Altair

HyperWorks

Altair HyperStudy 2019 Tutorials

HS-1021: Working with a Parameterized File Model for Shape Variables

HS-1021: Working with a Parameterized File Model for Shape Variables

In this tutorial you will learn how to create a template file for shape variables and how to import them to HyperStudy. The input variables are three shape variables; xtrans, ytrans and radius. Each of these shapes are created by perturbing the mesh in the corresponding direction by 1 unit.

The sample base input template used in this tutorial can be found in `<hst.zip>/HS-1021/`. Copy the tutorial files from this directory to your working directory.

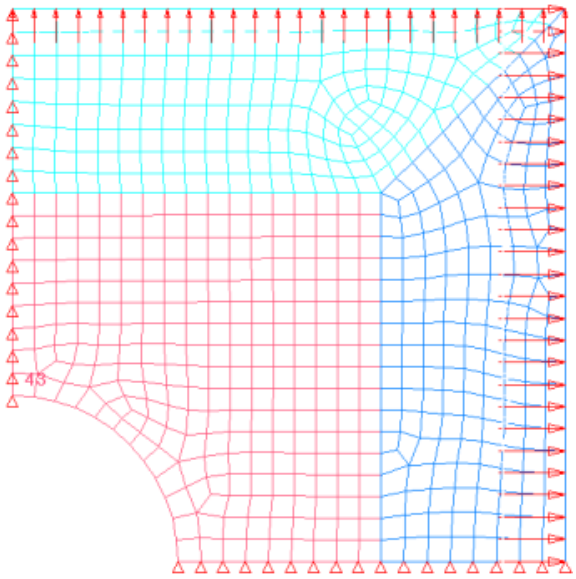


Figure 1: Double Symmetric Plate Model

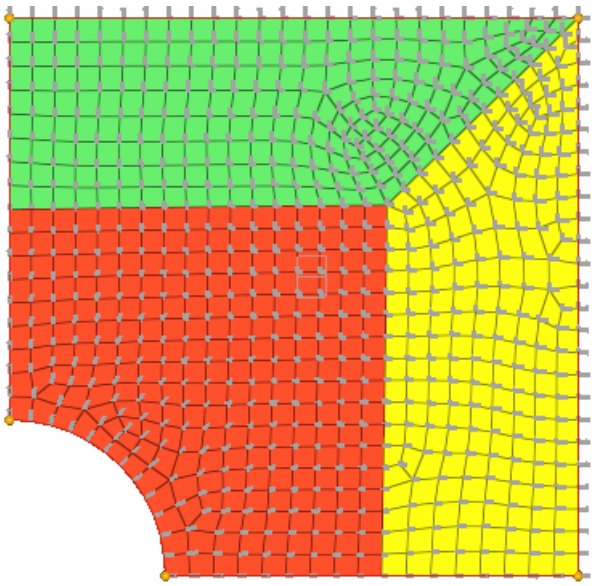
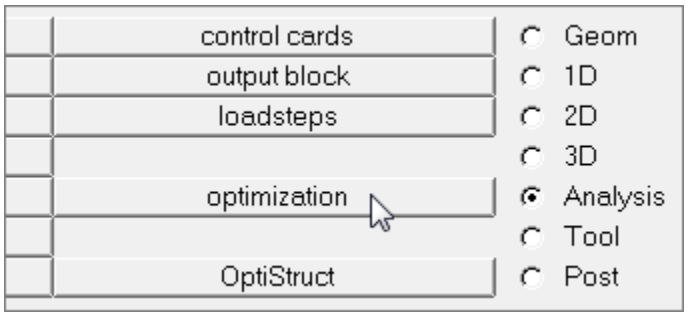


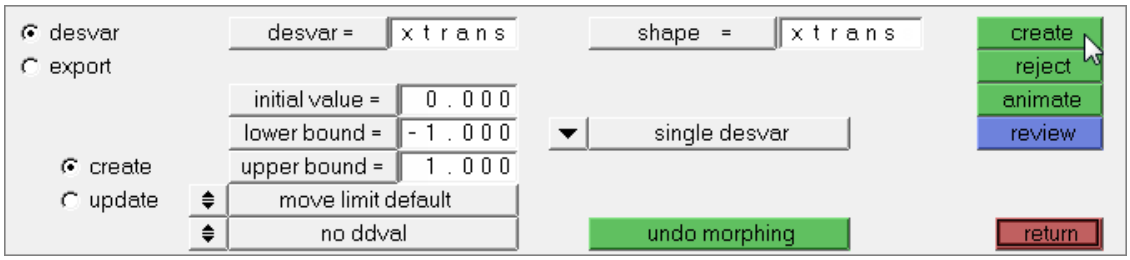
Figure 2: Double Symmetric Plate Model with Shape Vectors

Step 1: Exporting Shape Variables from HyperMesh

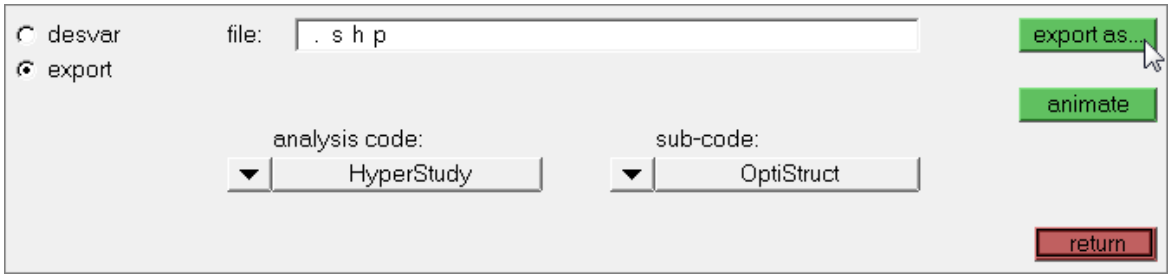
1. Start HyperMesh Desktop.
2. In the **User Profiles** dialog, set the user profile to **OptiStruct**.
3. From the menu bar, click ***File*** > ***Open*** > ***Model***.
4. In the **Open Model** dialog, open the `plate_with_shapes.hm` file. A model appears in the graphics area.
5. From the **Analysis** page, click ***optimization***.



6. Click ***shape***.
7. Go to the **desvar** subpanel. In this subpanel you will create three design variables (XTrans, YTrans and Rad) for three shapes.
8. Create a design variable.
   a. In the **desvar=** field, enter `XTrans`.
   b. Click ***Shape=***.
   c. Select the shape, ***xtrans***.
   d. Click ***create***.



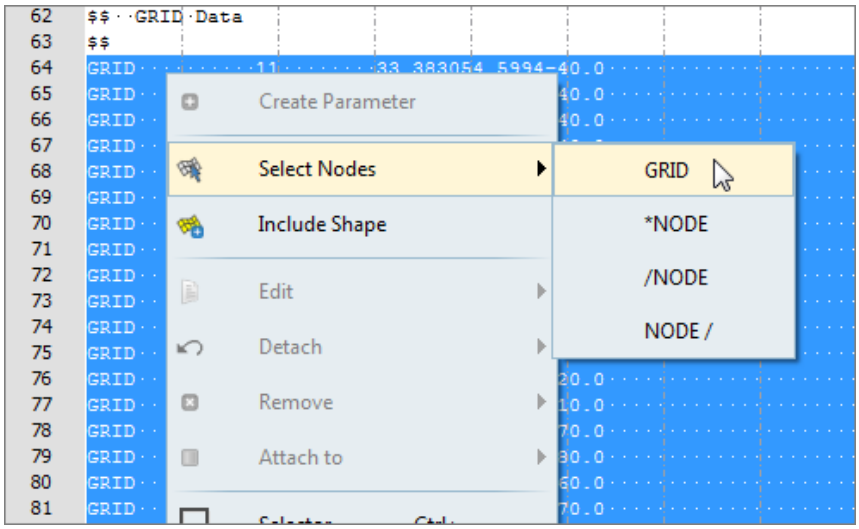
9. Create two more design variables labeled YTrans and Rad. Select the shape **ytrans** for design variable **YTrans**, and select the shape **radius** for design variable **Rad**.
10. Go to the **export** subpanel to export the shape variables.
11. Set **analysis code** to ***HyperStudy***.
12. Set **sub-code** to ***OptiStruct***.
13. Click ***export as***.



14. In the **Save As** dialog, save the file as `plate_with_shapes.shp`.
15. Quit HyperMesh by clicking ***File*** > ***Exit*** from the menu bar.

Step 2: Create the Base Input Template in HyperStudy

1. Start HyperStudy
2. From the menu bar, click ***Tools*** > ***Editor***. The **Editor** opens.
3. In the **File** field, open the `plate_with_shapes.fem` file.
4. In the editor, right-click and select ***Select Nodes*** > ***GRID*** from the context menu. All of the GRID cards in the `plate_with_shapes.fem` file highlight.

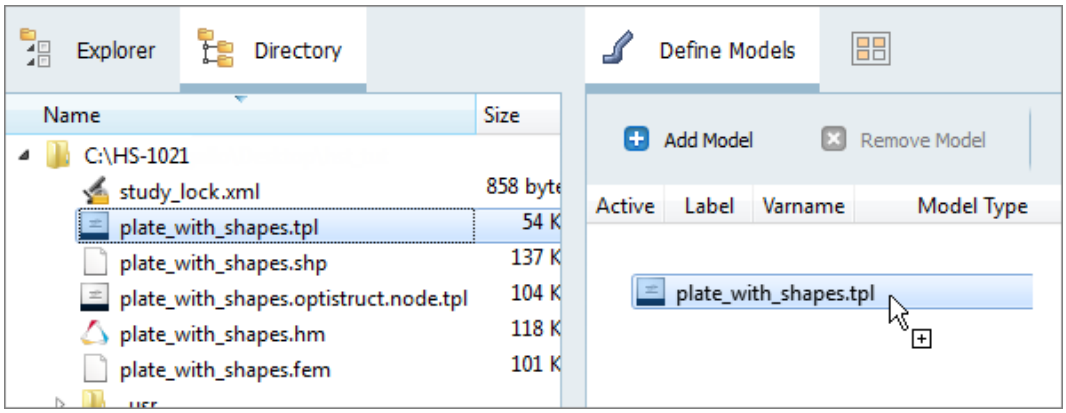


5. Right-click on the highlighted cards and select ***Include Shape*** from the context menu.
6. In the **Shape Template** dialog, open the `plate_with_shapes.optistruct.node.tpl` file.
7. Click ***Save***.
8. In the **Save Template** dialog, save the file as `plate_with_shapes.tpl`.
9. Close the **Editor**.

Step 3: Perform the Study Setup

In this step, you will import the design variables (known as input variables in HyperStudy) created in Step 1: Exporting Shape Variables from HyperMesh to HyperStudy.

1. To start a new study, click ***File*** > ***New*** from the menu bar, or click on the toolbar.
2. In the **HyperStudy – Add** dialog, enter a study name, select a location for the study, and click ***OK***.
3. Go to the **Define Models** step.
4. Add a Parameterized File model.
   a. From the **Directory**, drag-and-drop the `plate_with_shapes.tpl` file into the work area.



b. In the **Solver input file** column, enter `plate.fem`. This is the name of the solver input file HyperStudy writes during any evaluation.

c. In the **Solver execution script** column, select ***OptiStruct (os)***.

	Active	Label	Varname	Model Type	Resource	Solver input file	Solver execution script	Solver input arguments
1	☑	Model 1	m_1	{ } Parameterized File	C:/.../HS-1021/plate_with_shapes.tpl	plate.fem	OptiStruct (os)	${file}

5. Click ***Import Variables***. Three input variables are imported from the `plate_with_shapes.tpl` resource file.
6. Go to the **Define Input Variables** step.
7. Review the input variable's lower and upper bound ranges.
8. Go to the **Specifications** step.

Step 4: Perform a Nominal Run or a System Bound Check

1. In the work area, set the **Mode** to ***Nominal Run*** or ***System Bound Check***.

   A Nominal Run performs one run, and sets the input variable's values to their initial values.

	XTrans	YTrans	Rad
1	0.0000000	0.0000000	0.0000000

   A System Bound Check performs three runs, and sets all of the input variable's values to their initial, lower bound and upper bound values.

	XTrans	YTrans	Rad
1	0.0000000	0.0000000	0.0000000
2	-1.0000000	-1.0000000	-1.0000000
3	1.0000000	1.0000000	1.0000000

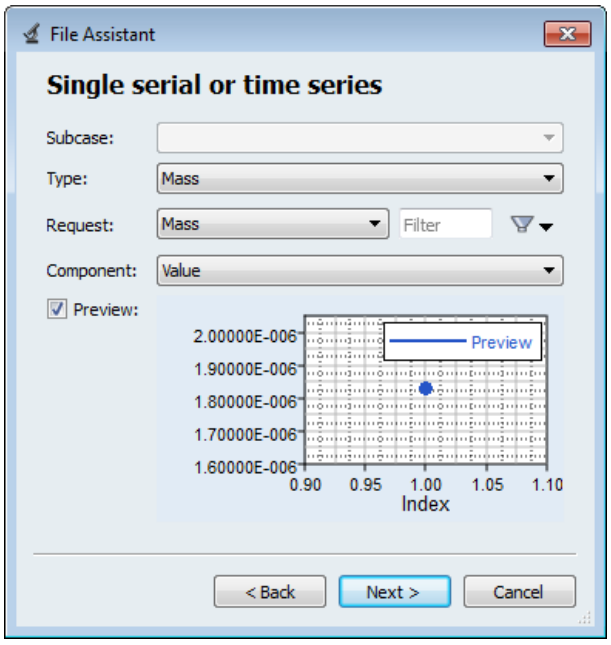
2. Click ***Apply***.

3. Go to the **Evaluate** step.
4. Click ***Evaluate Tasks***.
5. Go to the **Define Output Responses** step.

Step 5: Create and Define Output Responses

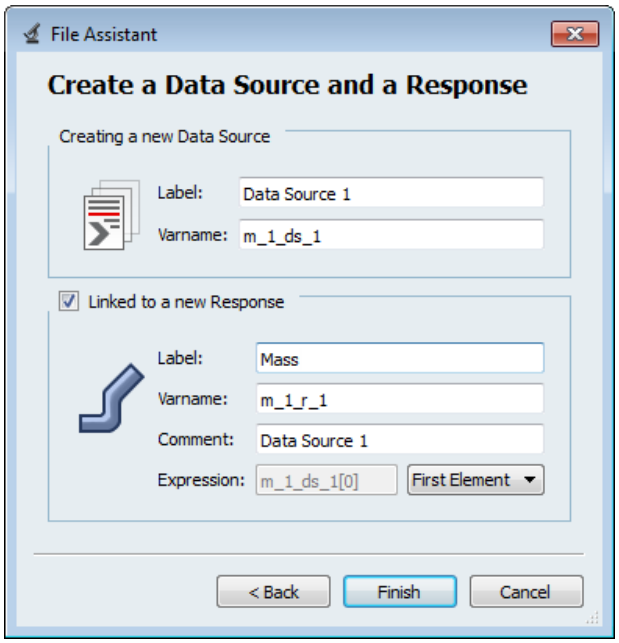
In this step you will create two output responses: Mass and Displacement.

1. Create the Mass output response.
   a. From the **Directory**, drag-and-drop the `plate.out` file, located in `approaches/nom_1/run_00001/m_1`, into the work area.
   b. In the **File Assistant** dialog, set the **Reading technology** to ***Altair® HyperWorks®*** and click ***Next***.
   c. Select **Single item in a time series**, then click ***Next***.
   d. Define the following options, and then click ***Next***.
      - Set **Type** to ***Mass***.
      - Set **Request** to ***Mass***.
      - Set **Component** to ***Value***.



   e. Label the output response Mass.
   f. Set **Expression** to ***First Element***.

      Note: Because there is only a single value in this data source, `[0]` is inserted after `m_1_ds_1`, thereby choosing the first (and only) entry in the data source.



g. Click ***Finish***. The Mass output response is added to the work area.

2. Create the Displacement output response.

   a. From the **Directory**, drag-and-drop the `plate.h3d` file, located in `approaches/nom_1/run_00001/m_1`, into the work area.

   b. In the **File Assistant** dialog, set the **Reading technology** to ***Altair® HyperWorks®*** and click ***Next***.

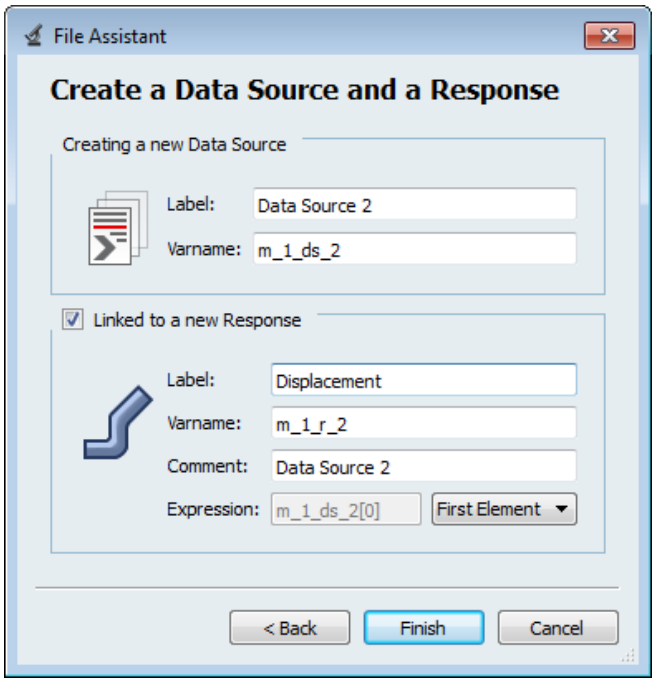
   c. Select ***Single item in a time series***, then click ***Next***.

   d. Define the following options, and then click ***Next***.

      - Set **Subcase** to ***Subcase 1 (Load)***.
      - Set **Type** to ***Displacement (Grids)***.
      - Set **Request** to ***N298***.
      - Set **Component** to ***MAG***.

   e. Label the output response Displacement.

   f. Set **Expression** to ***First Element***.



g. Click ***Finish***. The Displacement output response is added to the work area.

3. Click ***Evaluate*** to extract the output response values.

	Active	Label	Varname	Expression	Value	Comment
1	☑	Mass	m_1_r_1	m_1_ds_1[0] ...	1.83e-06	Data Source 1 ...
2	☑	Displacement	m_1_r_2	m_1_ds_2[0] ...	0.0024906	Data Source 2 ...

4. Proceed to the desired study type (DOE, Optimization, of Stochastic study).

Last modified: v2017.2 (12.1156684)