Altair

HyperWorks

Altair HyperMesh 2019 Tutorials

HM-4200: Setting Up Nastran Static Analysis in HyperMesh

HM-4200: Setting Up Nastran Static Analysis in HyperMesh

In this tutorial, you will learn how to:

- Define a model in HyperMesh
- Apply boundary conditions in HyperMesh
- Write the Nastran input deck
- View the results

You will use the HyperMesh Nastran interface to create finite elements on the geometry of a plate with a hole, apply boundary conditions, and perform finite element analysis.

Model Files

This tutorial uses the `plate_hole.hm` file, which can be found in `<hm.zip>/interfaces/nastran/`. Copy the file(s) from this directory to your working directory.

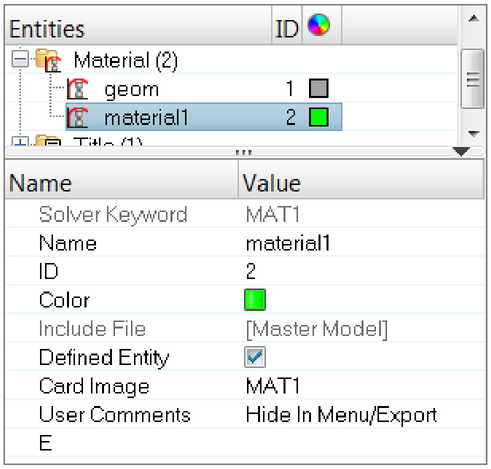
Exercise

Step 1: Retrieve the model file and select the Nastran user profile

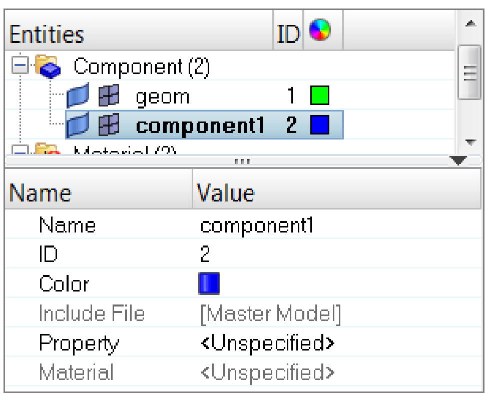
1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to ***Nastran***.
3. Open a model file by clicking ***File*** > ***Open*** > ***Model*** from the menu bar, or clicking  on the **Standard** toolbar.
4. In the **Open Model** dialog, open the `plate_hole.hm` file.

Step 2: Create material collectors and components

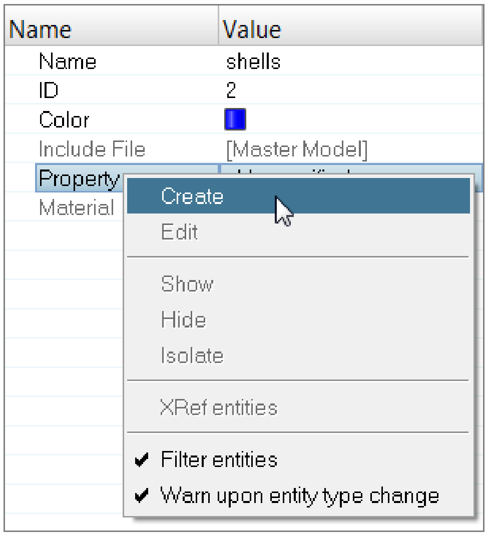
1. In the **Model** browser, right-click and select ***Create*** > ***Material*** from the context menu. A new material opens in the **Entity Editor**.



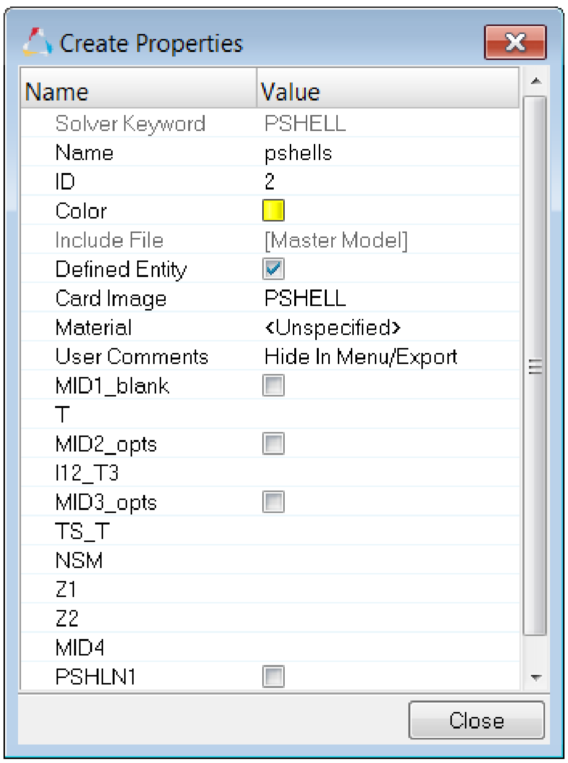
2. For **Name**, enter `steel`.
3. Set **Card Image** to ***MAT1***.
4. For **E**, enter `2e5`.
5. For **NU**, enter `0.30`.
6. For **RHO**, enter any number (if needed).
7. In the **Model** browser, right-click and select ***Create*** > ***Component*** from the context menu. A new component opens in the **Entity Editor**.



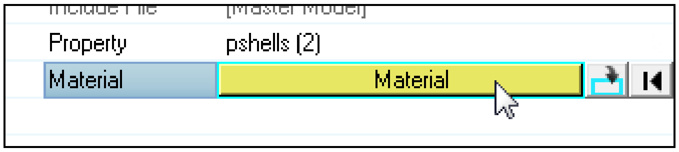
8. For **Name**, enter `shells`.
9. Right-click on **Property** and select ***Create*** from the context menu.



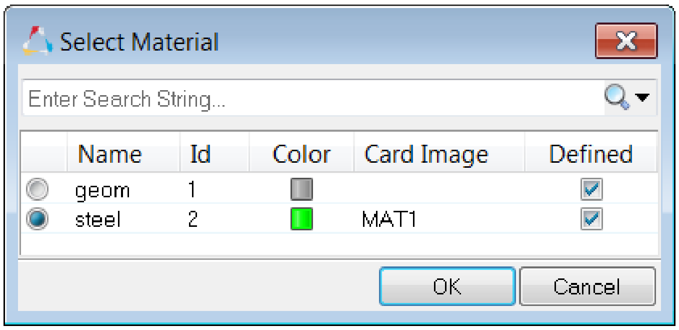
10. In the **Create Properties** dialog, **Name** field, enter `pshells`.



11. Set **Card Image** to ***PSHELL***.
12. For **T** (thickness), enter `1.0`.
13. Click ***Close***. HyperMesh assigns the property **pshells** to the component **shells**.
14. For **Material**, click ***Unspecified*** >> ***Material***.



15. In the **Select Material** dialog, select ***steel*** and then click ***OK***. HyperMesh assigns the material **steel** to the component **shells**.



Step 3: Mesh the geometry

Use the **Automeshing** panel to mesh interactively on surfaces. This panel contains tools for manipulating surface edges and meshing fixed points (locations where the mesher is required to place a node.) The elements generated are organized into the current component and shells.

1. Open the **Automesh** panel by pressing ***F12***.
2. Go to the **size and bias** subpanel.
3. Click ***surfs*** >> ***displayed***.
4. In the **element size** field, enter `40`.
5. Set the **elems to surf comp**/**elems to current comp** toggle to ***elems to current comp***.
6. Click ***mesh***. HyperMesh meshes the selected surfaces, and the meshing module opens.

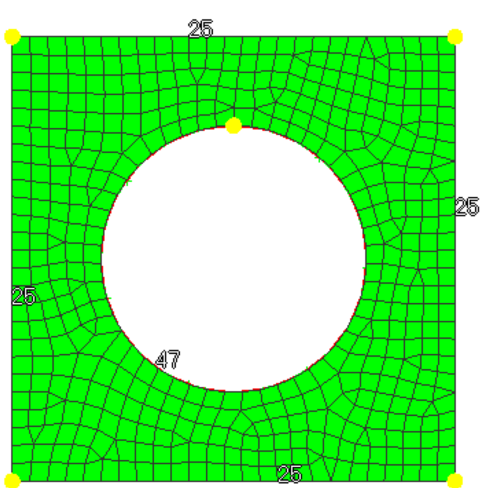


Plate mesh using element size of 40mm

7. Accept the mesh in the **shells** component by clicking ***return***.
8. Close the **Automesh** panel by clicking ***return***.

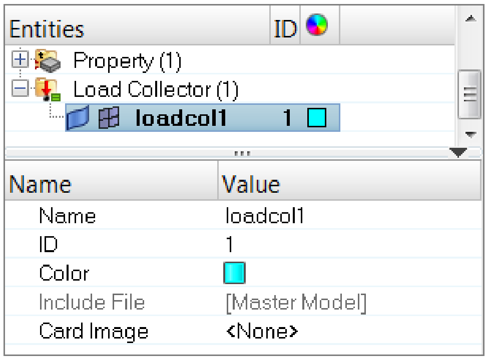
Steps 4-6: Apply boundary conditions to the model

In this section, the model is constrained so that two of the four edges cannot move. A total lateral load of 1000N is applied at the edge of the hole so that all forces point in the positive z-direction.

Step 4: Create collectors

Before creating boundary condition and loads, load collectors are created first. These load collectors are used for boundary conditions and loads.

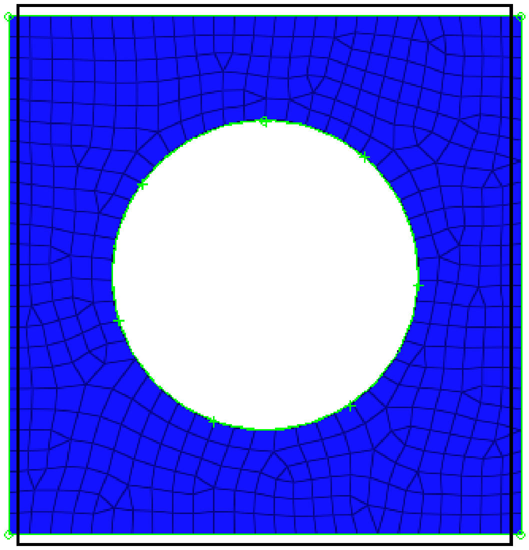
1. In the **Model** browser, right-click and select ***Create*** > ***Load Collector*** from the context menu. A new load collector opens in the **Entity Editor**.



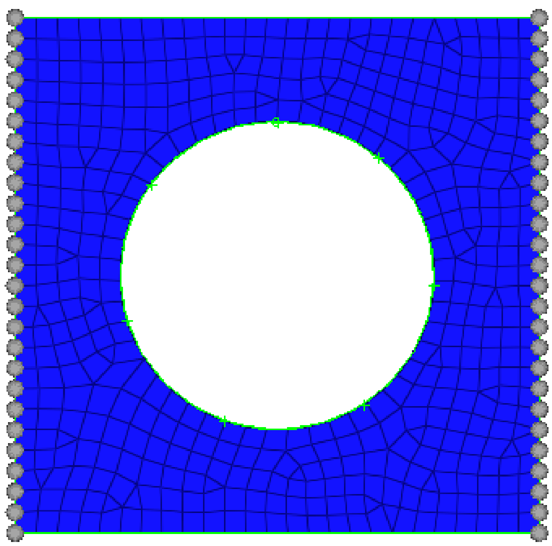
2. For **Name**, enter `spcs`.
3. Click the ***Color*** icon and select a color to display the load collector.
4. Create a second load collector labeled, **forces**.

Step 5: Create constraints

1. In the **Model** browser, **Load Collector** folder, right-click on ***spcs*** and select ***Make Current*** from the context menu.
2. Open the **Constraints** panel by clicking ***BCs*** > ***Create*** > ***Constraints*** from the menu bar.
3. Go to the ***create*** subpanel.
4. Set the entity selector to ***nodes***.
5. Click ***nodes*** >> ***by window***.
6. Select the ***exterior*** checkbox.
7. With the exception of the nodes at the ends, draw a box around all of the displayed nodes.

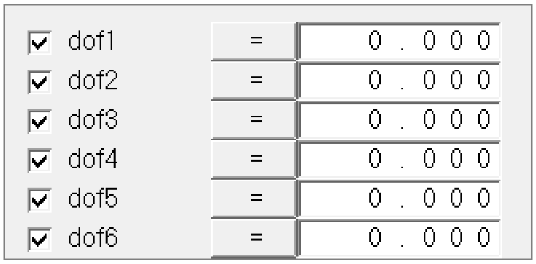


8. Click ***select entities***. HyperMesh selects all of the nodes outside of the window you drew.

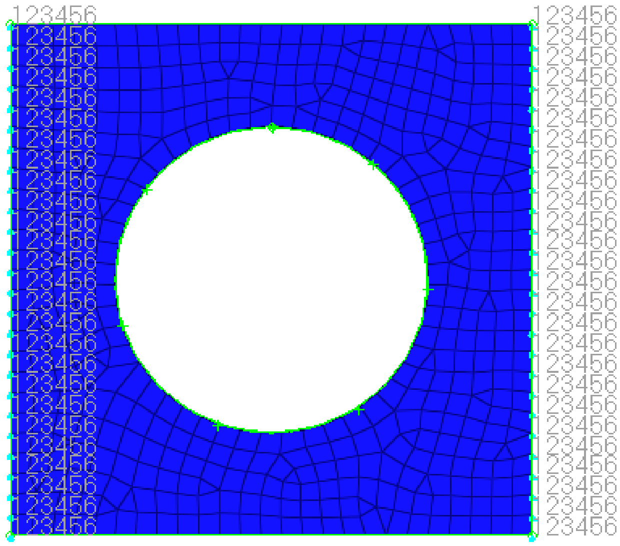


9. Select all of the ***dof*** (degree of freedom) checkboxes.

   Note: Dofs that are checked are constrained. Dofs 1, 2, and 3 are x, y, and z translation degrees of freedom, and dofs 4, 5, and 6 are x, y, and z rotational degrees of freedom.



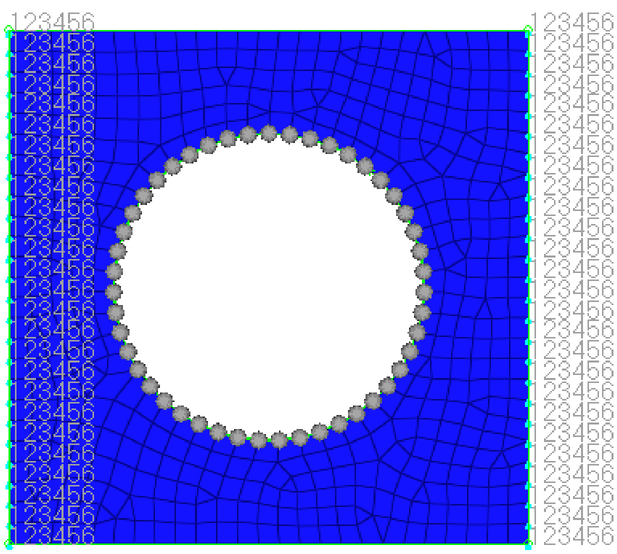
10. Apply these constraints to the selected nodes by clicking ***create***.



11. Click ***return***.

Step 6: Create forces on the nodes around the hole

1. In the **Model** browser, **Load Collector** folder, right-click on ***forces*** and select ***Make Current*** from the context menu.
2. Open the **Forces** panel by clicking ***BCs*** > ***Create*** > ***Forces*** from the menu bar.
3. Click ***nodes*** >> ***by path***.
4. Select all of the nodes around the hole of the model.



5. Click ***nodes*** >> ***save***.
6. Click ***return***.

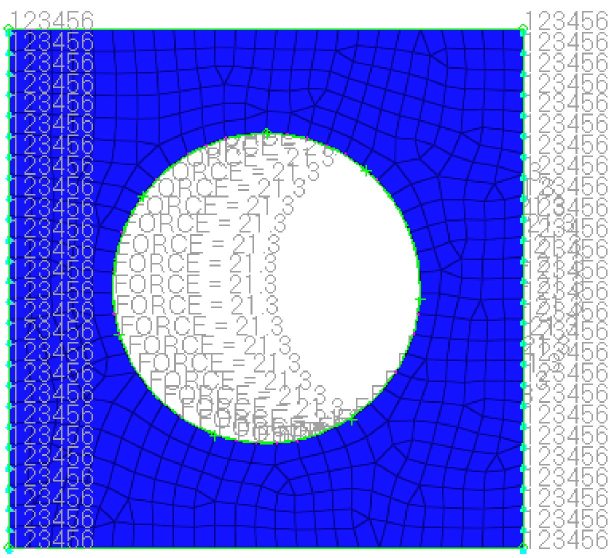
7. Open the **Count** panel by going to the **Tool** page and clicking ***count***.
8. Go to the ***FE entities*** subpanel.

   Note: Nodes are automatically counted so that a calculation can be made to create a total force of 1000N.

9. Set the entity selector to ***nodes***.
10. Retrieve the nodes you saved in the **Forces** panel by clicking ***nodes*** >> ***retrieve***.
11. Click ***selected***. HyperMesh counts the number of nodes around the hole.
12. Click ***return***.
13. Open the **Forces** panel.
14. Click ***nodes*** >> ***retrieve***.
15. In the **magnitude =** field, enter `21.277` (this is 1000/47).

    Note: The total load on the nodes around the hole is 1000N.

16. Set the orientation selector to ***z-axis***.
17. Click ***create***.

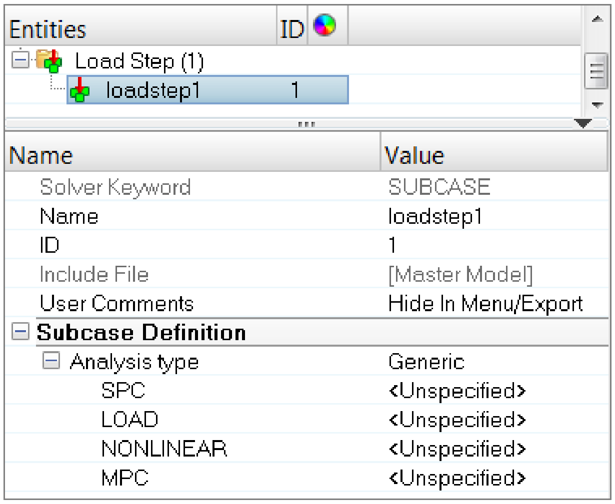


18. Click ***return***.

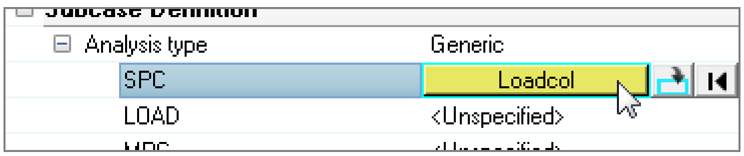
Steps 7-8: Create a Nastran subcase (a load step in HyperMesh)

Step 7: Create the loadstep

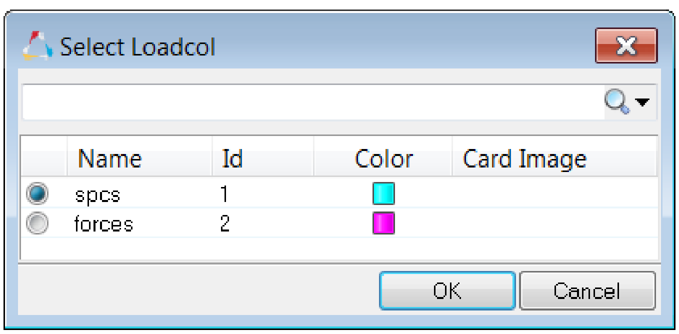
1. In the **Model** browser, right-click and select ***Create*** > ***Loadstep*** from the context menu. A new loadstep opens in the **Entity Editor**.



2. For **Name**, enter `lateral force`.
3. For **SPC**, click ***Unspecified*** >> ***Loadcol***.



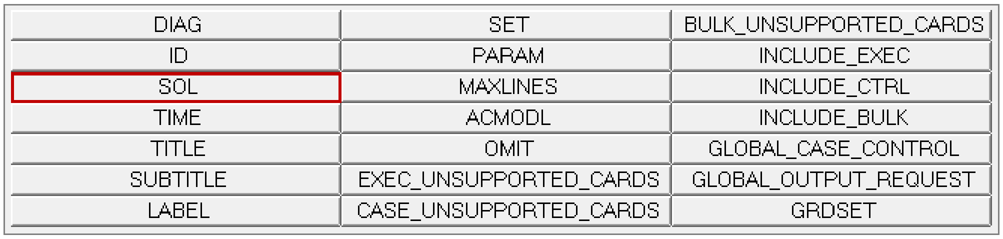
4. In the **Select Loadcol** dialog, select ***spcs*** and then click ***OK***.



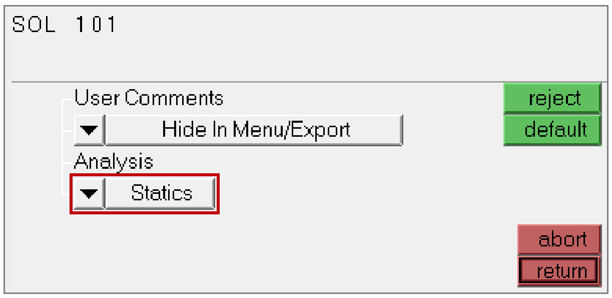
5. For **LOAD**, click ***Unspecified*** >> ***Loadcol***.
6. In the **Select Loadcol** dialog, select ***forces*** and then click ***OK***.
7. Under **SUBCASE OPTIONS**, select the ***OUTPUT*** checkbox.
8. Under **OUTPUT**, select the ***DISPLACEMENT*** checkbox.
9. Under **OUTPUT**, select the ***STRESS*** checkbox.

Step 8: Create control cards

1. From the menu bar, click ***Setup*** > ***Create*** > ***Control Cards***.
2. In the **Card Image** panel, click ***SOL***.

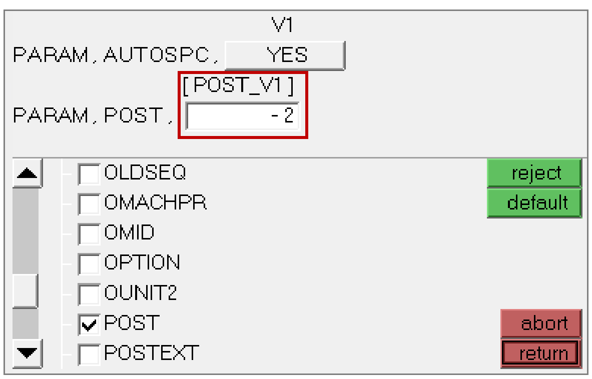


3. Set ***Analysis*** to ***Statics***.



4. Click ***return***.
5. Click ***PARAM***.
6. Select the ***AUTOSPC*** and ***POST*** checkboxes.
7. In the **card edit** field, click ***POST_V1*** and enter `-2` in the editable field.

   Note: This option specifies that an `op2` file should be created.



8. Click ***return*** twice.

Steps 9-10: Write the Nastran input deck

In this section, write the **Nastran** input deck file, specified with the `.dat` extension, before running Nastran.

Step 9: Write your file

1. From the menu bar, click ***File*** > ***Export*** > ***Solver Deck***. The **Export - Solver Deck** tab opens.

2. Set **File type** to ***Nastran***.
3. In the **File** field, navigate to your working directory an save the file as `plate_hole.dat`.
4. Click ***Export***. HyperMesh writes your HyperMesh database as a Nastran ASCII input deck.
5. Click ***Close***.

Step 10: Save your file and exit HyperMesh

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 as `plate_hole_new.hm`.

Steps 11-14: View the results

After running Nastran, the punch file `plate_hole.op2` is created. This file contains displacement and stress results for your linear static analysis. This section describes how to view those results in HyperMesh.

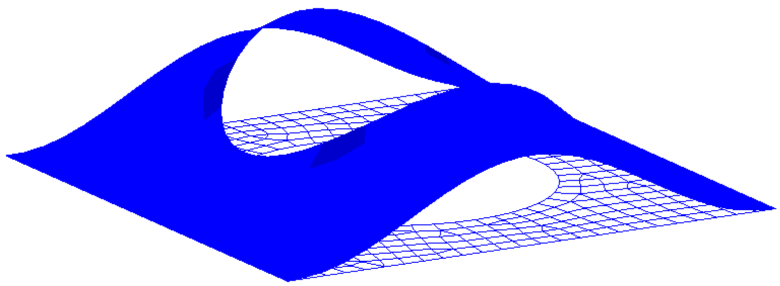
Step 11: Add a HyperView page to the session and load the fem and op2 files

1. On the **Page Controls** toolbar, click .
2. Set the **Client Selector** toolbar to ***HyperView***.
3. From the menu bar, click ***File*** > ***Open*** > ***Model***. The **Load Model** panel opens.
4. In the **Load model** field, navigate to your working directory and open the `plate_hold.dat` file.
5. In the **Load results** field, navigate to your working directory and open the `plate_hold.op2` file.
6. Click ***Apply***. HyperView loads the model and results.

Step 12: View a deformed shape

1. Open the **Deformed** panel by clicking ***Results*** > ***Plot*** > ***Deformed*** from the menu bar.
2. Set **Result type** to ***Displacement***.
3. Set **Scale** to ***Model units***.
4. Set **Type** to ***Uniform***.
5. For **Value** enter `25`.
6. Under **Undeformed shape**, set **Show** to ***Wireframe***.

7. View a deformed plot of your model overlaid on the original, undeformed mesh by clicking ***Apply***.



Step 13: View a contour plot of stresses and displacements

1. Open the **Contour** panel by clicking ***Results*** > ***Plot*** > ***Contour*** from the menu bar.
2. Set **Result type** to ***Displacement (v)***.
3. On the **Standard Views** toolbar, click .
4. Click ***Apply***.
5. Set **Result type** to ***Stress (t)***.
6. Set **Averaging method** to ***Simple***.
7. Click ***Apply***.