

altairhyperworks.com

HM-4330: Defining *STEP using Abaqus Step Manager

In this tutorial, you will learn how to:

- Load the Abaqus user profile
- Retrieve the HyperMesh model file
- Define the *STEP card and specify *STATIC as an analysis procedure
- Define loads (*CLOAD) and boundary conditions (*BOUNDARY)
- Define pressure loads (*DLOAD) with an element set
- Define output requests
- Export the database to an Abaqus input file

Model Files

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

Exercise

Step 1: Load the Abaqus user profile and the model

A set of standard user profiles is included in the HyperMesh installation. User profiles change the appearance of a panel, however they do not affect the internal behavior of each function.

Complete the steps below to load the Abaqus user profile and the model.

- 1. Start HyperMesh Desktop.
- 2. In the **User Profile** dialog, set the user profile to **Abaqus**, **Standard 3D**.
- 3. Open a model file by clicking *File* > *Open* > *Model* from the menu bar, or clicking key on the **Standard** toolbar.
- 4. In the **Open Model** dialog, open the <code>abaqus_StepManager_tutorial.hm</code> file.
 - **Note:** The abaqus_StepManager_tutorial.hm file contains pre-defined model data. Use this file in the following steps to define the history data portion of this model.

Step 2: Define a *STEP card and specify *STATIC as the analysis procedure

In this step, you will create a *STEP card with the *STATIC analysis procedure.

1. From the menu bar, click *Tools* > *Load Step Browser*. The **Step Manager** opens.



- 2. Click *New*. The **Create New Step** dialog opens.
- 3. In the Name field, enter step1.
- 4. Click *Create*. A step, labeled **step1**, opens the **Load Step** dialog.
- 5. In the first pane, select *Title*. The **Step heading** option with a disabled field is displayed.



6. Select the **Step heading** checkbox, and enter 100kN in the text field.

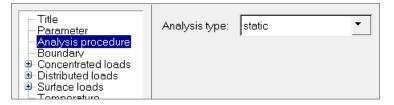


- 7. Click *Update* to store the heading information in **step1**.
- 8. In the first pane, select **Parameter**.
- 9. Select the **Name** and **Perturbation** checkboxes.

Note: Notice Name is already set to step1.

- Title - Parameter	🔽 Name:	step1	Increment:	0
−Analysis proced −Boundary ⊕-Concentrated loε	Amplitude	step 💌	🗖 Nigeom:	yes 💌
Distributed loads Surface loads	Extrapolation:	linear 💌	🗖 ConvertSdi:	yes 💌
 Temperature Inertia relief Interface controls 	Unsymmetric:	yes 💌	Perturbation:	

- 10. Click Update.
- 11. In the first pane, select *Analysis procedure*.
- 12. Set Analysis type to static



- 13. Click Update.
- 14. Click the **Dataline** tab.
- 15. Select the **Optional dataline** checkbox to add an additional dataline.



- 16. Add individual data, such as Initial increment, by selecting the appropriate checkbox and entering a value.
 - **Note:** When a checkbox is disabled, a space will be added in the ASCII file, and the Abaqus solver will use the default value.

Title Parameter	Analysis type:	static 💌	Parameter D	ataline Opt	ions
 Analysis procedure Boundary Concentrated loads Distributed loads 		,	🔽 Optional data	line	
B-Surface loads			🗖 Initial increm	ment:	0.0
-Inertia relief			🗖 Time perio	d:	0.0
B Output request Monitor			🗖 Minimum in	crement:	0.0
Adaptive Mesh Print File format			🗖 Maximum ii	ncrement:	0.0

17. Click Update.

Steps 3 - 6: Define the Loads (*CLOAD) and Boundary Conditions (*BOUNDARY)

In the following steps you will add the *CLOAD and *BOUNDARY keywords to the current load collector by defining loads and boundary conditions.

Step 3: Create constraints (*BOUNDARY)

- 1. In the first pane, select **Boundary**.
- 2. Click *New*. The **Create Load Collector** dialog opens.
- 3. In the Name field, enter loads_and_constraints.
- 4. Click *Create*.
- 5. Optional. In the **Load collector** table, **Display** column, click the color icon to select a color for the load collector.

Title	Load c	ollector:		
-Parameter	Status	Name	Display	
 Analysis proced Boundary 	•	loads_and_constraints	<u> </u>	
∃ Concentrated lot	Γ			
B Distributed loads B Surface loads	Γ			

6. Verify that the **Status** checkbox for **loads_and_constraints** is selected.

Note: By selecting this checkbox, you are adding this load collector into the loadstep.

7. Click the *loads_and_constraints* load collector. A set of new tabs displays.

8. From the **Define** tab, verify **Type** is set to **default(disp)**.

Load collector: loads_and_constraints	Define Delete Parameter
Status Name Display Image: Constraints Image: Constraints Image: Constraints Image: Constraints	Type: default (disp) Define boundary on: Image: Constraints on the sets Map Loads on Geometry View Loads

9. Click **Define from 'Constraints' panel**. The **Constraints** panel opens. Use this panel to create constraints.

Step 4: Create constraints from the Constraints panel

- 1. On the **Standard Views** toolbar, click $\stackrel{\checkmark}{=}$ (**XZ Right Plane view**).
- 2. In the **Constraints** panel, click *nodes* >> *by window*.
- 3. With the exception of the nodes at the ends of the cradle, draw a rectangle around all of the displayed nodes.



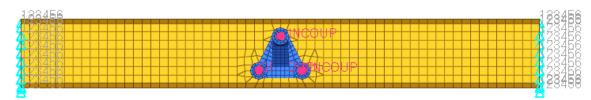
- 4. Select the *exterior* checkbox.
- 5. Click *select entities*. HyperMesh selects all of the nodes outside the window you drew.

253		Т								С,																																								(L.,				100
28	0	Т				Т			Г																	te	-	4		1	ų,	Ł.	du		51																			20
- 28	Э	Т		Т	Т	Т	Т		Г	Т			Т	Т	T			T	Т				Т		1	£		1	IJ	ų	д	Ρ	Ψ	T	1			Г					Г							Т	Т		Т	×
- 24	П	Т	T	Т	Т	Т	Т		Т	Т			Т	Т	Т	Т		T	Т	T		Γ	Т		À	28	F	ŧÇ	П				Т					Г		Γ	Γ	Г	Г							T	Т	T	Т	×
1	Э	Т		Т	Т	Т			Т	T			Т	Т	T				Т				Д	J	X			B	Ø.	7	1	Ζ	Т					Γ	Г		Γ	Γ	Г							T	Т		Т	X
- 26	а	Т	Т	Т	Т	Т	Т		Т	Т			Т	Т	Т	Т		T	Т	Т			4	C				F	5	¥.	4	1	ŀ	Ų	5				Т	Г	Г	Г	Г	Г						Г	Т	E	Т	X
- 25	С	Т	Т	Т	Т	Т			Т	Т			Т	Т	T	Т		T	Т	Т		Н	8		н	Ŧ	Ħ	Ŧ	신		ÿ.	Ņ	F	4	2	υ	Г	Г	Г	Г	Г	Г	Г							E	Т		Т	X
20	а	Т		Т	Т	Т			Т	T			Т	Т	I				Т			0	×	2	1		1	t	ĩ	₹	5	t	Т					Г					Г								Т			X
Sec.										<u>ار ا</u>													7							_																				(in 1997)				Sec.

6. Verify that all six dof checkboxs are selected.

🔽 dof1	=	0.000
🔽 dof2	=	0.000
🔽 dof3	=	0.000
🔽 dof4	=	0.000
🔽 dof5	=	0.000
🔽 dof6	=	0.000

7. Click *create*. HyperMesh creates constraints at the selected nodes.





- 8. Click *return* to go back to the **Load Step** dialog.
- 9. At the bottom of the **Load Step** dialog, look at the **Load type** line. Bc (short for BOUNDARY) appears on this line, which indicates step1 is a load type created in the load_and_constraints load collector. The corresponding load type in the first pane is also highlighted.

New.	Review Reset	Organize
Loa	id type: Bc	

Step 5: Create Forces (*CLOAD)

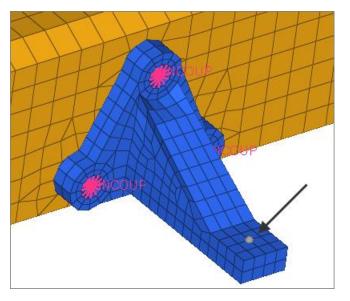
1. In the first pane of the **Load Step** dialog, expand *Concentrated loads*, and select *CLOAD-Force*. A new set of tabs displays.

Title	Load collector: loads_and_constraints	Define Delete Parameter
 Parameter Analysis procedure Boundary Concentrated loads CLOAD-Force 	Status Name Display Image: constraints Image: constraints Image: constraints Image: constraints	Define CLOAD-Force on:
CLOAD-Moment CFILM CRADIATE CRADIATE Distributed loads Surface loads Temperature Inertia relief Interface controls Output request		Node sets Define from 'Forces' Panel Map Loads on Geometry View Loads

2. From the **Define** tab, click **Define from 'Forces' Panel**. The **Forces** panel opens. Use this panel to create forces.

Step 6: Create forces from the Forces panel

1. Select the central node on the top side of the bracket arm.



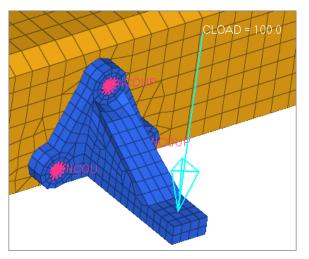
2. In the **Forces** panel, **magnitude=** field, enter -100.



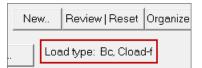
3. Set the orientation selector to *z-axis*.



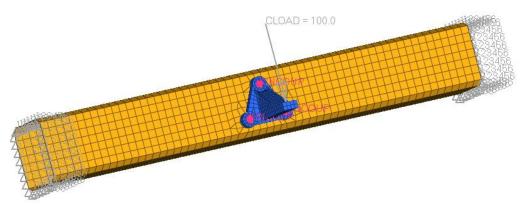
4. Click *create*.



- 5. Click *return* to go back to the **Load Step** dialog.
- 6. At the bottom of the **Load Step** dialog, notice the **Load type** now reads **Cload-f**, which indicates CLOAD-force as another load type created in the loads_and_constraints load collector. The corresponding load type in the first pane is highlighted.



7. Click **Review | Reset**. The constraints and forces that belong to the loads_and_constraints load collector highlight.



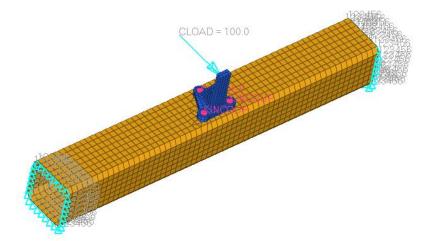


8. Revert the highlighted constraints and forces to the load collector color by right-clicking on *Review*.

Step 7: Define pressure loads (*DLOAD) with element set

In this step, you will create a *DLOAD pressure load and add it to the current load collector with an element set.

- 1. In the first pane of the **Load Step** dialog, expand *Distributed loads*, and select *DLOAD*. A new set of tabs displays.
- 2. From the **Define** tab, set **Define DLOAD on** to *Element sets*. The element sets table displays.
- 3. Rotate the model to the view as shown in the image below.

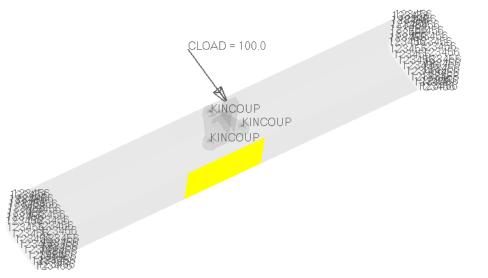


- 4. From the **Load Step** dialog, set **Type** to *default (Pressure)*.
- 5. Set **Element sets** to *pressure_set*.
 - Tip: To view the entire list of element sets, click . Use **Filter** and **Sort** to narrow your search.
- 6. Click the right arrow to add the selected set to the element sets table.

Define Delete Parameter]				
Turner alafault (Dranaura)	Elset	Label	Magnitude	Load Id	A
Type: default (Pressure) 💌	pressure_set	P1 -			_
Define DLOAD on:					
 Elements or geometry 					
 Element sets 					
pressure_set 💌					
	·				
Review Reset Set					
Create/Edit set.					
			-		
Disular /Day involvement				1	
Display/Review from panel					
					_
	4				<u> </u>
	4				P
	Show face	s	Review I	Reset Updati	e
				1	



7. Under Element sets, click *Review* | *Reset Set*. The element set highlights.

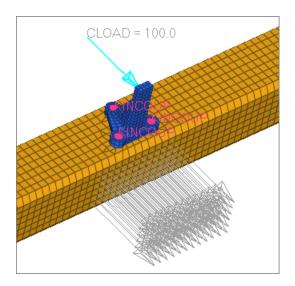


- 9. Revert the load collector back to its original color by right-clicking on *Review* | *Reset Set*.
- 10. In the element sets table, **Label** column, select **P** for the newly added **pressure_set**.

Define Delete Parameter]			
Type: default (Pressure) 🔻	Elset	Label	Magnitude	Load Id 🔺
	pressure_set	P 🔻		
Define DLOAD on:				
C Elements or geometry				
Element sets				
pressure_set 💌				
pressure_set				
Review Reset Set	1			
Create/Edit set				
Display/Review from panel				
				-
	4			Þ
	Show face	s	Review I	Reset Update

11. Because the **pressure_set** contains shell elements, the direction of normal to the elements must be known to determine the sign of the magnitude. Find the direction of the normal by selecting the *pressure_set* element from the table and clicking *Show faces*.

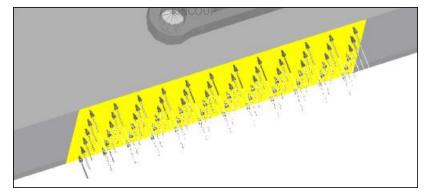




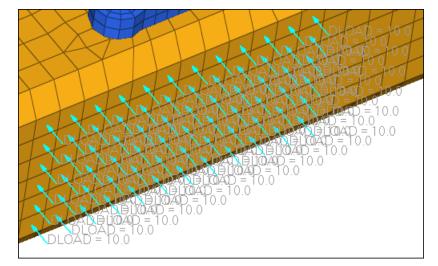
- 12. Clear the display by right-clicking on *Show faces*.
- 13. In the element sets table, Magnitude column, enter -10 for pressure_set.
 - **Note**: The negative magnitude means pressure load in the opposite direction of the underlying shell element normals.
- 14. Click **Update**. The HyperMesh database updates. The **Load type** line, at the bottom of the dialog, now displays Dload, which indicates DLOAD as another load type created in the loads_and_constraints load collector. The corresponding load type is the first pane is also highlighted.



- 15. In the element sets table, **Elset** column, click *pressure_set*.
- 16. Click *Review* | *Reset Set* to review the loads.







17.Revert back to the standard view by right-clicking on *Review* | *Reset Set*.

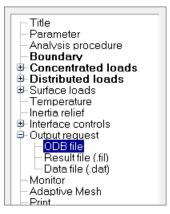
In this exercise, you constrained and applied distributed loads on the model using HyperMesh panels. The loads (*DLOAD) information is automatically stored in **step1**. Next, you will specify the output requests for this step.

Steps 8-9: Define Output Requests

In this exercise, you will specify several output requests for **step1**. There are two methods for defining output request described below.

Step 8: Request ODB file outputs

1. In the first pane of the Load Step dialog, expand *Output request*, and click *ODB file*.



- 2. Click New.
- 3. In the Create Output block dialog, Name field, enter step1.
- 4. Click *Create*.



5. In the **Output block** table, click *step1*. A new set of tabs displays.

Output block: step1	Output Node Output	Element Output Conta	act Output	Energy Output	Radiatior	n Output
Status Name Status Step1	□ Output: field	 Optional parame 	eters:			
	☐ Node output	□ Name:	step1	🗖 Mode lis	st	
	Element output	E OP:	new	👻 🔲 No. of a	datalines:	0 Se
	Contact output	🗖 Variable:	all	¥		
	Energy output	Frequency:				
	Energy output	Friequency.	, 			

- 6. In the **Output** tab, select the **Output** checkbox. Leave **Output** set to **field**.
- 7. Select the **Node output** and **Element output** checkboxes. The **Node Output** and **Element Output** tabs become active.

Output Node Output Elemer	t Output
✓ Output: field ▼	Optional parameters:
, I ■ Node output	🗖 Name:
Element output	D OP:
Contact output	🗆 Variable:
Energy output	Frequency:
Radiation output	Time marks:

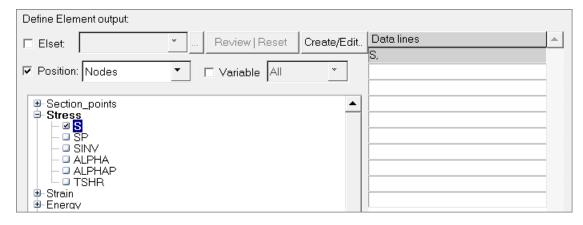
- 8. Click the **Node Output** tab.
- 9. Expand *Displacement* and select *U*. The **Data lines** table now displays "U", which allows you to request displacement results in the .obd file.
 - Tip: You can manually type output request into the **Data lines** table, including unsupported requests. They will be written just as they are entered in the table.

Define Node output:							
□ Nset:	*	Review Reset	Create/Edit.	Data lines U,	-		
🗖 Variable: 📶	*	🗖 Global:	YES 💌	o,			
Displacement D							

10. Click Update.



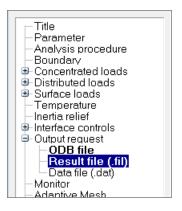
- 11. Click the *Element Output* tab.
- 12. Select the **Position** checkbox, and set it to **Nodes**.
- 13. Expand *Stress* and select *S*. The **Data lines** table now displays "S", which allows you to request stress results in the .obd file.



14. Click Update.

Step 9: Request results file (.fil) outputs

1. In the first pane of the **Load Step** dialog, expand **Output request**, and click **Result** *file* (*.fil*).



2. In the **Define** tab, select the **Node file** and **Element file** checkboxes. The Node File and Element File tabs become active.

Define	Node File	Element File	Contact File] [
🔽 Node	efile			
✓ Element file				
Contact file				
🗆 Energ	gy file			

3. Click the **Node File** tab.



p.12



4. Expand **Displacement**, and select **U**. The **Data lines** table displays "U", which allows you to request displacement results in the .fil file.

Define Node output:	
□ Nset: Review Reset Create/Edit.	Data lines
Frequency: 0 Last mode:	
🗆 Global: yes 🎽 🗖 Mode:	
Displacement Displacement Displacement Displacet Displacet Displacet Displacet Disp	

- 5. Click Update.
- 6. Click the **Element File** tab.
- 7. Select the **Position** checkbox, and set it to **averaged at nodes**.
- 8. Expand **Stress**, and select **S**. The Data line table displays "S", which allows you to request stress results in the .fil file.

Define Element output:	
Elset Review Reset Create/Edit.	Data lines
Directions: no * 🗖 Mode: 0	<u>,</u>
□ Frequency: 0	
Last Mode: 0	
Section_points Section_	

9. Click **Update**.

10. Under the **Output block** table, click *Review* | *Reset*. The **Review output block** dialog opens, and displays the output requests you made.

Note: This is the format used in the Abaqus input file (.inp).

🛆 Review output block: step1	×
*OUTPUT, FIELD,	
*NODE OUTPUT, U,	
*ELEMENT OUTPUT, POSITION = NODES S,	
*NODE FILE, U,	
*EL FILE, POSITION = AVERAGED AT NODES S,	
	-
<u> </u>	
	Close



- 11. Click *Close* to exit the **Review output block** dialog.
- 12. In the first pane of the **Load Step** dialog, click **Unsupported cards**.
- 13. Optional. Select the **Unsupported cards** checkbox to add any unsupported card.
- 14. Click *Close* to exit the **Load Step** dialog and return to the **Step Manager**. The **Step Manager** dialog displays all of the information you defined for **step1**.

💪 Step N	lanager						
Step	Load Case						
Export	Name	Analysis Type	Load Collector	Output Block	Interface Controls	Display	A
	Initial Condition	Model data					
N	step1	Static, PERTURB	loads_and_constraints	step1		N	
Г						Γ	+
Γ							
Γ						Γ	
						Γ	▲
Γ						Γ	
						Γ	
							-
•							Þ
Help	New	Edit Rev	iew Reset Text	Rename	Delete	Sync	Close

15. Click *Close* to exit the **Step Manager** dialog.

Steps 10-11: Export the database to an Abaqus input file

The data currently stored in the database must be output to an Abaqus .inp file for use with the Abaqus solver. The .inp file can then be used to perform the analysis using Abaqus outside of HyperMesh.

Step 10: Export the .inp file

- 1. From the menu bar, click *File* > *Export* > *Solver Deck*.
- 2. In the File: field, enter job1.inp.
- 3. To the left of **Export Options**, click 🔽.
- 4. Set Export to all.
- 5. Click *Export*.



Step 11: Save the .hm file and quit HyperMesh

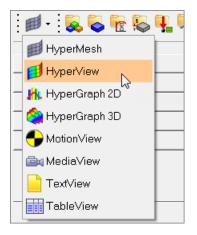
- 1. From the menu bar, click *File* > *Save as* > *Model*.
- 2. In the **Save Model As** dialog, enter job1.hm as the file name.
- 3. Click Save.

Notes:

- After you quit HyperMesh, you can run the Abaqus solver using the job1.inp file that was written from HyperMesh.
- At your site, you can use the ABAQUS license to run this model.
- If the batch mode option is being used, then enter the name of the .inp file exported in the previous step as the input file.
- After you have successfully completed the analysis, the result file will be available in your working directory with the name <jobname.odb>.
- Use <u>HvTrans</u> to translate the Abaqus solver result file to an H3D file.

Step 12: Open HyperView from the Application Menu

1. On the **Client Selector** toolbar, select *HyperView*. The HyperView environment displays.



2. In the panel area, load the model and results files.

Note: Load *.h3d files for both the model and result files.

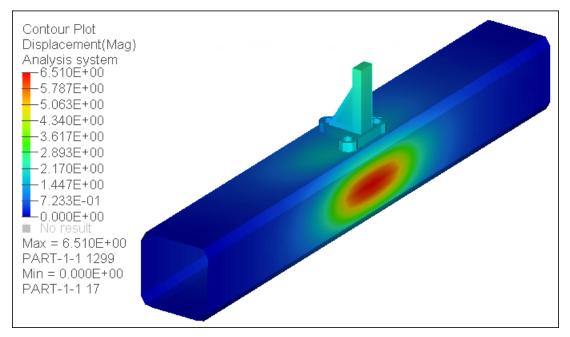
Load model and resul	ts:			
🔽 Load model	🖻 C:\job2-hv	rtrans.h3d		🗖 Overlay
Load results	🖻 C:\job2-hv	trans.h3d		
Result-	Math template:	Standard	 Reader Optic 	ons Apply



- 3. Click Apply.
- 4. On the **Results** toolbar, click **1** to open the **Contour** panel.
- 5. Review displacement (v) results by setting the **Result type** to **Displacement (v)**.

Result type:	Selection:	Averaging method: Value filter:	Display Legend Result
Displacement (v)	✓ Components I	None 💌 None 👻	Max: 0
Mag	Resolved in:	□ Variation < 10 (%)	Min: 0
Layers:	, 靀 🛛 Analysis System 💽	Averaging Options	Multiplier: 1
Use corner data	System	Envelope trace plot:	Offset: 0
	Use tracking system	None <u>Cache</u>	Edit Legend
		Apply	

6. Click **Apply**.

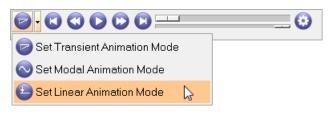


7. In the **Results** browser, review steps and increments.

Results ×	i i i i i i i i i i i i i i i i i i i			
Files	ID 💊 🤮			
😞 Model 1 🗌 🥥				
	· _ · · · · · · · · · · · · · · · · · ·			
step1:100k				



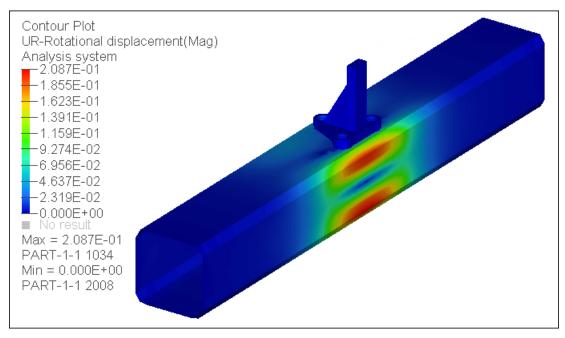
8. On the **Animation** toolbar, set the animation mode to **linear**.



- 9. Review the animation by clicking \mathbf{O} .
- 10. Review UR-Rotational displacement (v) results by setting the **Result type** to **UR**-**Rotational displacement (v)** in the **Contour** panel.

Result type:	Selection:	Averaging method: Value filter:	Display Legend Result
UR-Rotational displacement (💌	✓ Assemblies II	None 🝸 None 🔻	Discrete color
Mag	Resolved in:	□ Variation < 10 (%)	Interpolate colors
Layers: 💌 👻 - 🧼	Analysis System 🔹	Averaging Options	
Use corner data	System II	Envelope trace plot:	
	Use tracking system	None * Cache	
		Apply	

11. Click Apply.



For additional tools and techniques, refer to the tutorial <u>Pre-Processing for Bracket and</u> <u>Cradle Analysis using Abaqus - HM-4340</u>.

