

altairhyperworks.com

HM-4410: Setting Up a Model in ANSYS

In this tutorial, you will learn how to:

- Load the ANSYS user profile
- Retrieve the HyperMesh model files for this tutorial
- Add an element type
- Apply the real constants for the elements of the model
- Apply the material properties for elements of the model
- Update each component with respective element type
- Update each component with respective real constants
- Update each component with respective material properties

The model setup includes: setting up of element type, real constants, material properties and component structure in HyperMesh for ANSYS.

Model Files

This exercise uses the <code>chapter2_1.hm</code> and <code>chapter2_2.hm</code> files, which can be found in <hm.zip>/interfaces/ansys/. Copy the file(s) from this directory to your working directory.

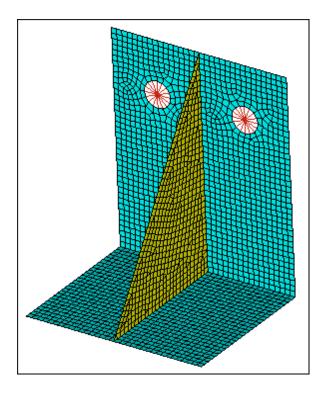
Exercise 1

Step 1: Load the ANSYS user profile

- 1. Start HyperMesh Desktop.
- 2. In the User Profile dialog, select Ansys.

Step 2: Retrieve the model file

- Open a model file by clicking *File* > *Open* > *Model* from the menu bar, or clicking so the **Standard** toolbar.
- 2. In the **Open Model** dialog, open the chapter2_1.hm file.
- 3. If your model's elements and mesh lines are not shaded, click $\widehat{\Psi}$ on the **Visualization** toolbar.



Step 3: Add an element type

1. In the **Model** browser, right-click and select *Create* > *Sensor* from the context menu. HyperMesh creates and opens a sensor (Et Type) in the **Entity Editor**.

Note: The **Entity Editor** displays the new Sensor's (Et Type) card details.

Entities	ID	
🕀 💫 Assembly Hierarchy		
🗄 🔞 Card (1)		
🗄 🍣 Component (4)		
🕀 🧊 Title (1)		
🖹 👼 Sensor (1)		
🛱 sensor1	1	
Name		 Value
Name		sensor1
ID		1
Color		
Element Type		SHELL181
KeyOpt1		
KeyOpt3		
KeyOpt5		
KeyOpt8		
KeyOpt9		
KeyOpt10		

- 2. For **Name**, enter a new name for the Et Type.
- 3. Optional: For **ID**, enter a new ID for the Et Type.

Note: By default, HyperMesh sets the ID to 1. If you create a new Et Type, HyperMesh will set the ID to n+1.



- 4. For **Element Type**, select a new element type.
 - **Note:** By default, HyperMesh set the **Element Type** to **SHELL181**. The elements in this model are of type SHELL181, therefore you do not need to change the element type for this tutorial.

Name	Value
Name	sensor1
ID	1
Color	
Element Type	SHELL181
KeyOpt1	SHELL163
KeyOpt3	SHELL181
KeyOpt5	SHELL208
KeyOpt8	SHELL209
KeyOpt9	SHELL28
KeyOpt10	SHELL281
	SHELL41
	SHELL43
	SHELL51
	SHELL57 -

SHELL181

- Suitable for analyzing thin to moderately-thick shell structures. It is a 4-node element with six degrees of freedom at each node: translations in the x, y, and z direction; rotations about the x, y, and z axes (if the membrane option is used, the element has translational degrees of freedom only). The degenerate triangular option should only be used as filler elements in mesh generation.
- Well-suited for linear, large rotation, and/or large strain nonlinear applications. Change in shell thickness is accounted for nonlinear analysis. In the element domain, both full and reduced integration schemes are supported. SHELL181 accounts for follower (load stiffness) effects of distributed pressures.
- May be used for layered applications for modeling laminated composite shells or sandwich construction. The accuracy in modeling composite shells is governed by the first order shear deformation theory.
- Set the element stiffness (KeyOpt1), integration (KeyOpt3), layer data storage (KeyOpt8), thickness (KeyOpt9), and/or initial stress (KeyOpt10) options by selecting their corresponding checkboxes in the Value column. A value appears below each KeyOpt you selected.

Name	Value
Name	sensor1
ID	1
Color	
Element Type	SHELL181
🖃 KeyOpt1	
value	0-Bending and membrane stiffness
KeyOpt3	
KeyOpt5	
🖃 KeyOpt8	
value	0-Store data for bottom of bottom layer and top of top layer
KeyOpt9	
🖃 KeyOpt10	
value	0-No user subroutine to provide initial stress

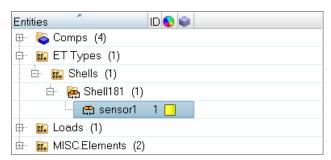


6. For each **KeyOpt** you selected, assign a **value**.

Note: For this tutorial, use the default value assigned to each KeyOpt.

Name	Value
Name	sensor1
ID	1
Color	
Element Type	SHELL181
🖃 KeyOpt1	
value	0-Bending and membrane stiffness
KeyOpt3	0-Bending and membrane stiffness 🛛 📐
KeyOpt5	1-Membrane stiffness only
🖃 KeyOpt8	2-Stress/strain evaluation only
value	0-Store data for bottom of bottom layer and top of top layer
KeyOpt9	
🖃 KeyOpt10	
value	0-No user subroutine to provide initial stress

- Open the Solver browser by clicking View > Browsers > HyperMesh > Solver from the menu bar.
- 8. Review the ET Type you just created.



Step 4: Define material properties

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

Entities	ID 😒
🕀 🧊 Title (1)	
🗄 🚰 Sensor (1)	
🖮 🍖 Material (1)	
🖹 material1	1 🗖
Name	Value
Name	material1
ID	1
Color	
Defined Entity	
Card Image	MPDATA
DifferentTem	
MPTEMP	1
Temp	
DENS	



- 2. For Name, enter Steel.
- 3. Optional: For **ID**, enter a new ID.

Note: By default, HyperMesh sets the ID to 1. If you create a new material, HyperMesh will set the ID to n+1.

4. Click the *Color* icon, and select a color.

5. Set Card Image to MATERIAL.

Name	Value
Name	Steel
ID	1
Color	
Defined Entity	
Card Image	MPDATA 🔽
DifferentTempTableForEachMatProps	<none></none>
MPTEMP	MATERIAL 📐
Temperature data	MPDATA
DENS	
EX	
NEXY	

- 6. Select the *EX* (Elastic moduli) checkbox.
- 7. For **MP_EX_LEN** (Number of Elastic moduli to input), enter 1.

Name	Value
Name	Steel
ID	1
Color	
Defined Entity	
Card Image	MATERIAL
■ MPTEMP	1
Temperature data	
DENS	
🗏 EX	
☐ MP_EX_LEN =	1
Data: C	
NUXY	
ALPX	

- 8. Under **MP_EX_LEN=**, next to **Data: C**, click 🔛.
- 9. In the **MP_EX_LEN=** dialog, enter 2.1e5.

/ MP_EX_LEN =	×
С	
1 2.1e5	
	Close

10. Click Close.



- 11. Select the **NUXY** (Minor Poisson's ratio) checkbox.
- 12. For **MP_NUXY_LEN** (Number of Minor Poisson's ratio to input), enter 1.
- 13. Under **MP_NUXY_LEN=**, next to **Data: C**, click **Sec.**
- 14. In the **MP_NUXY_LEN=** dialog, enter 0.3.
- 15. Click Close.
- 16. Go to the **Solver** browser and review the material you just created.

Step 5: Create the section card for the shell elements in the model

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

Entities	ID 💊	-
🗄 🙀 Materials (1)		
🛱 🕵 Properties (1)	=
broperty		
🕀 🚘 Sensors (1)		
Name	Value	
Name	property1	
ID	1	
Color		
Defined		=
Card Image	SHELL63p	
🖃 Real Constant	S	
ТКІ	0.0	
TKJ	0.0	
ТКК	0.0	
TKL	0.0	
EFS	0.0	
THETA	0.0	
RMI	0.0	
CTOP	0.0	
opot	~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~	

- 2. For Name, enter SECT1.
- 3. Optional: For **ID**, enter a new ID.

Ansys sections are supported under property, with a separate ID pool. Sections are created by setting the card image to SECTYPE. All cards created with the SECTYPE card image are organized under one ID pool. Properties created with a card image other than SECTYPE are organized under another ID pool. If you want to use the same IDs in these two pools, enable the **allow duplicate IDs** option in **Preferences** > **Meshing Options**. In earlier versions of HyperMesh (14.0 or before), Ansys sections were supported under beam section collectors.

Note: By default, HyperMesh sets the ID to 1. If you create another new beam section collector, HyperMesh will set the ID to n+1.

- 4. Click the *Color* icon, and select a new color.
- 5. Set Card Image to SECTYPE.
- 6. Set **TYPE** to **SHELL**.



7. Under SECDATA, for PLIES, enter 1.

Name	Value	-
Name	SECT1	
ID	1	
Color		
Defined		
Card Image	SECTYPE	=
TYPE	SHELL	
SECDATA		
PLIES	1	
Data: TK,		
SECOFFSET		
Location	MID	
SECCONTROLS		
E11	0.0	
E22	0.0	
====	<u></u>	

- 8. Under **PLIES**, next to **Data: TK**, click 🔛.
- 9. In the **Plies** dialog, enter 10 for **TK** (ply thickness).
- 10. For **MAT**, click **Unspecified** >> **Material**.

🛆 PLIES			×
TK	MAT	THETA	NUMPT
1 10.0	Material 🔂 📑 🛛	0.0	3
	*0		
			Close

11. In the **Select Material** dialog, select **Steel** and then click **OK**.

4	Select Ma	terial			×
Ent	er Search S	String			Q. •
	Name	Id	Color	Card Image	Defined
۲	Steel	1		MATERIAL	
				ОК	Cancel

- 12. For **THETA** (ply angle), keep the default value 0.0.
- 13. For **NUMPT** (Integration points through ply thickness), enter 3.0.
- 14. Click *Close*.

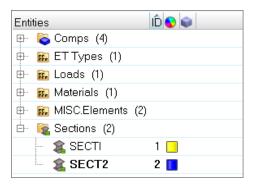


🛆 PLIES			×
ТК	MAT	THETA	NUMPT
1 10.0	Steel (1)	0.0	3
			Close

- 15. In the **Model** browser, **Property** folder, right-click on **SECT1** and select **Duplicate** from the context menu. HyperMesh creates a new property using the same card data as **SECT1**, except a different **Name** and **ID** are specified.
- 16. For Name, enter SECT2.
- 17. Click the **Color** icon, and select a new color.
- 18. Under PLIES, next to Data: TK, click 🔛.
- 19. In the **PLIES** dialog, change the value for **TK** from 10 to 5.
- 20. Leave MAT, THETA, and NUMPT unchanged.
- 21. Click Close.

🛆 PLIES			×
ТК	MAT	THETA	NUMPT
1 5.0	Steel (1)	0.0	3
			Close

22. Go to the **Solver** browser and review the two sections you just created.





Step 6: Update each component with the respective element type, property, material, and section information

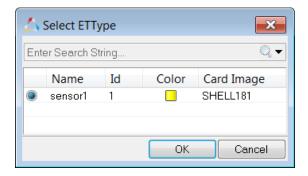
1. In the **Model** browser, **Component** folder, click **Base**. The **Entity Editor** opens and displays the component's corresponding data.

Entities	ID 😒		^
📄 💊 Component (4)			
🗾 🖪 CERIG	1 📕		=
- 🗾 🖽 Base	2 🗖		
🚽 🖪 Rib	3 🗖		
🗖 🗗 mass	4 🔳		
<u>і</u> . 🛱 Тана (1)			
Name		Value	
Name		Base	
ID		2	
Color			
Card Image		HM_COMP	
Туре		<unspecified></unspecified>	
Property		<unspecified></unspecified>	
Material		<unspecified></unspecified>	

2. For **Type**, click **Unspecified** >> **ETType**.

000	
Card Image	HM_COMP
Туре	ETType 🚬 📑 📢
Property	<unspecified> ^し</unspecified>
Motorial	d Inappositions

3. In the **Select ETType** dialog, select *sensor1* (SHELL181) and then click *OK*.



- 4. For **Property**, click **Unspecified** >> **Property**.
- 5. In the **Select Property** dialog, select **SECT1** and then click **OK**.

Note: You do not have to assign a **Property** or **Material** to this component, because this information is already defined in **SECT1**.



	🛆 Select Property						
Ente	er Search	String			Q -		
0	Name SECT1 SECT2	ID 1 2	Color	Card Image SECTYPE SECTYPE	Defined		
				ОК	Cancel		

- 6. In the **Model** browser, **Component** folder, click *Rib*. The **Entity Editor** opens and displays the component's corresponding data.
- 7. For **Type**, click **Unspecified** >> **ETType**.
- 8. In the **Select ETType** dialog, select *sensor1* (SHELL181) and then click *OK*.
- 9. For **Property**, click **Unspecified** >> **Property**.
- 10. In the **Select Property** dialog, select **SECT2** and then click **OK**.
 - **Note:** You do not have to assign a **Property** or **Material** to this component, because this information is already defined in **SECT2**.

Value	
Rib	
3	
HM_COMP	
sensor1 (1)	
SECT2 (2)	
<unspecified></unspecified>	
	Rib 3 HM_COMP sensor1 (1) SECT2 (2)

11. In the **Model** browser, **Component** folder, click *mass*. The **Entity Editor** opens and displays the component's corresponding data.

Note: The **mass** component does not currently have a type, property, or material attached to it.

 For Type, click Unspecified >> ETType. You need to attach the element type MASS21 to the mass component. This element type is not available in the Select ETType dialog because it does not exist in the model, therefore you need to create and attach it to the component.

MASS21

- A point element that can have up to six degrees of freedom: translations in the nodal x, y, and z directions; rotations about the nodal x, y, and z axes. A different mass and rotary inertia may be assigned to each coordinate direction.
- Defined by a single node, concentrate mass components (Force*Time²/Length) in the element coordinate directions and rotary inertias (Force*Length* Time²) about the element coordinate axes. The element coordinate system may be initially parallel to the global Cartesian coordinate system or to the nodal coordinate system (KEYOPT(2)). The coordinate system rotates with the nodal coordinate rotations



during a large deflection analysis. Options are available to exclude the rotary inertia effects and to reduce the element to a 2-D capability (KEYOPT(3)). If the element requires only the mass input, it is assumed to act in all appropriate coordinate directions

- 13. Click *Cancel*.
- 14. Right-click on *Type* and select *Create* from the context menu. The **Create Sensors** dialog opens.

Name		Value	9	
Name		mass	:	
ID		4		
Color				
Card Imag	ge		COMP	
Туре 🗖		ZLInc	pacified>	
Propert	Create		cified>	
Materia	Edit	~8	cified>	
	Show			
	Hide			
	Isolate			
	XRef entities			
•	 Filter entities 			
	 Warn upon er 	ntity type change		

15. Set Element Type to MASS21.

16. Set KeyOpt3 to 3-D mass without rotary inertia.

17. Click *Close*. HyperMesh creates and attaches the new sensor to the **mass** component.

💪 Create Sensors	×
Name	Value
Name	sensor2
ID	2
Color	
Element Type	MASS21
KeyOpt1	
KeyOpt2	
🗏 KeyOpt3	3-D mass without rotary inertia
value	2
1	Close

18. Create a property card that associates a small mass to the mass elements by rightclicking on **Property** and selecting **Create** from the context menu.

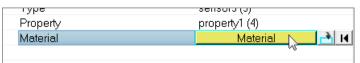


Name	Value	
Name	mass	
ID	4	
Color	E.	
Card Imag	e HM_COMP	
Туре	sensor2 (2)	
Property	0	
Material	Create	
	Edit	
	Show	
	Hide	
	Isolate Only	
	XRef entities	
	✓ Filter entities	
	Warn upon entity type change	

- 19. In the **Create Properties** dialog, the **Card Image** is automatically set to **MASS21p** because the element type attached to the **mass** component is **MASS21**.
- 20. Set KeyOpt3 to 3-D mass without rotary inertia.
- 21. Under Real Constants, enter 0.001 for MASS.
- 22. Click *Close*. HyperMesh creates and attaches the new property to the **mass** component.

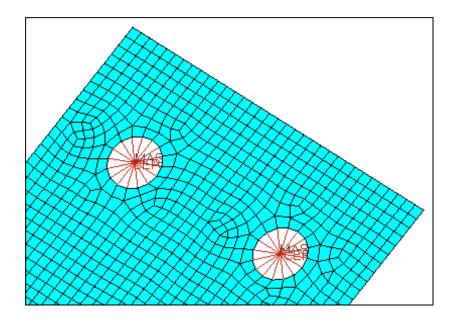
💪 Create Properties			
Name	Value		
Name	property1		
ID	4		
Color			
Defined Entity			
Card Image	MASS21p		
kopt3	3-D mass without rotary inertia		
🖃 Real Constants			
MASS	0.001		
	Close		

23. For Material, click *Unspecified* >> *Material*.



- 24. In the Select Material dialog, select Steel and then click OK.
- 25. The component **CERIG** contains ANSYS rigid elements. These elements define the rigid region and do not require an element type, property, or material, therefore you do not have to assign a card to this component.





Step 7: Save your model

- 1. From the menu bar, click *File* > *Save As* > *Model*.
- 2. In the **Save Model As** dialog, navigate to your working directory and save the file.
- 3. To apply boundary conditions and create load steps for your model, proceed to **Exercise 2**.

Exercise 2

Introduction to ANSYS Load Steps

This exercise introduces the concept of ANSYS load steps in HyperMesh. In HyperMesh, you need to have each load or constraints in a separate load collector (load cols). With the help of these load collectors, you can create multiple load steps depending on the requirement. The combination of loads with constraints, form a load step. If you have created load steps in your model, the exported *.cdb file will have all of the load step information. This *.cdb file when imported into ANSYS, automatically creates the *.so files in the working directory which can be used later if needed.

In this tutorial, you will learn how to:

- Load the ANSYS user profile.
- Retrieve the HyperMesh model file for this tutorial.
- Create constraint load collectors.
- Apply the constraints to the model.
- Apply the force on mass elements with force1 load collector.
- Apply the force on mass elements with force2 load collector.
- Apply the force on mass elements with force3 load collector.



- Create multiple load steps.
- Add /SOLU & LSSOLVE in control cards
- Export the deck to ANSYS *.cdb format

Optional - Step 1: Load the ANSYS user profile

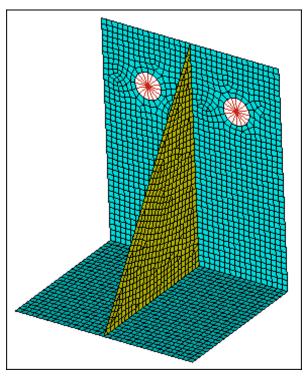
You only need to perform this step, if you did not complete Exercise 1.

- 1. Start HyperMesh.
- 2. In the **User Profile** dialog, set the user profile to **Ansys**.

Optional - Step 2: Retrieve the HyperMesh model file

You only need to perform this step, if you did not complete Exercise 1.

- 1. From the menu bar, click *File* > *Open* > *Model*.
- 2. In the **Open Model** dialog, open the <code>chapter2_2.hm</code> file.
- 3. If you model's elements and mesh lines are not shaded, click $\widehat{\Psi}$ on the **Visualization** toolbar.



Step 3: Create a constraints load collector

- 1. In the **Model** browser, right-click and select *Create* > *Load Collector* from the context menu. HyperMesh creates and opens a load collector in the **Entity Editor**.
- 2. For Name, enter constraints.



- 3. Click the *Color* icon, and select a new color for the load collector.
- 4. Create three more load collectors labeled **force1**, **force2**, and **force3**.

Entities	ID	
🕀 🌆 Material (1)		
🕀 🍓 Beam Section Collector (2)		
🕀 😂 Property (1)		
🖻 辑 Load Collector (4)		
📁 🗾 🖽 constraints	1	
🗂 🗾 🖽 force1	2	
🗂 🗾 🖽 force2	3	
🗖 🗗 force3	4	

Step 4: Apply the constraints to the model

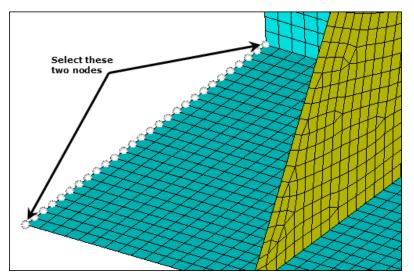
1. In the **Model** browser, **Load Collector** folder, right-click on **constraints** and select **Make Current** from the context menu.

Note: When new loads are created, Hypermesh will place them in this collector.

- 2. Open the **Constraints** panel by clicking **BCs** > **Create** > **Constraints** from the menu bar.
- 3. Select all of the *dof* (degree of freedom) checkboxes.

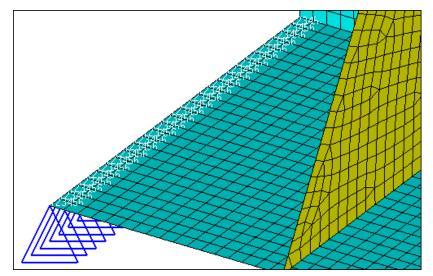
create	▼ nodes I	🔽 dof1	=	0.000	create
C update		🔽 dof2	=	0.000	create/edit
	size = 10.000	🔽 dof3	=	0.000	reject
	label constraints	🔽 dof4	=	0.000	review
		🔽 dof5	=	0.000	
	♦ constant value	🔽 dof6	=	0.000	
		load type	3S =	D_CONSTRNT	return

- 4. Click *nodes* >> *by path*.
- 5. Select a starting node and an end node on the left side of the model as indicated in the following image.

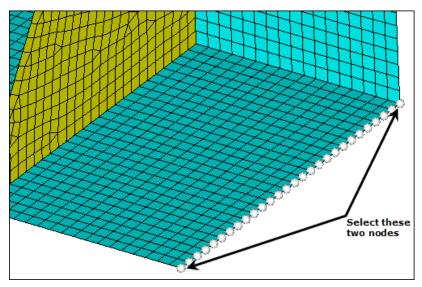




6. Click *create*.

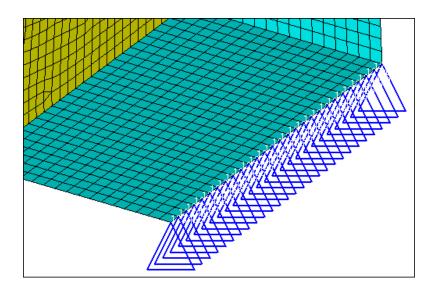


7. Repeat steps 4.4 and 4.5 to select a starting node and an end node on the right side of the model as indicated in the following image.

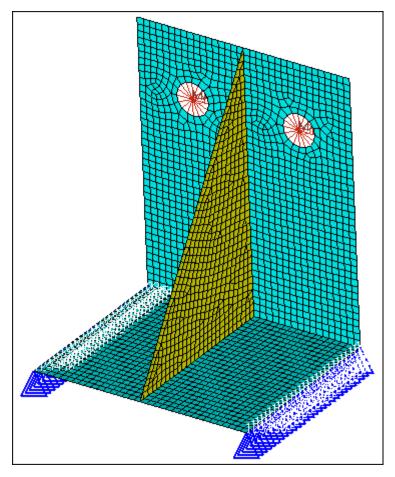


8. Click *create*.





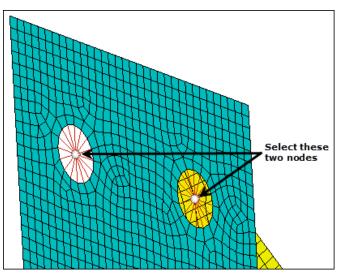
9. Click *return* to exit the **Constraints** panel.



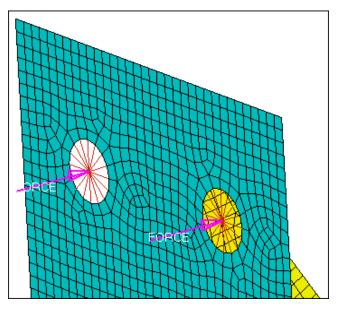


Step 5: Apply the force on mass elements with the force1 load collector

- 1. In the **Model** browser, **Load Collector** folder, right-click on **force1** and select **Make** *Current* from the context menu.
- 2. Open the **Forces** panel by clicking **BCs** > **Create** > **Forces** from the menu bar.
- 3. Verify that the entity selector is set to *nodes*.
- 4. Select the two nodes in the center of the two bolt holes as indicated in the following image.

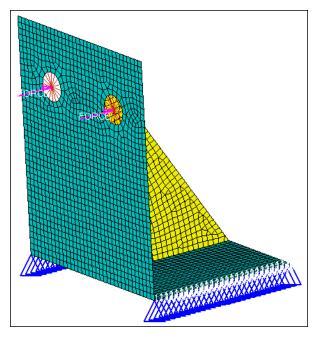


- 5. In the **magnitude**= field, enter 500.
- 6. Set the orientation selector to **z-axis** for the direction of application of the force.
- 7. In the **uniform size=** field, enter 20.
- 8. Click *create*.



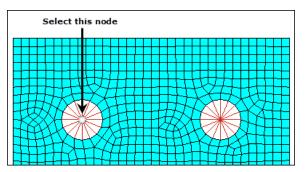


9. Click *return* to exit the **Forces** panel.



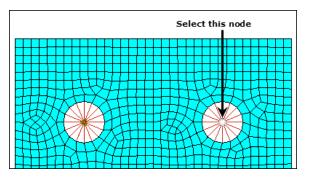
Step 6: Apply the force on mass elements with the force2 load collector

- 1. In the **Model** browser, **Load Collector** folder, right-click on **force2** and select **Make** *Current* from the context menu.
- 2. For better visualization, press **F5** to open the **Mask** panel.
- 3. Set the entity selector to *loads*.
- 4. Select the two forces you created in step 5.8.
- 5. Click *mask*.
- 6. Click *return*.
- 7. Open the **Forces** panel.
- 8. Verify that the entity selector is set to *nodes*.
- 9. On the **Standard Views** toolbar, click \coprod .
- 10. Select the left side node in the center of the bolt hole as indicated in the following image.

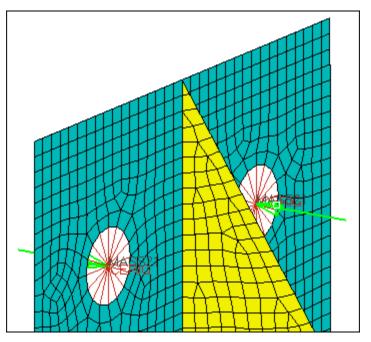




- 11. In the **magnitude=** field, enter 500.
- 12. Set the orientation selector to **z-axis** for the direction of application of the force.
- 13. Click *create*.
- 14. Select the right side node in the center of the bolt hole as indicated in the following image.



- 15. In the **magnitude=** field, enter -500.
- 16. Set the orientation selector to **z-axis** for the direction of application of the force.
- 17. Click *create*.



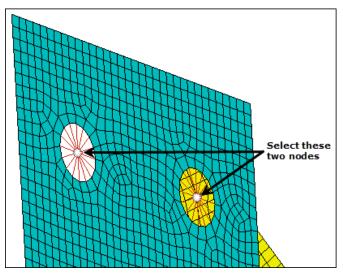
18. Click *return* to exit the Forces panel.

Step 7: Apply the force on mass elements with the force3 load collector

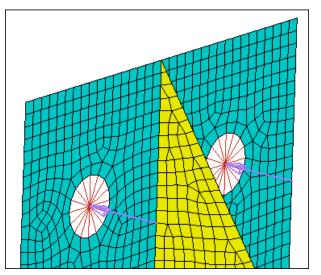
1. In the **Model** browser, **Load Collector** folder, right-click on **force3** and select **Make** *Current* from the context menu.



- 2. Open the **Mask** panel.
- 3. Verify that the entity selector is set to **loads**.
- 4. Select the two forces you created in steps 6.13 and 6.17.
- 5. Click *mask*.
- 6. Click *return*.
- 7. Open the *Forces* panel.
- 8. Verify that the entity selector is set to *nodes*.
- 9. Select the two nodes in the center of the two bolt holes as indicated in the following image.

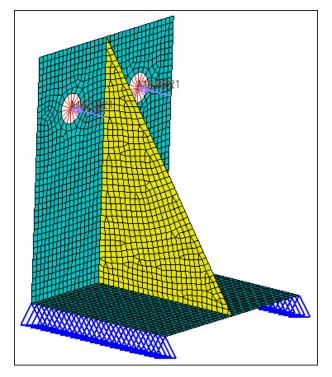


- 10. In the magnitude= field, enter -500.
- 11. Set the orientation selector to **z-axis** for the direction of application of the force.
- 12. Click *create*.





13. Click *return* to exit the **Forces** panel.



Step 8: Create multiple load steps

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

Entities	ID 😒	*
🕀 😂 Property (1)		
🖽 👫 Load Collector (4)		
🖹 📬 Load Step (1)		=
📥 👍 loadstep1	1	
Name	Value	
Name	loadstep1	
ID	1	
Card Image	<none></none>	
Loadcol IDs	0 Loadcols	
/TITLE		
ACEL		
CECMOD		

- 2. For Name, enter Step1.
- 3. For Loadcol IDs, click *0 Loadcols* >> Loadcols.

Card Image	<none></none>
Loadcol IDs	Loadcols 🔂 📑 📢
/TITLE	



4. In the **Select Loadcols** dialog, select *constraints* and *force1*.

🛃 Select Loadcols			×
Ent	er Search Strir	ig	Q, •
	Name	Id	Color
	constraints	1	
	force1	2	
	force2	3	
	force3	4	
			2 selected.
		ОК	Cancel

- 5. Click **OK**.
- 6. Create a second load step labeled **Step2**, and assign it the load collectors **constraint** and **force2**.
- 7. Create a third load step labeled **Step3**, and assign it the load collectors **constraint** and **force3**.
- 8. In the **Model** browser, review the **Load Collectors** and **Load Steps** you created.

Entities	ID 😒	
🗐 👫 Load Collector (4)		
🚽 🗗 constraints	1 🗖	
🚽 🗗 force1	2 🗖	
🚽 🗗 force2	3 🗖	
🚽 🗗 force3	4 🔲	
🖹 🔂 Load Step (3)		
🚽 📥 Step1	1	
🚽 📥 Step2	2	
🗆 📥 Step3	3	

- Open the Solver browser by clicking View > Browsers > HyperMesh > Solver from the menu bar.
- 10. Review the Load Collectors and Load Steps you created.

Step 9: Add /SOLU, ANTYPE, and LSSOLVE in the control cards

- Open the **Control Cards** panel by clicking *Setup* > *Create* > *Control Cards* from the menu bar.
- 2. In the **Card Image**, click **/SOLU** to exit the **PREP7 preprocessor** and enter the **SOLU preprocessor**.



-					
	LUMPM	MODOPT	OUTRES	ETABLE	AUTOTS
	ACEL	MXPAND	/POST1	KBC	/BATCH
	CGLOC	EQSLV	PRESOL	LNSRCH	BFUNIF
\rightarrow	CGOMGA	ALPHAD	RSYS	MODE	/COM
	CMDOMEGA	BETAD	/SOLU	NEQIT	CNVTOL
	CMOMEGA	PSTRES	SOLU	NLGEOM	DELTIM
	DCGOMG	EXPASS	ANTYPE	NSUBST	DOF
1					

```
! Exit PREP7 processor
FINISH
! Enter SOLU processor
/SOLU
```

3. Click *return*.

4. Because you are solving the model for static analysis, click **ANTYPE**.

AUTOTS	ETABLE	OUTRES	MODOPT	LUMPM
/BATCH	KBC	/POST1	MXPAND	ACEL
BFUNIF	LNSRCH	PRESOL	EQSLV	CGLOC
/COM	MODE	RSYS	ALPHAD	CGOMGA
CNVTOL	NEQIT	/SOLU	BETAD	CMDOMEGA
DELTIM	NLGEOM	SOLU	PSTRES	CMOMEGA
DOF	NSUBST	ANTYPE	EXPASS	DCGOMG

5. Set **type** to **STATIC** and **status** to **NEW**.



6. Click *return*.

7. Click *LSSOLVE*.

Tip: If you do not see the **LSSOLVE** control card, click *next*.

SUBOPT	SOLVE	UNSU_PREP_END	
EMUNIT	LSSOLVE	MXPAND ACEL	
EORIENT	STEF SOL	EQSLV CGLOC	
UNSU_PREP7	HEMIOPT	ALPHAD CODMCA	
UNSU_END	RADOPT	BETAD CMDDMEGA	
UNITS IN	SPCTEMP	PSTRES CMOMEGAN	
NUMOFF	UNSU_PREP_MID	EXPASS DCD.0MG	

- 8. Set the minimum number of load steps by entering 1 in the **LSMIN** field.
- 9. Set the maximum number of load steps by entering 3 in the LSMAX field.



10. Set the load step increment by entering 1 in the $\ensuremath{\text{LSINC}}$ field.

LSSOLVE 1 SSOLVE 1	
User Comments ▼ Do Not Export	reject default
	abort return

This card image commands the solver to solve all three load steps.

- 11. Click *return* to exit the card image.
- 12. Click *return* to exit the **Control Cards** panel.

Step 10: Export the deck to ANSYS *.cdb format.

- 1. Open the **Export** tab by clicking *File* > *Export* > *Solver Deck* from the menu bar.
- 2. Set File type to Ansys.
 - **Note:** If you are in the **ANSYS** user profile, HyperMesh automatically sets the **File type** to Ansys and loads ANSYS as the default **Template**.
- 3. In the **File** field, navigate to your working directory and save the file as 4410_export.cdb.
- 4. Click *Export*.

