# **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.

User Profiles Customize user interface: HyperMesh • Application: Default (HyperMesh) RADIOSS Block140 OptiStruct Abaqus Standard3D Actran Ansys Exodus C LsDyna Keyword971\_R6.1 Madymo Marc Marc3D Nastrar NastranMSC Pamcrash amcrash2G2012 Permas Samcef Always show at start-up OK Cancel

**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.

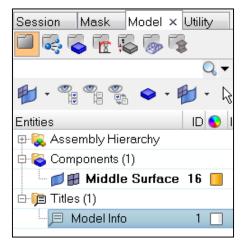
For this analysis, the Millimeter-Ton-Second scheme will be utilized.

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.

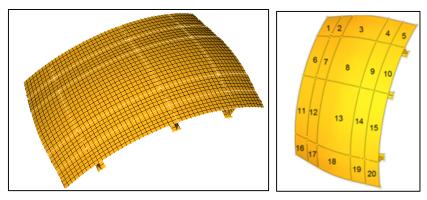


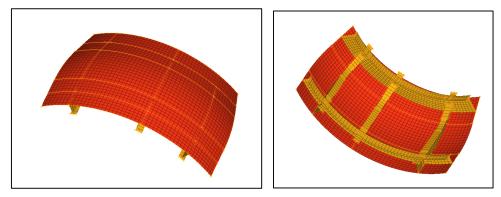
Component collectors are, as previously discussed, used for model organization. One of the most logical organization schemes for this model would be a component for the Hull elements and then another for the Ribs. This, of course, is only one method and could be altered for any number of organizational 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 so r 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<sup>2</sup> (MPa) and the Density in Ton/mm<sup>3</sup>. 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

	_
odel × Utility	
in 19	
🗢 - 📂 -	रे 📑
ID 😵 Inc	lude
chy	
16 🔲	0
17 🔳	0
_	
1 🕅	0
	0
1 🗖	Ω
	U
Value	
MAT1	
Steel	
1	
[Master Mode	I]
MAT1	
Hide In Menu/	Export
240000.0	
0.3	
7.85e-009	
	16 17 1 17 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1

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.* 

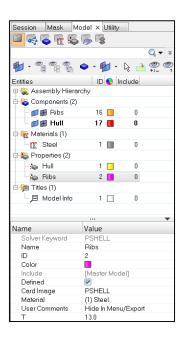
While the elements (quads and trias) have been created, they need to be defined as an entity the solver can analyze. In the case of OptiStruct, these 2D elements are defined as PSHELL. Creating the PSHELL property will give these elements their definition (card Image) and will allow for the definition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

Session Mask M	odel × Utility		Session Mask M	lodel × Utility	
🗖 🗟 🖉 🛱 🖏	in 12		🔲 🗟 🖏 🔂	, 🧓 🔹	
		Q 🕶 🗧			Q 🛨 🗧
	🗢 - 📂 - 🕻	} <u>→</u> ♥ ♥	₩ • ♥ ♥ ♥ •	🗢 - 📂 - 🔓	è 🧶 🦉
Entities	ID 😯 Inc	ude	Entities	ID 😮 Incluc	le
🕀 💫 Assembly Hierar	rchy		🕀 🙀 Assembly Hiera	ırchy	
🖶 🛜 Components (2)			🖹 🛱 🛜 Components (2)		
🚽 🖪 Ribs	16 📒	0	🚽 💋 📆 Ribs	16 📃	0
🚽 🗾 🛃 Hull	17 📕	0	🚽 🗾 🚮 Hull	17 📕	0
🕀 🙀 Materials (1)			🕀 🙀 Materials (1)		
Tr Steel	1	0	🔤 🍸 Steel	1 🔲	0
🕀 🌇 Properties (1)			🕀 😂 Properties (1)		
Hull	1 🗖	0	Hull	1 🔲	0
🖻 📋 Titles (1)		-	🖻 🗊 Titles (1)	_	
□ □ □ Model Info	1	0	🗐 🗐 🗐 🗐	1 🗔	0
		Ū			
		•			-
Name	Value		Name	Value	
Solver Keyword	PSHELL		Solver Keyword	PSHELL	
Name	Hull		Name	Hull	
ID	1		ID	1	
Color			Color		
Include	[Master Mode	]	Include	[Master Model]	
Defined			Defined	PSHELL	
Card Image Material	PSHELL Materia	al 📑 🛃 📢	Card Image Material	(1) Steel	
User Comments	Hide In Menu/		User Comments	Hide In Menu/Ex	nort
T	19.0	Export	Т	19.0	pon
🛆 Select Material			×		
Enter Search String			Q. <del>-</del>		
Name ID	Color	Card Image	Defined		
Steel 1		MAT1			

- 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.

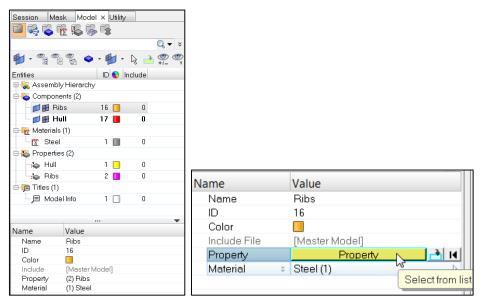


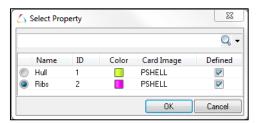
## Step 5: Property and Material Assignment.

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.

Session Mask Mode	I × Utilit∨				
~~ <		Q <del>-</del> *			
🗗 - ": ": ": ": 😜	- 🛃 -	•			
Entities	ID 😵 In				
🕀 💫 Assembly Hierarchy					
🕀 💫 Components (2)					
– 💋 🗭 Ribs	16 📃	0			
🖉 🔀 Hull	17 📕	0			
🖻 🙀 Materials (1)					
E Steel	1 🔲	0			
🖻 😂 Properties (2)					
Hull	1	0	L	2 × 1	
l‰ Ribs ⊡-1j⊟ Titles (1)	2 🔲	0	Name	Value	
□-□ Model Info	1	0	Name	Hull	
<u> </u>		-			
		•	ID	17	
Name Value			Color		
Name Hull ID 17			Include File	[Master Model]	
Color 📕			Property	Property	M 🛃
Include [Master M Property (1) Hull	lodelj		Material		
Material (1) Steel			Material	Steel (1)	
			u		
Select Property			23		
			Q, -		
Name ID	Color	Card Image	Defined		
Hull 1		PSHELL			

# **Step 6: Load Collector Creation**

PSHELL

OK

1. From the Model Browser, click right mouse button to create a LoadCollector.

Cancel

- 2. The Entity Editor will open and name it Pressure.
- 3. Assign it a unique color

2

Ribs

- 4. Leave the Card image as <None>.
- 5. Follow the previous steps to create another LoadCollector called Constraints.

Session Mask	Model × Utility		Session Mask	Model × Util	ity	
🗖 🗟 🖗	S 🖗 🕏		🔲 🗟 🖗 🖻	to 💿 😨		
		Q 🛨 🗧				Q, - ×
🔁 - 📲 📲	🖇 🗢 • 📂 •	k 🎽 🖤 🥙	🖶 - 📲 📲	🗞 🗢 - 📂	- 🗟 🏓	() +/_ ()
Entities	ID 😵 In	clude 🔄 📩	Entities	ID 📀	Include	-
🕀 💫 Assembly Hi	ierarchy		🖽 💫 Assembly H	lierarchy		
🖨 😂 Components	(2)		🖨 🛜 Component	s (2)		
🚽 🗾 🖪 Ribs	16 📃	0 =	— 💋 🖽 Ribs	16 📒	0	=
🚽 🗾 🛃 Hull	17 📕	0	🚽 🗾 🖽 Hull	17 📕	0	
🖻 🖳 Load Collect	ors (2)		🕀 🚱 Load Collec	tors (2)		
Press	ure 1 🔲	0	🚽 💋 🖪 Press	sure 1 🚺	0	
🖉 🛃 Cons	traints 2 📘	0	📈 🗾 🖽 Cons	straints 🛛 2 📘	) 0	
🕀 🙀 Materials (1)			🕀 🙀 Materials (1)	)		
🔤 🈰 Steel	1 🔳	0	🔤 🏦 Steel	1 🔲	) 0	
· · ·		+				
Name	Value		Name	Value		
Name	Pressure		Name	Constraints		
ID	1		ID	2		
Color			Color			
Include	[Master Model]		Include	[Master Mode	l]	
Card Image	<none></none>		Card Image	<none></none>		

## 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

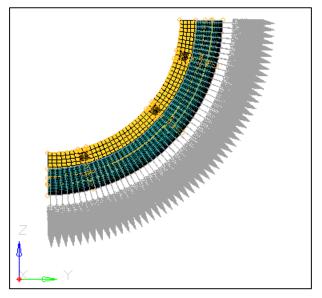
	and i	
• • • •		
ICON	100	

2. In the **elements** sub-panel select all of the elements in the **Hull** collector.

∉ elements ⊂ surfs	▼ comps I orientation:	♦ vector display size = 0.000	display adjust
	_ auto	🔽 display adjusted only	reverse
			return

#### 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.

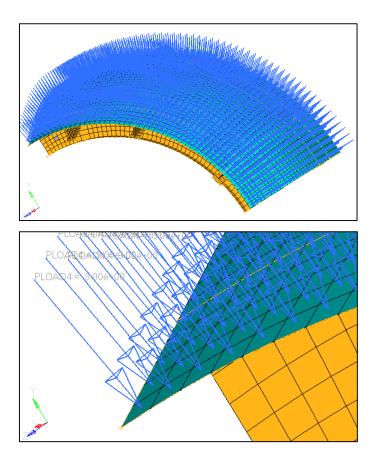
This option establishes the size of the arrow that will graphically represent the load. "Relative size" will make the arrow length the set percentage of the value of the load in model units. For example in our case of a 3.0 magnitude load, a relative size= value of 200 would result in a load arrow of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

<ul><li>create</li><li>⊂ update</li></ul>	• •	elems	I	♦ label load	uniform size = ds	200.000	create create/edit reject
• •	magnitude = normal	- 3 . 0	0 0	\$	nodes on face: face angle =		review
					load types =	PLOAD	return

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).

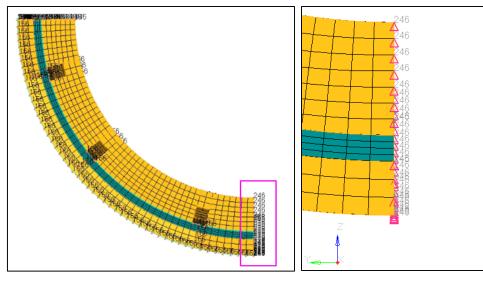
In the case of this model, a special constraining system called Symmetric Constraining is used. This is a common practice when analyzing a part with some form of symmetry. In the case of this Submarine Hull model, it represents ¼ of the complete hull circle. Analyzing only part of a symmetric model saves 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**.



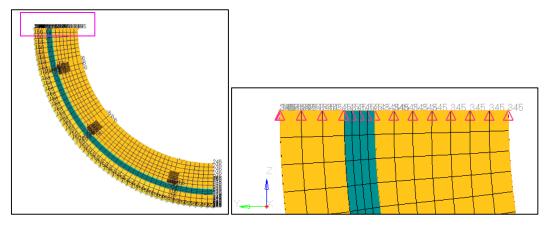
6. Using a box select (*HINT:* Shift+Left mouse button Drag a box) to pick the nodes shown in the image below.



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

create	• •	nodes	Κ			∣ dof1	-	0.000	create
C update						🔲 dof2	-	0.000	create/edit
		relative size =		20.000 🕅 labe	el constraint	s🔽 dof3	=	0.000	reject
	\$	constant value		fixed	ł	🔽 dof4	=	0.000	review
			_			🔽 dof5	-	0.000	
						🕅 dof6	=	0.000	
					lo	oad types =		SPC	return

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

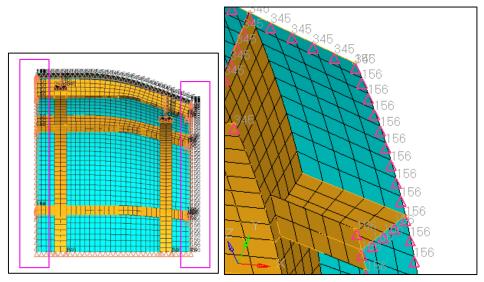


- 10. Click *create*.
- 11. Select the XY Top Plane View L.
- 12. Select and de-select the appropriate check boxes so that the only **DOFs** selected are **1**, **5** and **6**.

create	• •	nodes	н	🔽 dof1	-	0.000	create
C update				🔲 dof2	=	0.000	create/edit
		relative size =		20.000 Tabel constraints dof3	=	0.000	reject
	\$	constant value		🗌 fixed 🔲 dof4	=	0.000	review
			_	🔽 dof5	-	0.000	
				🔽 dof6	=	0.000	
				load types =		SPC	return

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".

Name	Value	•					
Solver Keyword	SUBCASE		۳.				
Name	pressure load		1	🔼 Select Loado	ol		×
ID	1						
Include File	[Master Model]						Q, <del>•</del>
User Comments	Hide In Menu/Export			Name	Id	Color	Card Image
Subcase Definition				Pressure	1		
🗆 Analysis type	Linear Static	=		Constraints	2		
SPC	Loadcol 🔁 🖂 📢						
LOAD	Pressure (1)						
SUPORT1	<unspecified> Select fro</unspecified>	om list	t			ОК	Cancel
PRETENSION	<unspecified></unspecified>						

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

Name     pressure load       D     1       nclude File     [Master Model]       Jser Comments     Hide In Menu/Export       Subcase Definition	Vame Value
D 1 nclude File [Master Model] Jser Comments Hide In Menu/Export Subcase Definition Analysis type Linear Static SPC Constraints (2)	Solver Keyword SUBCASE
nclude File [Master Model] Jser Comments Hide In Menu/Export Subcase Definition Analysis type Linear Static SPC Constraints (2)	Name pressure load
Jser Comments Hide In Menu/Export Subcase Definition  Analysis type Linear Static SPC Constraints (2)	ID 1
Subcase Definition           Analysis type         Linear Static           SPC         Constraints (2)	Include File [Master Model]
Analysis type Linear Static SPC Constraints (2)	User Comments Hide In Menu/Export
SPC Constraints (2)	Subcase Definition
	🖃 Analysis type 🛛 🛛 Linear Static
	SPC Constraints (2)
	LOAD Loadcol 💦 🎦 🖬
SUPORT1 * <unspecified></unspecified>	SUPORT1 * <unspecified></unspecified>
PRETENSION <unspecified> Select from list</unspecified>	PRETENSION <unspecified> Select fr</unspecified>

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.

Subcase Definition					
Analysis type	Linear Static				
SPC	Linear Static				
LOAD	Heat transfer (steady state)				
SUPORT1	Heat transfer (transient)				
DDETENSION	Normal modes				

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

Session Mask Model	×	Utility	y
🗖 🗟 🖕 🖗 🖏	5	2	
			0 <del>-</del> ×
4			
- • • • • • • • • • • • • • • • • • • •		Ŷ	• 🖶 • 🗟 🚵 💭 🖤
Entities	ID		Include
🕀 💫 Assembly Hierarchy			
🛱 🛜 Components (2)			
🚽 🗗 Ribs	16		0
🚽 🗗 Hull	17		0
🕀 🤑 Load Collectors (2)		_	
- 💋 🖽 Pressure	1		0
🚽 🗗 Constraints	2		0
🕀 🔂 Load Steps (1)		_	
👍 pressure load	1		0
🕀 🙀 Materials (1)			
Steel	1		0
🕀 🌇 Properties (2)			0
Hull	1		Π
i Ribs	2	-	0
	2		U
È-j≡ Titles (1)			
	1		0
			-
Name			Value
Solver Keyword			SUBCASE
Name			pressure load
ID			1
Include			[Master Model]
User Comments			Hide In Menu/Export
Subcase Definition			
Analysis type			Linear Static
SPC			(2) Constraints
LOAD			(1) Pressure

# 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.

🛆 Card Imag	ge	
	FORMAT_	
FORMAT	H3D	
FORMAT	HM	
	number_of_formats =	2
		_

💪 Card Image			X
		1	
ACMODL	CONTPRM	DMIGMOD	delete
ANALYSIS	DEBUG	DMIGNAME	disable
ASSIGN	DENSITY	DTI_UNITS	enable
B2GG	DENSRES	ECHO	
BULK_UNSUPPORTED_CAR	DESHIS	EIGVNAME	next
CASE_UNSUPPORTED_CAR	DESVARG	ELEMQUAL	
CHECK	DGLOBAL	FORMAT	return

5. Click *return* and then use **next** to find the **SCREEN** card.

# 6. Set the **SCREEN\_V1** to **OUT**

💪 Card Image	e
	SCREEN_V1
SCREEN	OUT

Card Image			2
OSDIAG	PFMODE	RESTARTW	delete
OSDIAG	PFMODE PFPANEL	RESTARTW	delete
OUTFILE	PFPANEL	RESULTS	disable
OUTFILE OUTPUT	PFPANEL PROPERTY	RESULTS SCREEN	disable
OUTFILE OUTPUT P2G	PFPANEL PROPERTY RADPRM	RESULTS SCREEN SENSITIVITY	disable enable

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

Session Mask Mode	I × Utili	У			Session Mask Mo	J × lab	Jtility				
🗖 🗟 🖕 🖕 🖏	, 🔁				🗖 🗟 🖧 🕵 (						
			Q, <del>-</del>	×						Q	• *
	Ş	- 🖶 -	k 🛁 🖤	<b>9</b> 1	- • • • • • •		🧼 •	•	$\searrow$	) (U) +/-	2 🖤
Entities	ID 😒	Include		-	Entities	ID	😵 Incl	ude			-
🕀 💫 Assembly Hierarchy					🖽 💫 Assembly Hierarc	hy					
🖶 🐻 Cards (2)					🛱 🐻 Cards (2)						
- 🍘 FORMAT	1	0			- 🍘 FORMAT	1		0			
SCREEN	2	0			CREEN	2		0			
🖶 🛜 Components (2)					🕀 🛜 Components (2)						
- 💋 🗭 Ribs	16 📃	0			– 🗾 🖪 Ribs	16		0			
🗖 🗭 Hull	17 📕	0			🗖 🗾 🖪 Hull	17		0			
🖶 🙀 Load Collectors (2)				Ξ	🖯 👯 Load Collectors (2	)					Ξ
– 💋 🗭 Pressure	1 🔲	0			🚽 🗾 🔗 Pressure	1		0			
🚽 🗭 Constraints	2 📃	0			🖉 🗗 Constraint	s 2		0			
🖶 🔂 Load Steps (1)					🕀 🔂 Load Steps (1)						
🚽 🖕 pressure load	1	0			🚽 🖕 pressure load	1		0			
🛱 🙀 Materials (1)					🛱 🙀 Materials (1)						
🔤 🌋 Steel	1 🔲	0			🔤 🈰 Steel	1		0			
🕂 😂 Properties (2)					🖯 😂 Properties (2)						
🛬 Hull	1 📃	0			- 🏣 Hull	1		0			
illi Ribs	2 📒	0		-	👆 🖕 Ribs	2		0			-
				•							•
Name	Valu		_			Value					
Include Status	[Ma	ster Model				SCREE		1			
Status number_of_formats =	2					[Master 🔽	Model	]			
Data: FORMAT_V1	2					<b>⊻</b> OUT					

## 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.

input file: << / 0 7 a - A	NALYSIS-SETUP-C	DPTISTRUCT-MR.fem	save as	OptiStruct
export options:	run options:	memory options:		HyperView
▼ custom	▼ optimization	♦ memory default		
				view .out
🔲 include connectors	options			return

- 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.

7a-ANALYSIS-SETUP-OPTISTRUCT-LB-FINAL.fem - HyperWorks	s Solver View	
Solver: optistruct_2017.2_win64.exe		
Input file: 7a-ANALYSIS-SETUP-OPTISTRUCT-LB-FINAL.fe Jo	b completed	
•		
Run command:/hwsolver.tcl -solver OS -screen/7a-ANALYSIS		B-FINAL fem
Message log:	Optimization s	·
Messages for the job:	Iteration	Subcase 🔺
ANALYSIS COMPLETED.	0	1 MaxDisp
	▼	<b>T</b>
		4
Run summary:		
***************************************	****** Find:	
**		=
** OptiStruct 2017	.2	
**		
** Advanced Engineering Analys		_
** Optimization Software from Altai	r Engineering,	Inc.
**		
** Windows 7 SP1 (Build 7601	) ITADSLAP35	
** 8 CPU: Intel(R) Core(TM) i7-47		OGHz
** 4162 MB RAM, 16188	MB swap	
** Build tag: 0870691 9433172 Ce64RBW8UH14	M-130911-2 400	0000004000
***************************************		****
** COPYRIGHT (C) 1996-2017	Altair Engine	
** All Rights Reserved. Copyright notice		
** Contains trade secrets of Altai	r Engineering,	Inc.
< [		•
ſ	Results View	Close
l	The startes	0.000

## Step 13: Post Processing

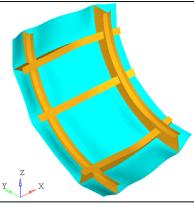
While the workings of HyperView will be discussed in greater length in the Post Processing section of the class, this step will cover basic post processing steps to review the analysis you just ran.

- In the HyperWorks Solver View dialog box, click the *Results* button to load the results in HyperView
- 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*.

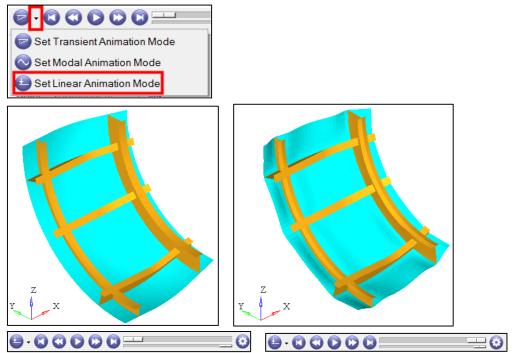
Load model and result	5.	
🔽 Load model	CH7-ANALYSIS-SETUP/ANALYSIS-RESULTS-DONE/07a-ANALYSIS-SETUP-OPTISTRUCT-MR.h3d	🔲 Overlay
🔽 Load results	CH7-ANALYSIS-SETUP/ANALYSIS-RESULTS-DONE/07a-ANALYSIS-SETUP-OPTISTRUCT-MR.h3d	
	Result-Math template: Standard   Reader Options	Apply

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

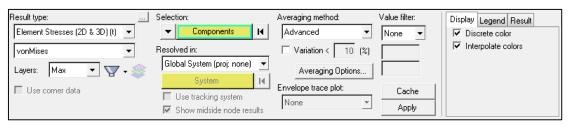
Deformed shape:			Resolved in:	Undeformed shape:			
Result type:	Displacement (v)	•	Global System (proj: none	•) 🔻	Show:	None	•
Scale:	Scale factor	•	System	M	Color:	Component	· 🔲
Type:	Uniform	•				ve with tracking	system
Value:	100						-)
			App	N I			

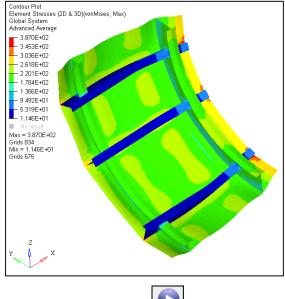


5. Change the **animation type** to **Set Linear Animation Mode**.



- 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.

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