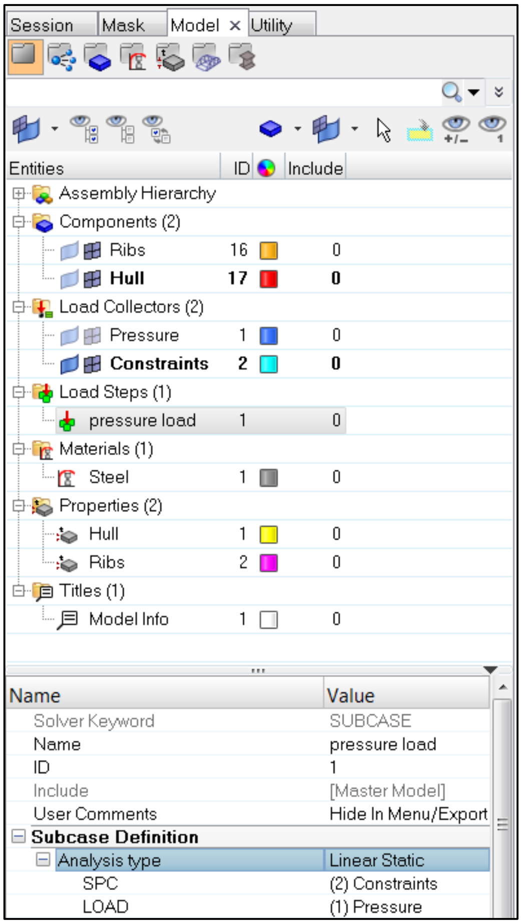




5. Go to the **Subcase Definition >** ***Analysis type*** option, change the Analysis type from **Generic** to **Linear Static** from the menu in the **Value** field.



6. The Load Step "**pressure load**" is defined.



Step 11: Control Cards

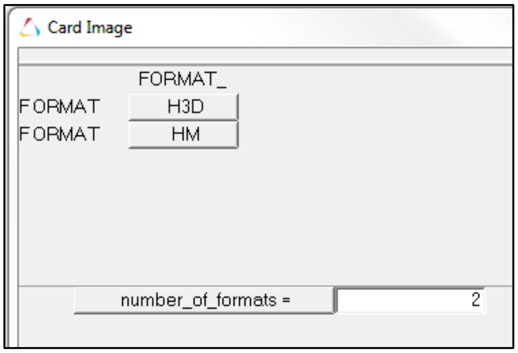
Control cards are special cards in the deck that control aspects of the solver run.

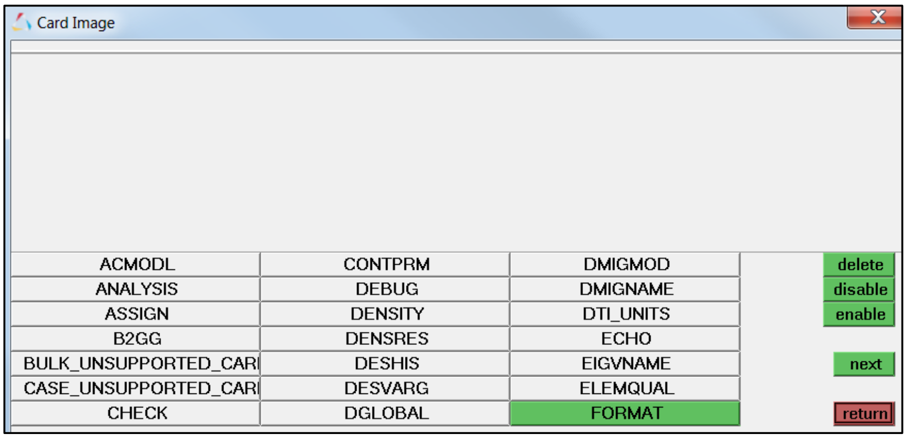
They can be used to:

- Set parameters of the analysis.
- Control aspects of the analysis.
- Request certain types of output.

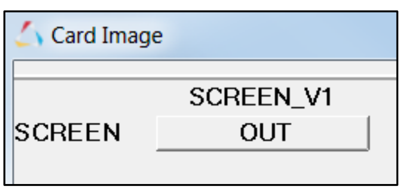
1. From the menu bar, click **Setup > Create > *Control Cards*** panel.
2. Find the **FORMAT** card and click on it. (Use the **next** button move scroll through the cards).
3. Change the **number_of_formats** field to `2`.
4. Change the second **FORMAT** card to ***HM***.

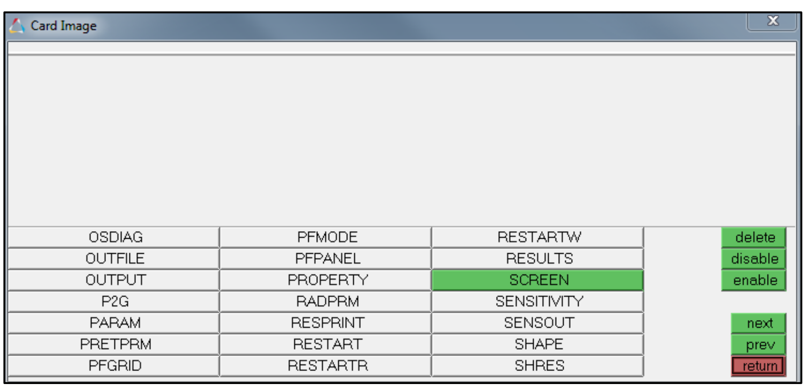
   This will provide output in both HyperView (H3D) and HyperMesh (HM) formats.



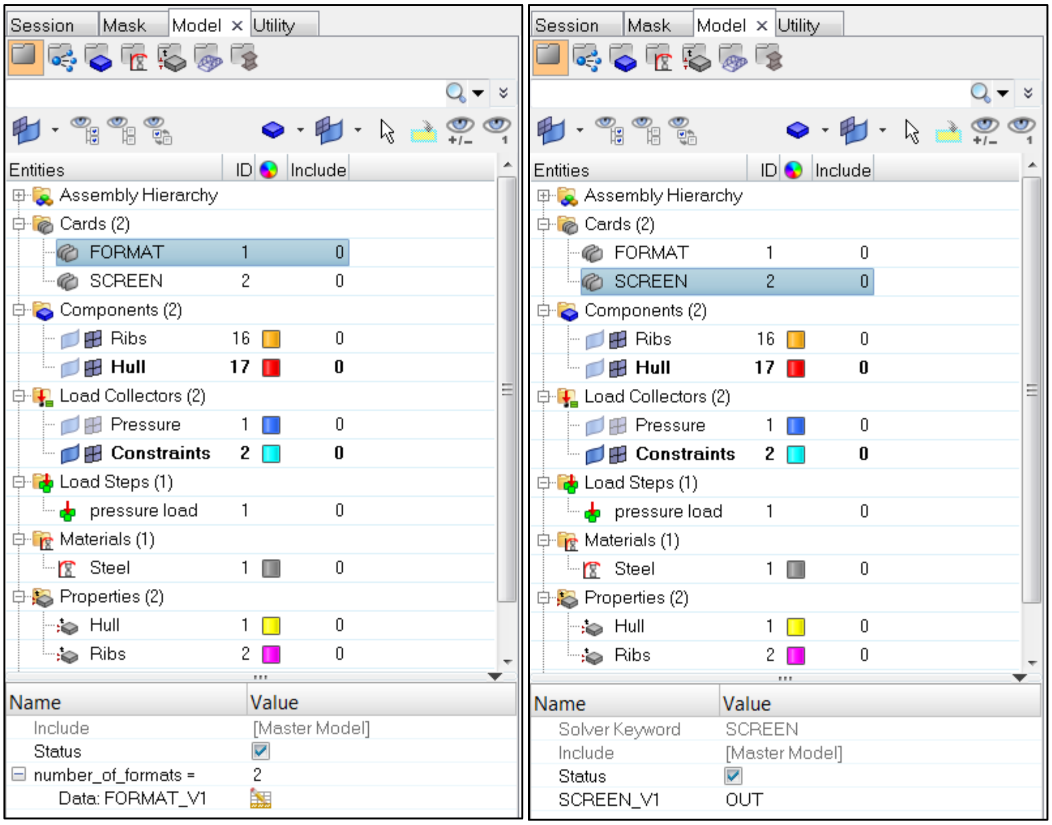


5. Click ***return*** and then use **next** to find the **SCREEN** card.
6. Set the **SCREEN_V1** to ***OUT***





7. From the **Model Browser,** have a look at the cards created.

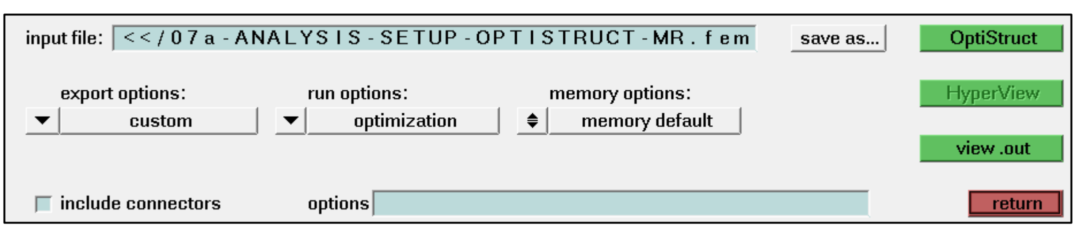


Step 12: Run the Analysis

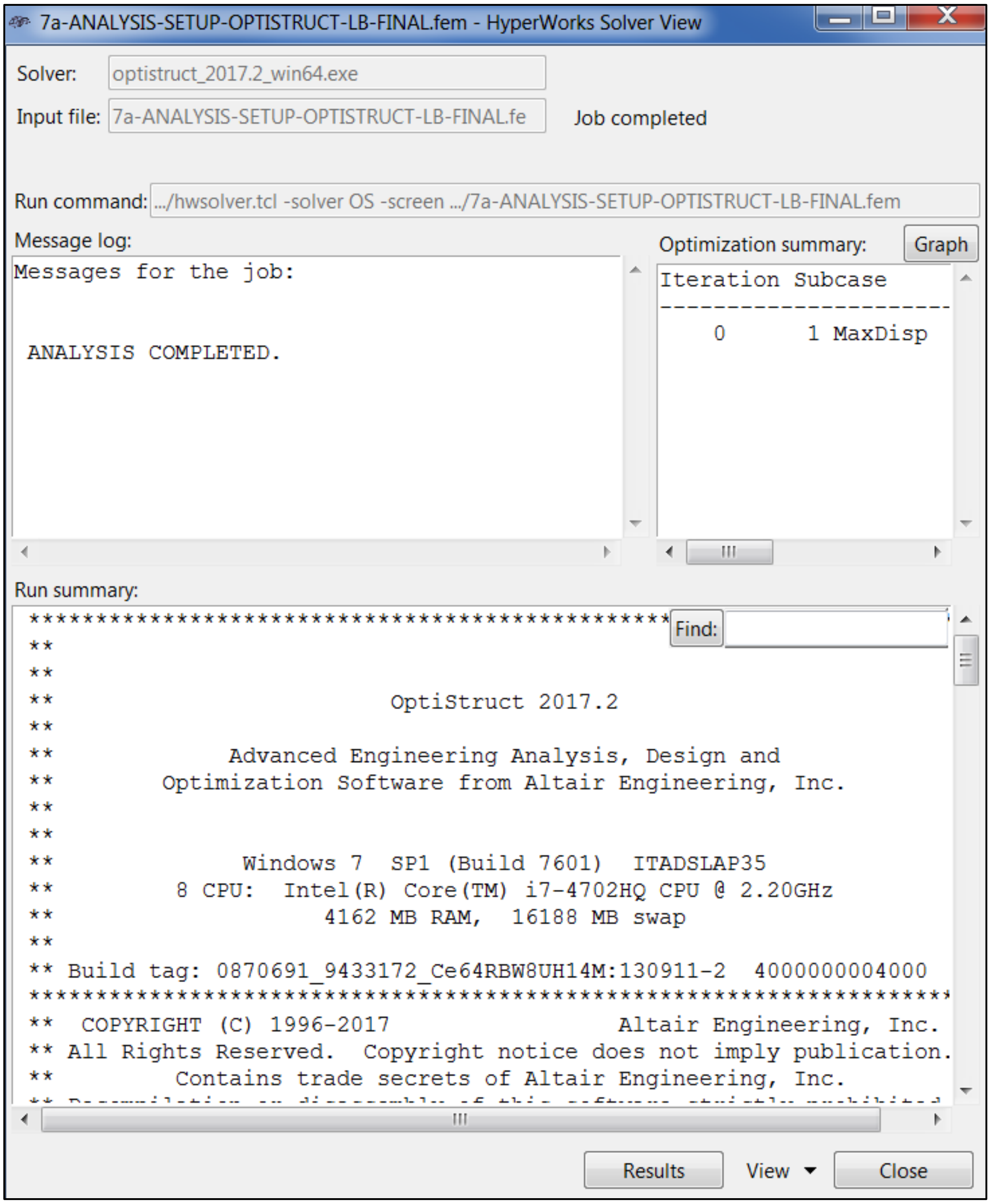
For any other solver the next step should be to export a solver deck and use the individual solver tools to being the study. As OptiStruct is an Altair product it can very easily be invoked from within HyperMesh.

1. From the menu bar, click **Optimization > *OptiStruct***.
2. Set the panel options to match those below.

   NOTE: Your model name and path will differ from the picture, leave the default.

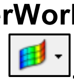


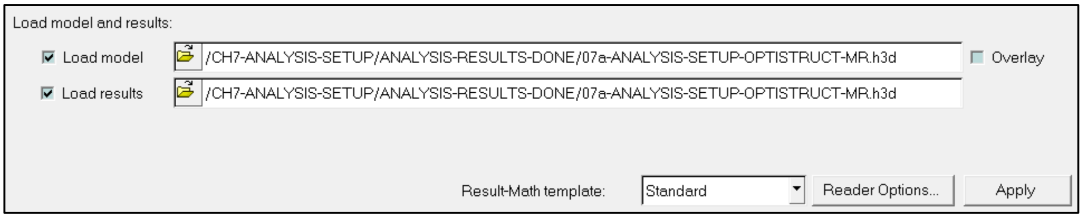
3. After the settings are made, click the ***OptiStruct*** button to begin the analysis.
4. A new window will open to show that the ***OptiStruct*** analysis is running.
5. When the message "**ANALYSIS COMPLETED**" appears, the run is complete and the window can be closed.



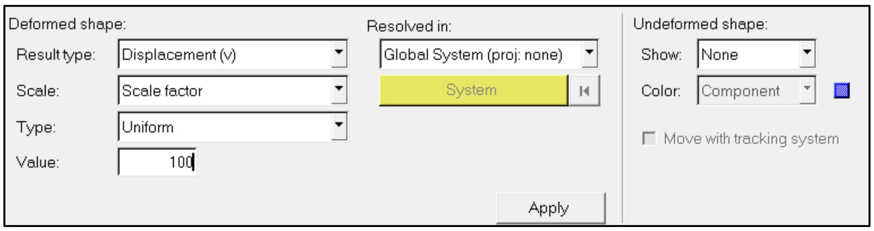
Step 13: Post Processing

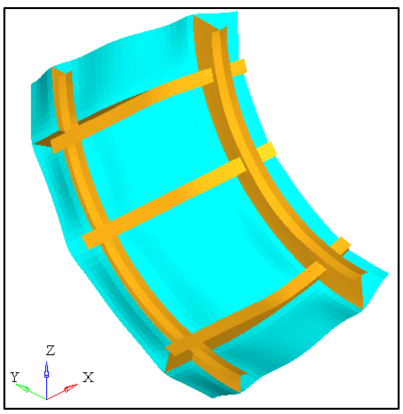
While the workings of HyperView will be discussed in greater length in the Post Processing section of the class, this step will cover basic post processing steps to review the analysis you just ran.

1. In the **HyperWorks Solver View** dialog box, click the ***Results*** button to load the results in ***HyperView*** .
2. If you want, you can load a different input/result file clicking on ***Load model*** (load **.fem** as input file) and ***Load results*** (load **.res** as result file); leave **h3d** format for now and click on ***Apply***.

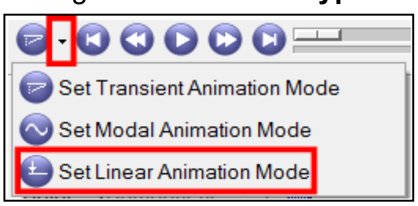


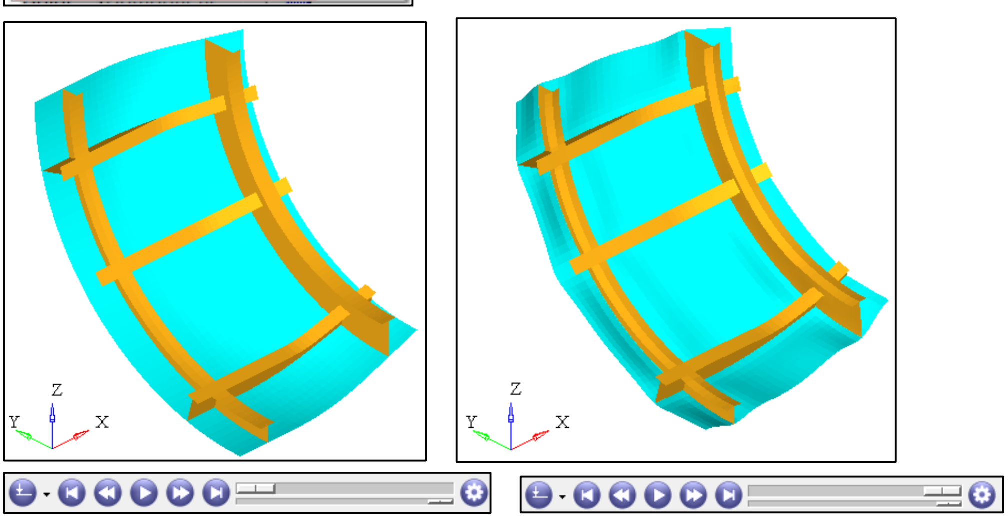
3. Enter the ***Deformed*** Panel .
4. Set the **Value** to `100` and click ***Apply***.



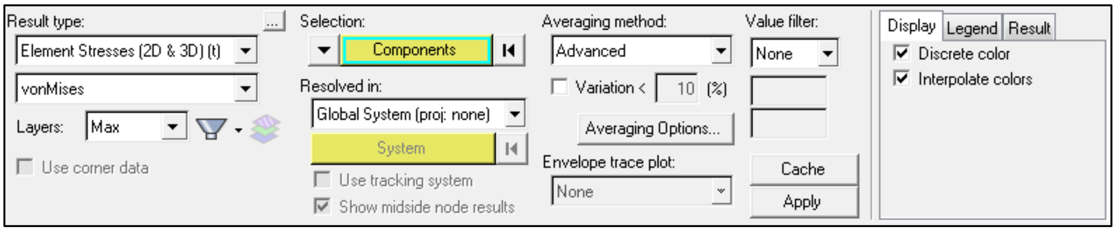


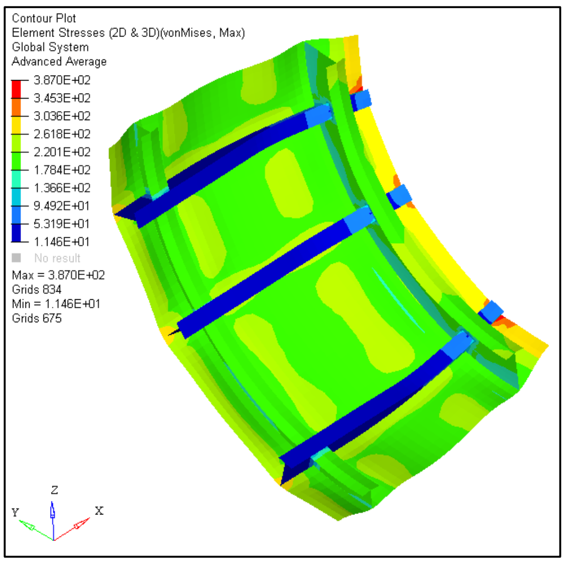
5. Change the **animation type** to ***Set Linear Animation Mode***.





6. Go to the **Contour Panel** .
7. Select the **Result Type** to be ***Element Stress 2D&3D (t)***.
8. Change **Averaging Method** to ***Advanced***.
9. Set **Display >** ***Interpolate Color***.
10. Click ***Apply***.





11. Click the animate icon .
12. Rotate the model to review it using the same keys and buttons as HyperMesh.

Step14: Engineering Review

1. Given that the Yield Strength of an HSLA Steel is around 360 MPa, do you think this structure, as designed, will survive a dive to a depth of 300 meters?
2. Using the Card Editing functions, experiment with thickness values to determine how the changes affect the stress and deformation of the model and achieve a model that does not exceed the yield strength.

   <u>NOTE:</u> The more weight of the structure, the less weight that can go in it so try to keep the materials as thin as possible.