Altair

HyperWorks

Altair HyperMesh 2019 Tutorials

HM-4330: Defining *STEP using Abaqus Step Manager

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 `abaqus_StepManager_tutorial.hm` 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  on the **Standard** toolbar.
4. In the **Open Model** dialog, open the `abaqus_StepManager_tutorial.hm` 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.

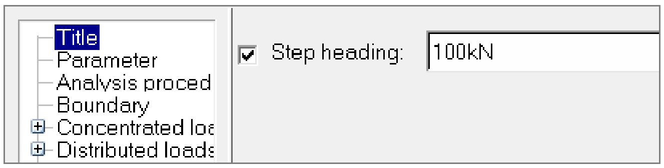
Step 2: Define a *STEP card and specify *STATIC as the analysis procedure

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.

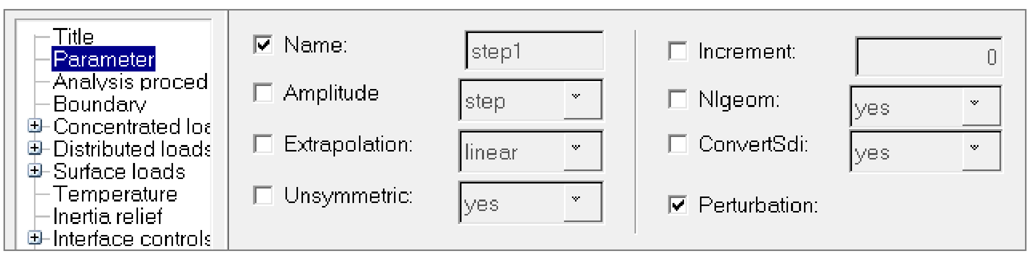
2. Click ***New***. The **Create New Step** dialog opens.
3. In the **Name** field, enter `step1`.
4. Click ***Create***. A step, labeled **step1**, opens the **Load Step** dialog.
5. In the first pane, select ***Title***. The **Step heading** option with a disabled field is displayed.

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

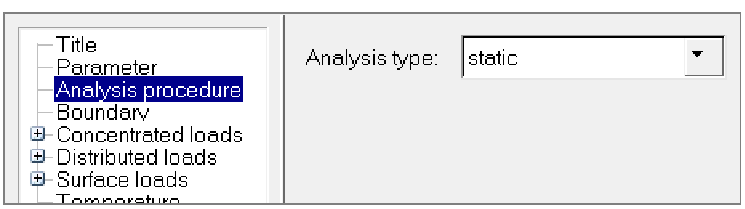


7. Click ***Update*** to store the heading information in **step1**.
8. In the first pane, select ***Parameter***.
9. Select the **Name** and **Perturbation** checkboxes.

   Note: Notice **Name** is already set to **step1**.



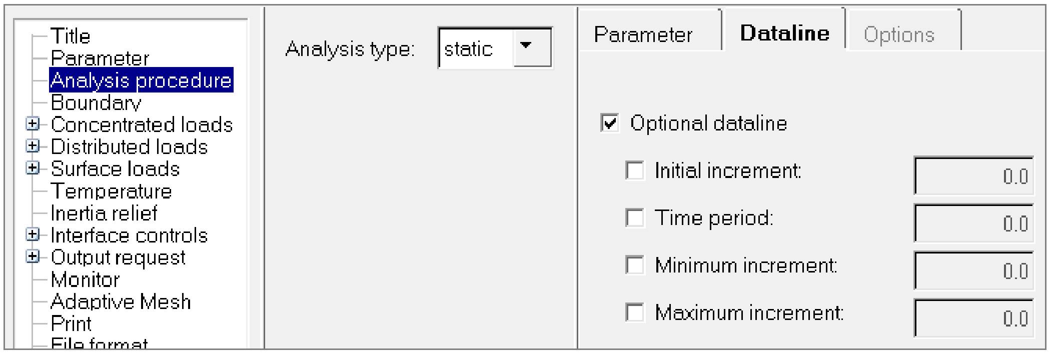
10. Click ***Update***.
11. In the first pane, select ***Analysis procedure***.
12. Set **Analysis type** to ***static***



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.



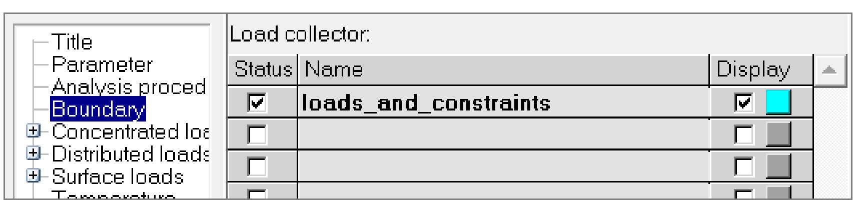
17. Click ***Update***.

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.

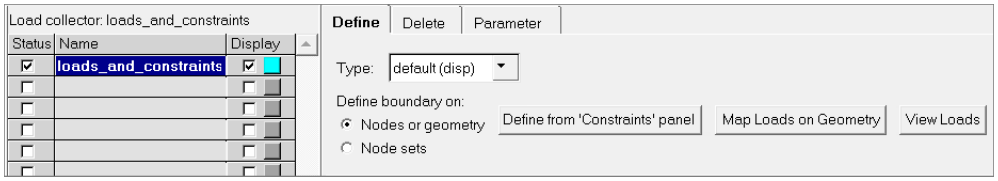


6. Verify that the **Status** checkbox for **loads_and_constraints** is selected.

   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.

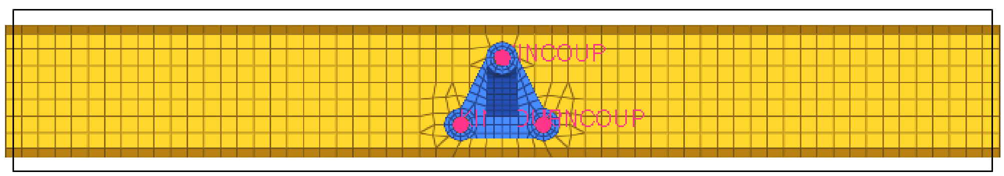
8. From the **Define** tab, verify **Type** is set to ***default(disp)***.



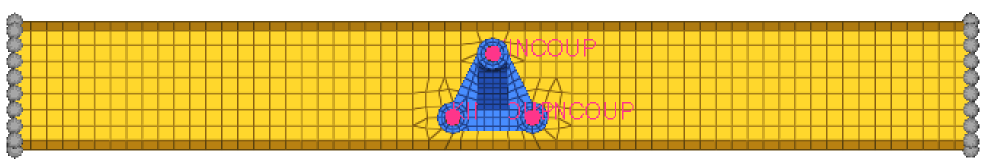
9. Click ***Define from 'Constraints' panel***. The **Constraints** panel opens. Use this panel to create constraints.

Step 4: Create constraints from the Constraints panel

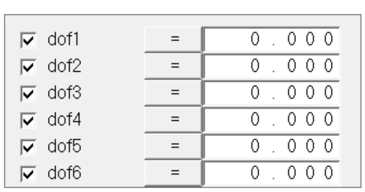
1. On the **Standard Views** toolbar, click (**XZ Right Plane view**).
2. In the **Constraints** panel, click ***nodes*** >> ***by window***.
3. With the exception of the nodes at the ends of the cradle, draw a rectangle around all of the displayed nodes.



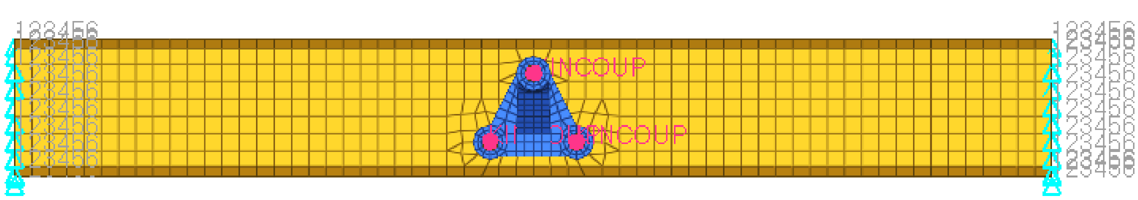
4. Select the ***exterior*** checkbox.
5. Click ***select entities***. HyperMesh selects all of the nodes outside the window you drew.



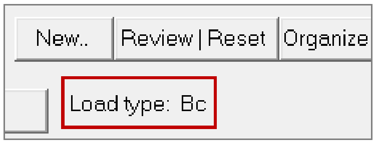
6. Verify that all six dof checkboxs are selected.



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

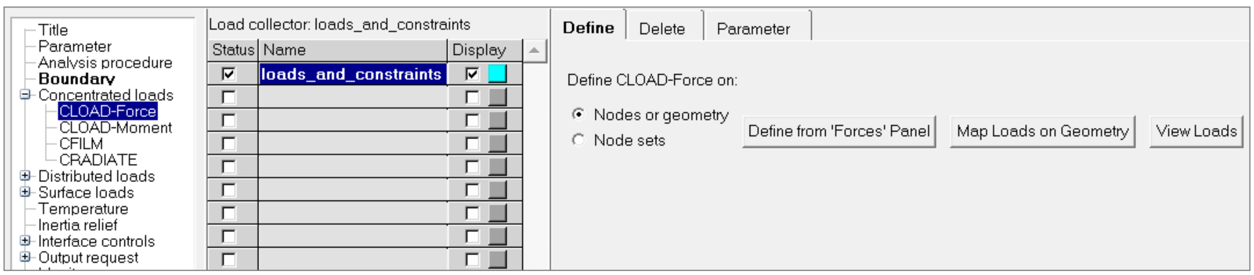


8. Click ***return*** to go back to the **Load Step** dialog.
9. At the bottom of the **Load Step** dialog, look at the **Load type** line. Bc (short for BOUNDARY) appears on this line, which indicates step1 is a load type created in the load_and_constraints load collector. The corresponding load type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



Step 5: Create Forces (*CLOAD)

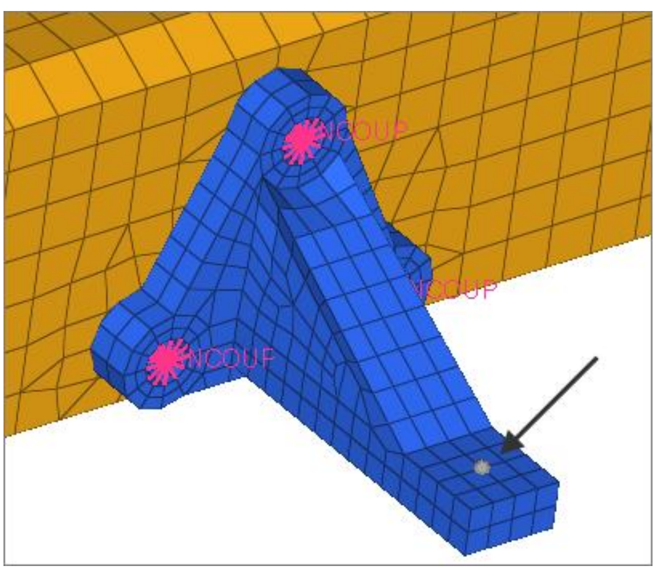
1. In the first pane of the **Load Step** dialog, expand ***Concentrated loads***, and select ***CLOAD-Force***. A new set of tabs displays.



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

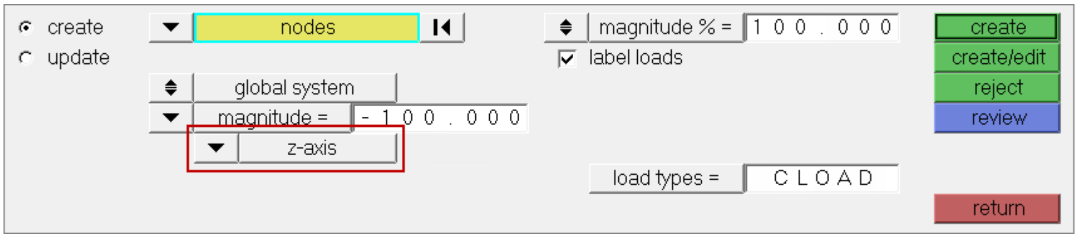
Step 6: Create forces from the Forces panel

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

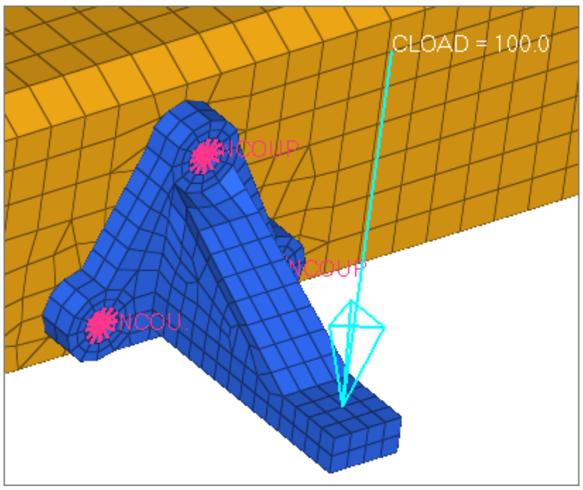


2. In the **Forces** panel, **magnitude=** field, enter `-100`.

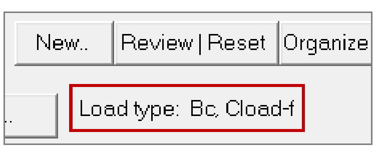
3. Set the orientation selector to ***z-axis***.



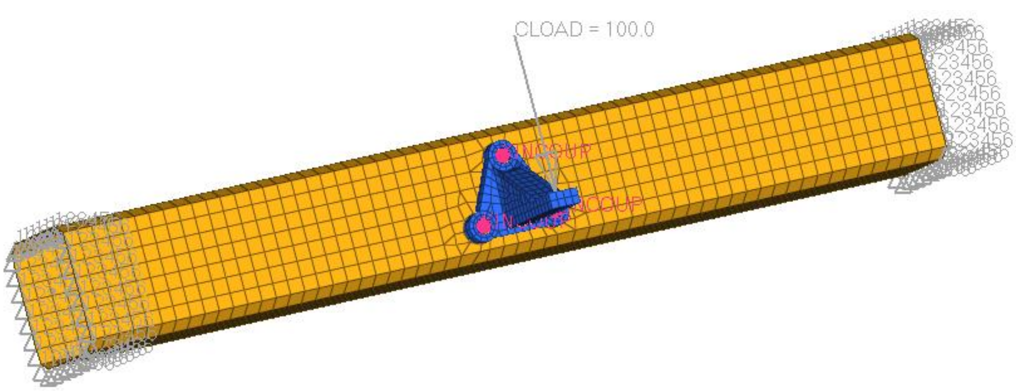
4. Click ***create***.



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.

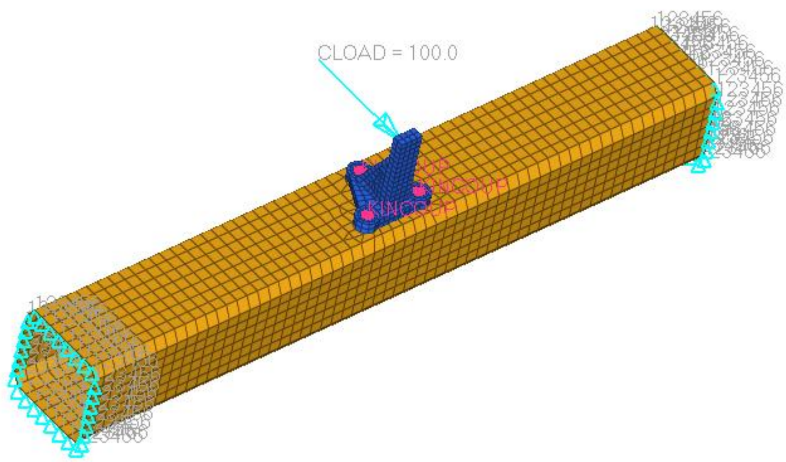


8. Revert the highlighted constraints and forces to the load collector color by right-clicking on ***Review***.

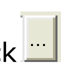
Step 7: Define pressure loads (*DLOAD) with element set

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

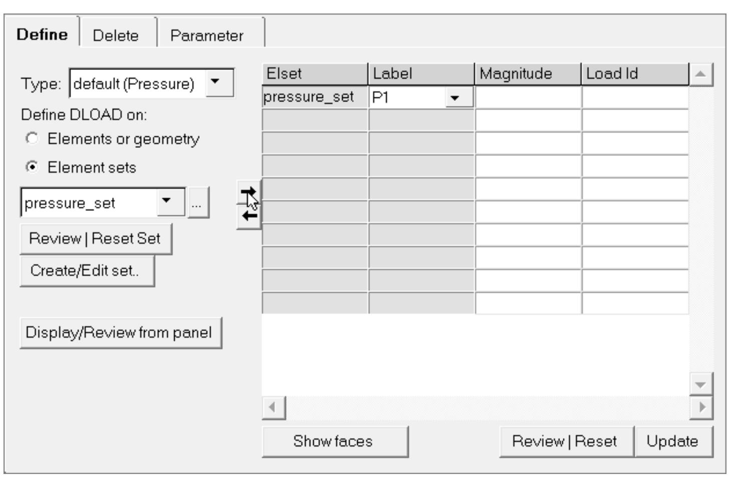
1. In the first pane of the **Load Step** dialog, expand ***Distributed loads***, and select ***DLOAD***. A new set of tabs displays.
2. From the **Define** tab, set **Define DLOAD on** to ***Element sets***. The element sets table displays.
3. Rotate the model to the view as shown in the image below.



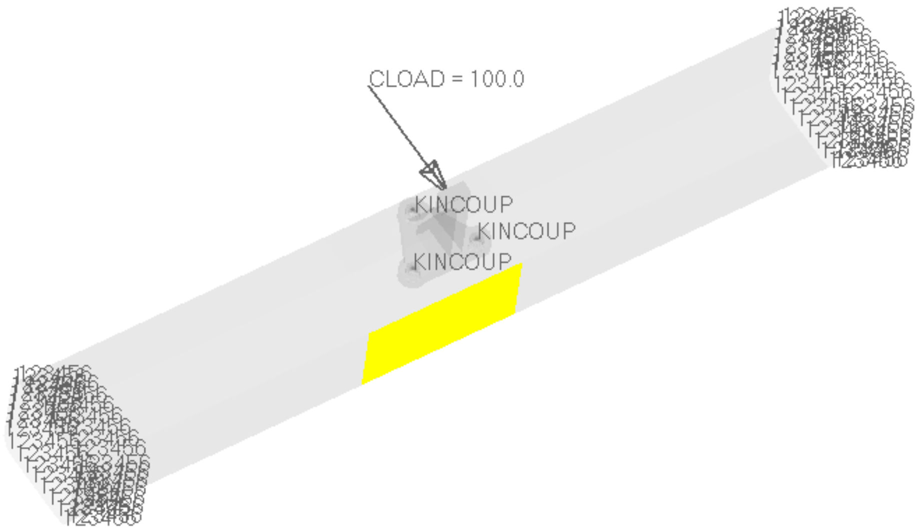
4. From the **Load Step** dialog, set **Type** to ***default (Pressure)***.
5. Set **Element sets** to ***pressure_set***.

   Tip: To view the entire list of element sets, click  . Use **Filter** and **Sort** to narrow your search.

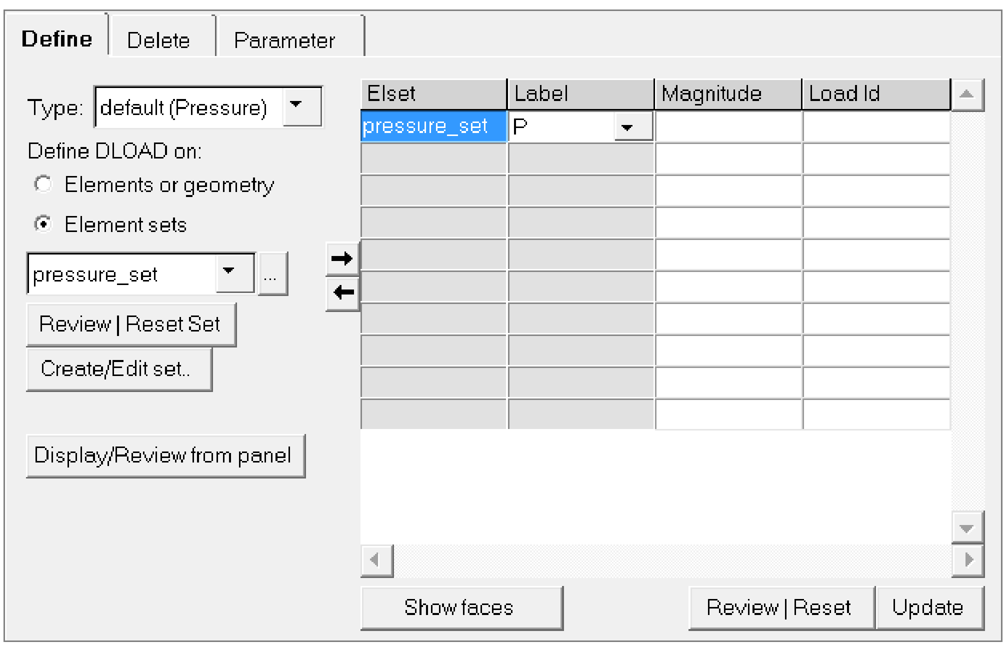
6. Click the right arrow to add the selected set to the element sets table.



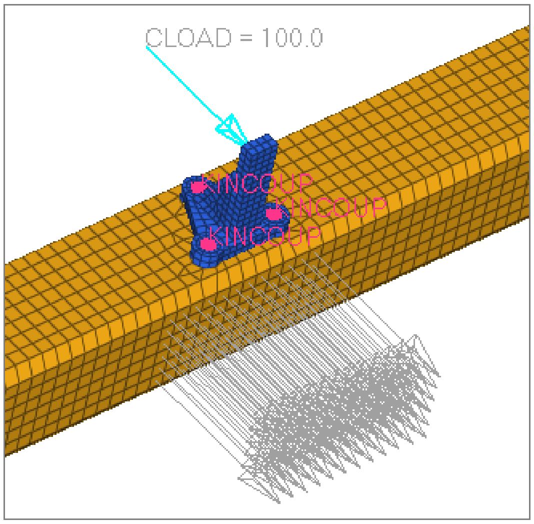
7. Under **Element sets**, click ***Review | Reset Set***. The element set highlights.



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**.

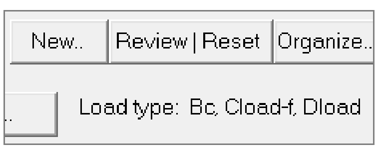


11. Because the **pressure_set** contains shell elements, the direction of normal to the elements must be known to determine the sign of the magnitude. Find the direction of the normal by selecting the ***pressure_set*** element from the table and clicking ***Show faces***.

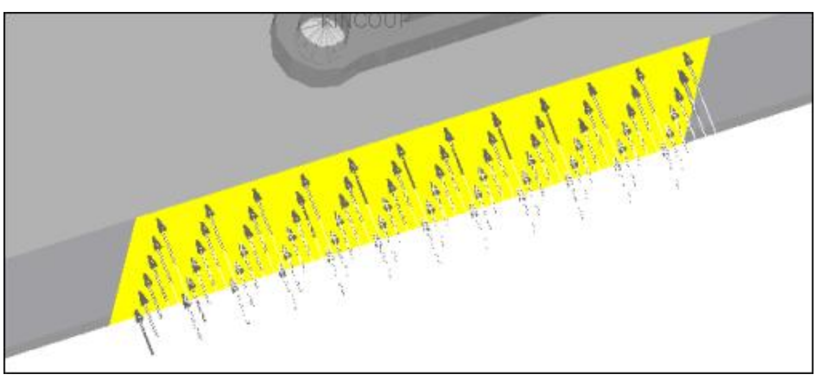


12. Clear the display by right-clicking on ***Show faces***.
13. In the element sets table, **Magnitude** column, enter `-10` for **pressure_set**.

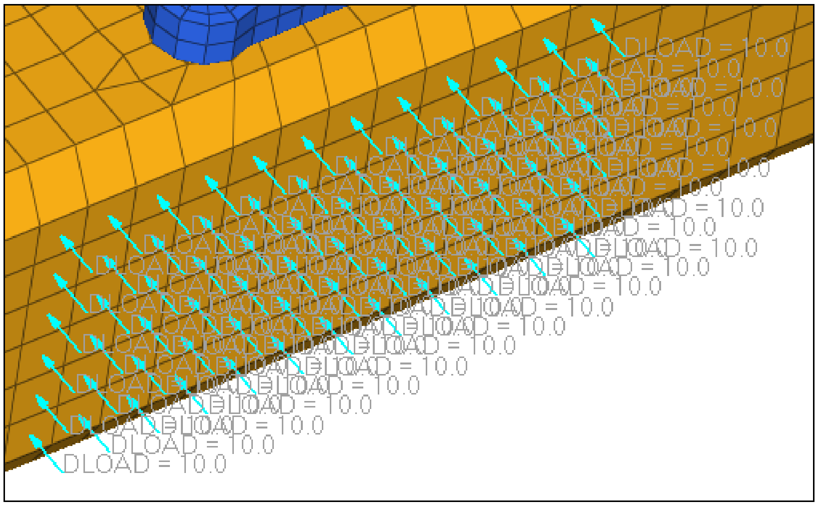
    Note: The negative magnitude means pressure load in the opposite direction of the underlying shell element normals.
14. Click ***Update***. The HyperMesh database updates. The **Load type** line, at the bottom of the dialog, now displays Dload, which indicates DLOAD as another load type created in the 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***.



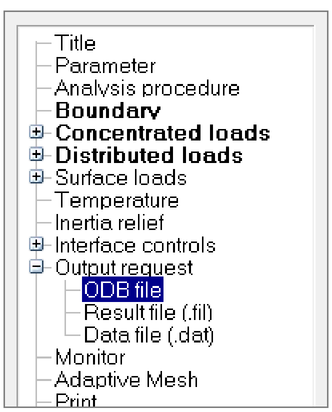
In this exercise, you constrained and applied distributed loads on the model using HyperMesh panels. The loads (*DLOAD) information is automatically stored in **step1**. Next, you will specify the output requests for this step.

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.

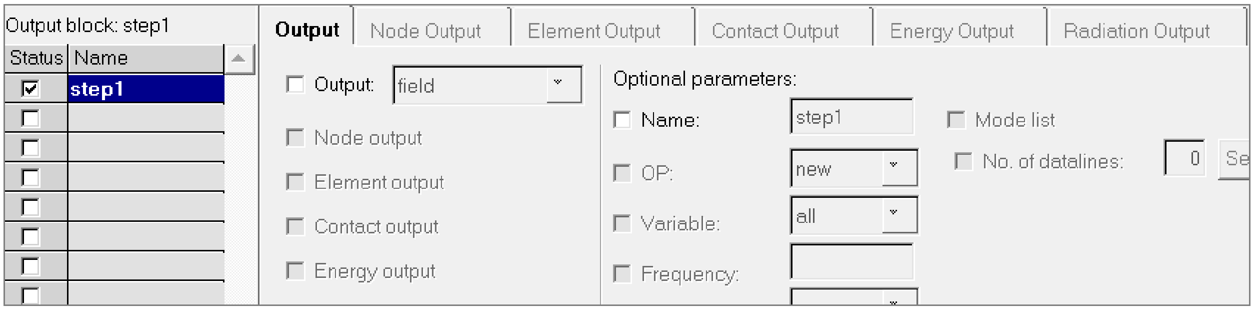
Step 8: Request ODB file outputs

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

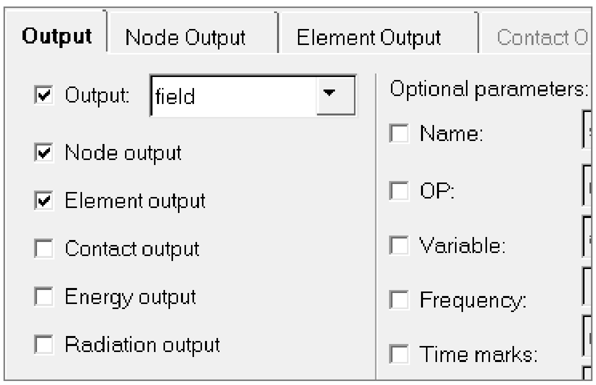


2. Click ***New***.
3. In the **Create Output block** dialog, **Name** field, enter `step1`.
4. Click ***Create***.

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



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.

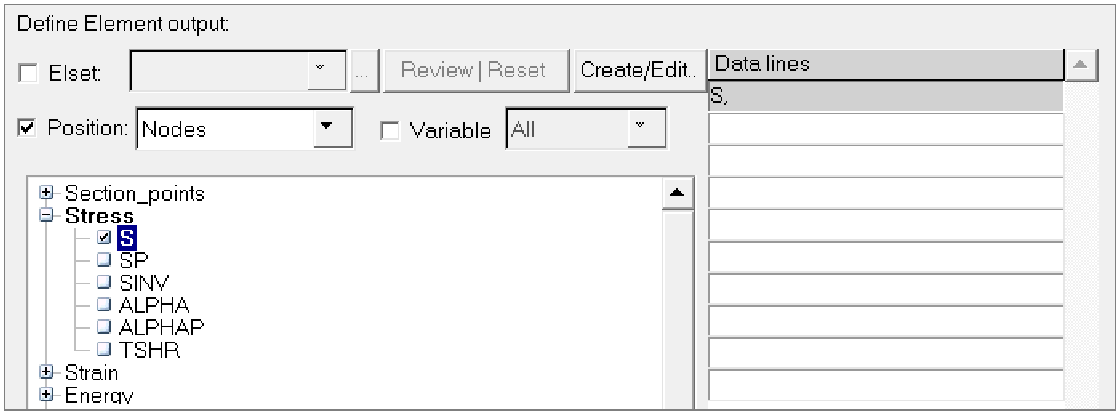


8. Click the ***Node Output*** tab.
9. Expand ***Displacement*** and select ***U***. The **Data lines** table now displays "U", which allows you to request displacement results in the `.obd` file.

   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.

10. Click ***Update***.

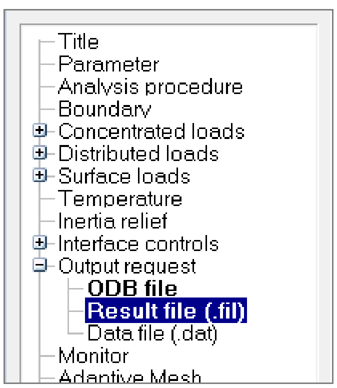
11. Click the ***Element Output*** tab.
12. Select the **Position** checkbox, and set it to ***Nodes***.
13. Expand ***Stress*** and select ***S***. The **Data lines** table now displays "S", which allows you to request stress results in the `.obd` file.



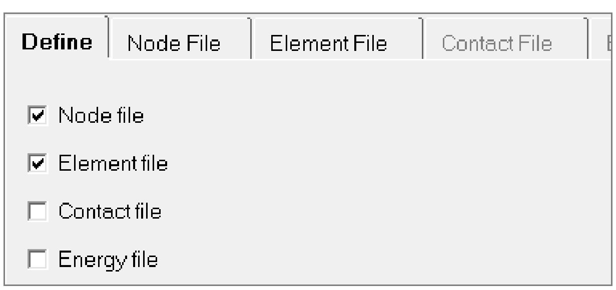
14. Click ***Update***.

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)***.

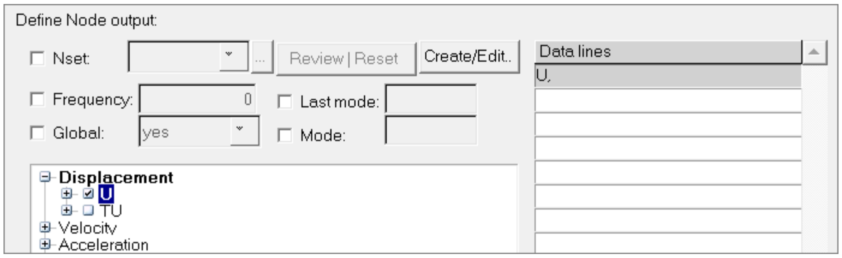


2. In the **Define** tab, select the ***Node file*** and ***Element file*** checkboxes. The Node File and Element File tabs become active.

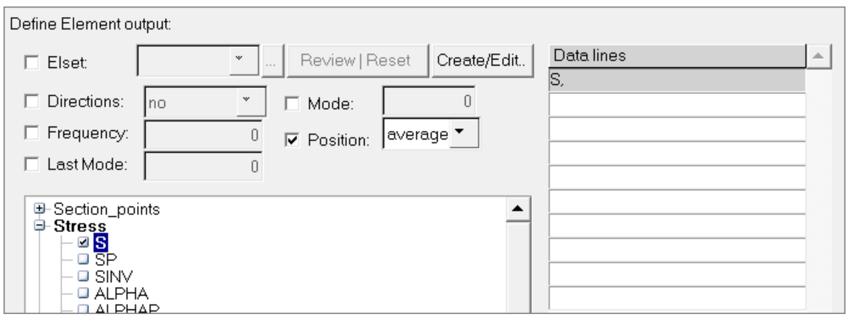


3. Click the **Node File** tab.

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

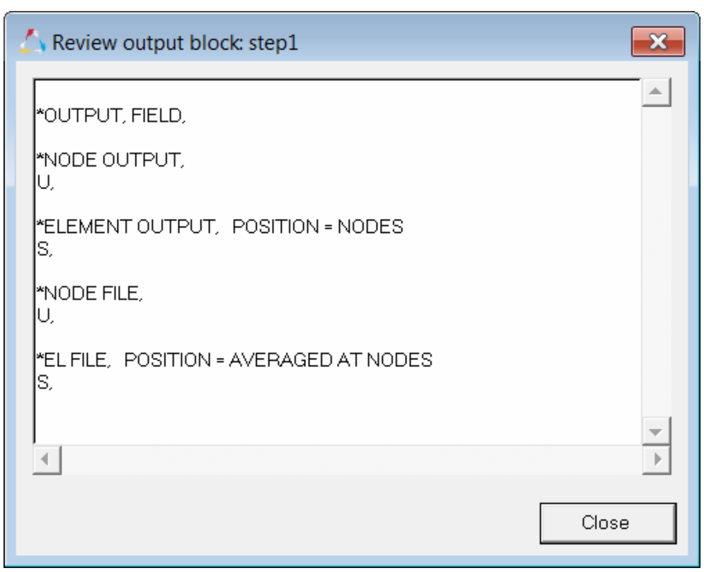


5. Click ***Update***.
6. Click the **Element File** tab.
7. Select the ***Position*** checkbox, and set it to ***averaged at nodes***.
8. Expand ***Stress***, and select **S**. The Data line table displays "S", which allows you to request stress results in the `.fil` file.

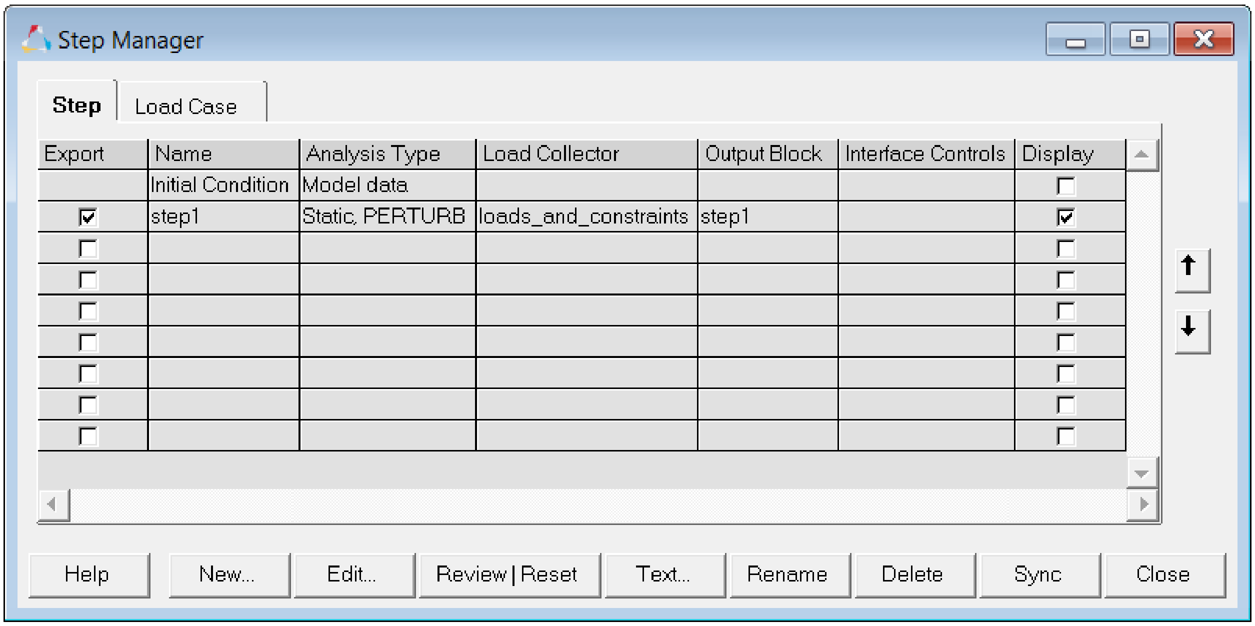


9. Click ***Update***.
10. Under the **Output block** table, click ***Review | Reset***. The **Review output block** dialog opens, and displays the output requests you made.

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



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**.



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

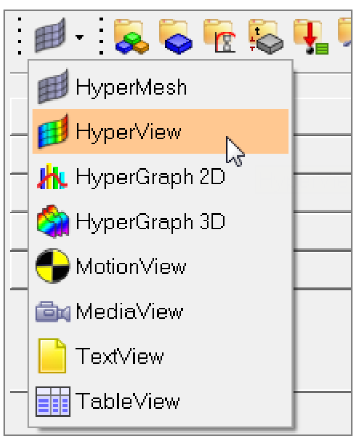
1. From the menu bar, click ***File*** > ***Save as*** > ***Model***.
2. In the **Save Model As** dialog, enter `job1.hm` as the file name.
3. Click ***Save***.

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 HvTrans to translate the Abaqus solver result file to an H3D file.

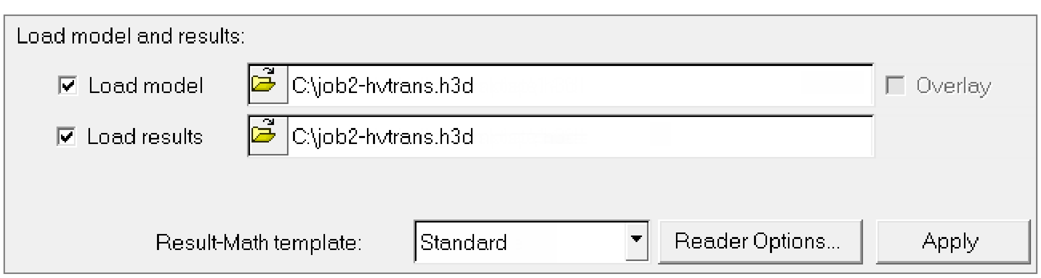
Step 12: Open HyperView from the Application Menu

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

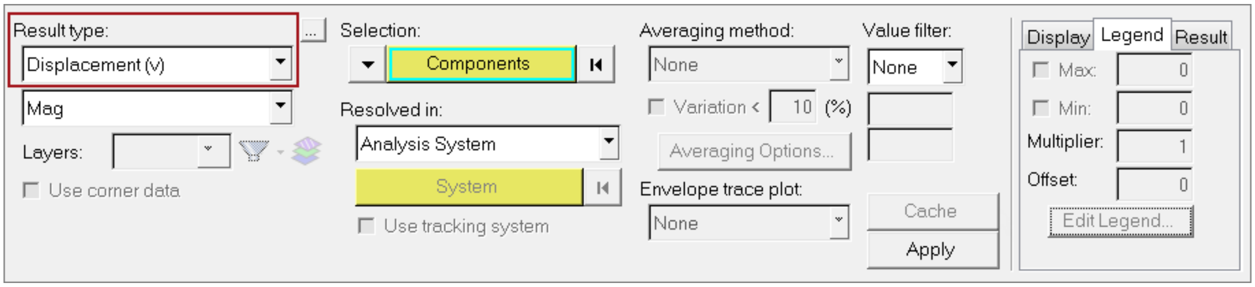


2. In the panel area, load the model and results files.

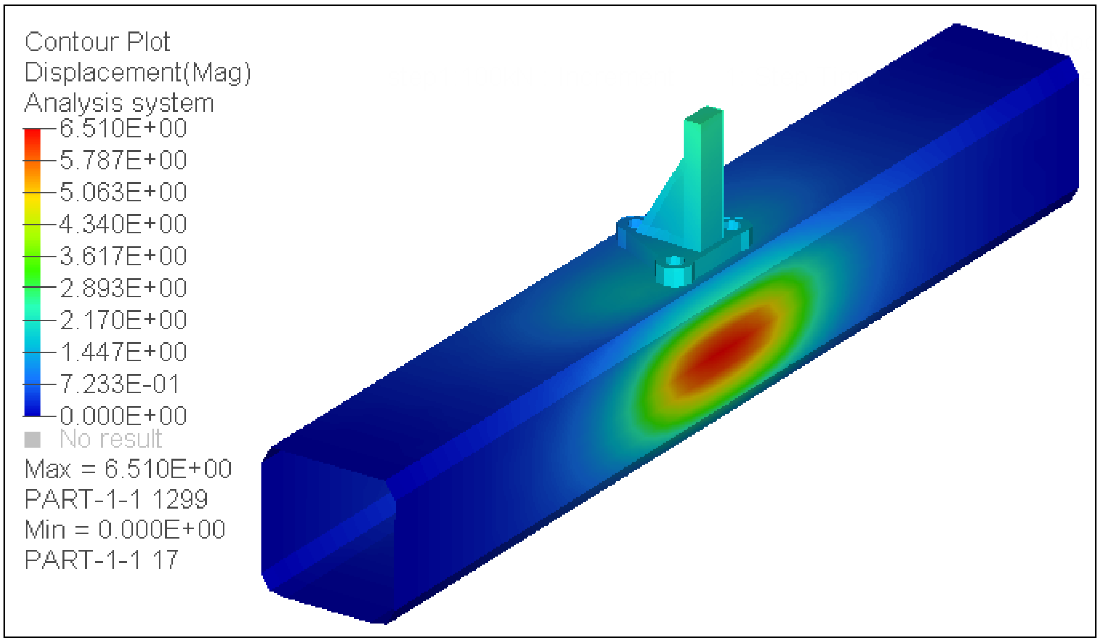
   Note: Load *.h3d files for both the model and result files.



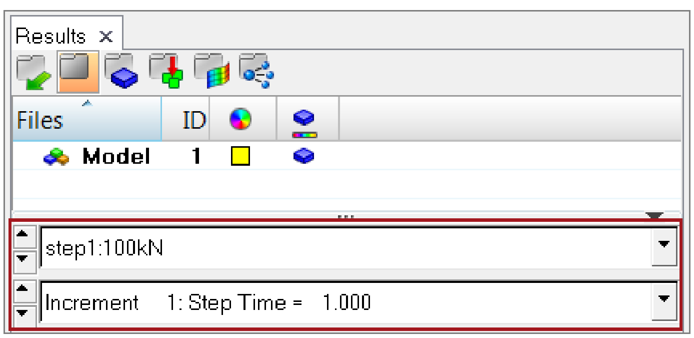
3. Click ***Apply***.
4. On the **Results** toolbar, click to open the **Contour** panel.
5. Review displacement (v) results by setting the **Result type** to ***Displacement (v)***.



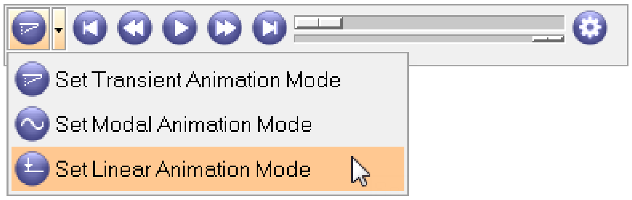
6. Click ***Apply***.



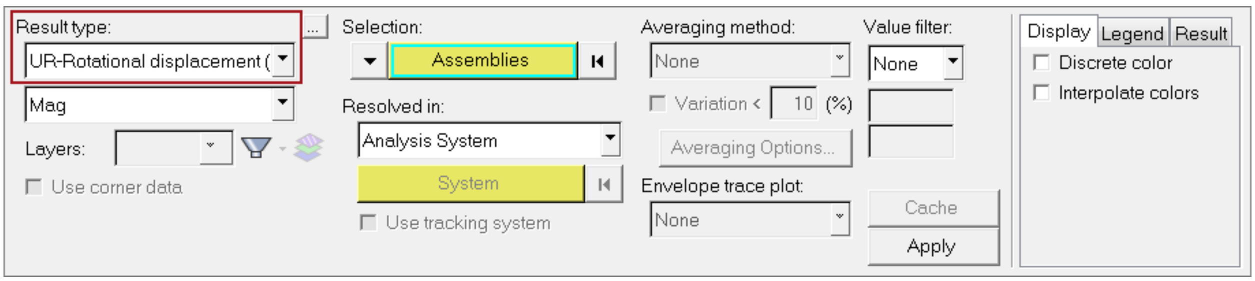
7. In the **Results** browser, review steps and increments.



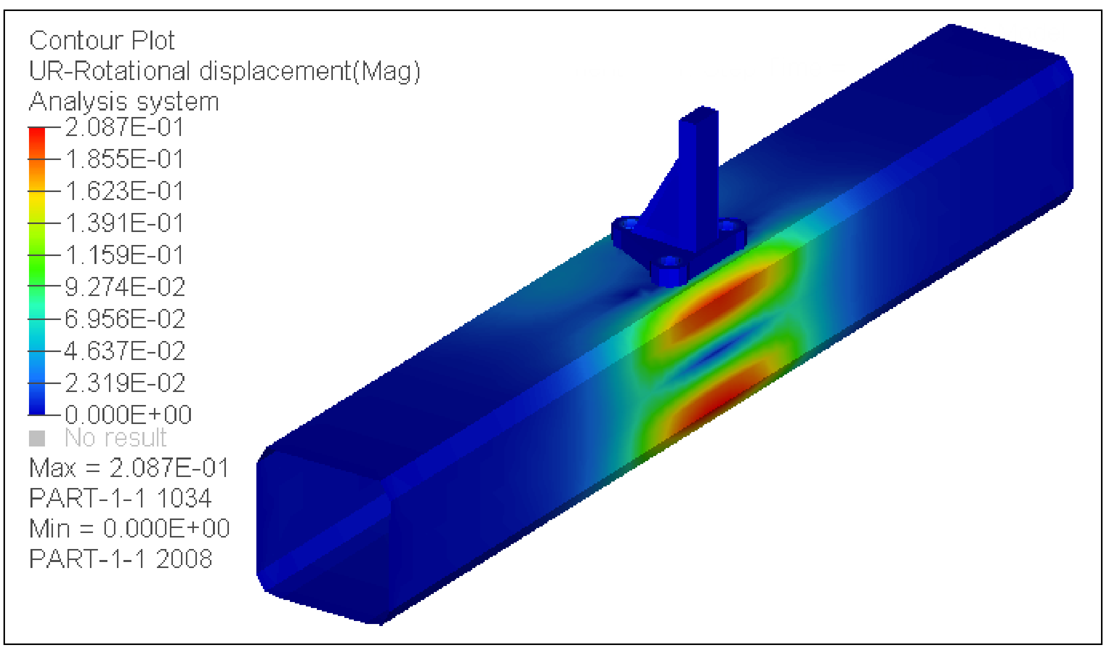
8. On the **Animation** toolbar, set the animation mode to **linear**.



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



11. Click ***Apply***.



For additional tools and techniques, refer to the tutorial [Pre-Processing for Bracket and Cradle Analysis using Abaqus - HM-4340].