Altair

HyperWorks

Altair HyperStudy 2019 Tutorials

HS-1020: Working with a Parameterized File Model for Size Variables

HS-1020: Working with a Parameterized File Model for Size Variables

In this tutorial you will learn how to create a template file for size variables, and how to import them to HyperStudy. The input variables are the thickness of each of the three components, defined in the input deck via the PSHELL card. The thickness should be between 0.05 and 0.15; the initial thickness is 0.1 (shown below).

The sample base input template used in this tutorial can be found in `<hst.zip>/HS-1020/`. Copy the tutorial files from this directory to your working directory.

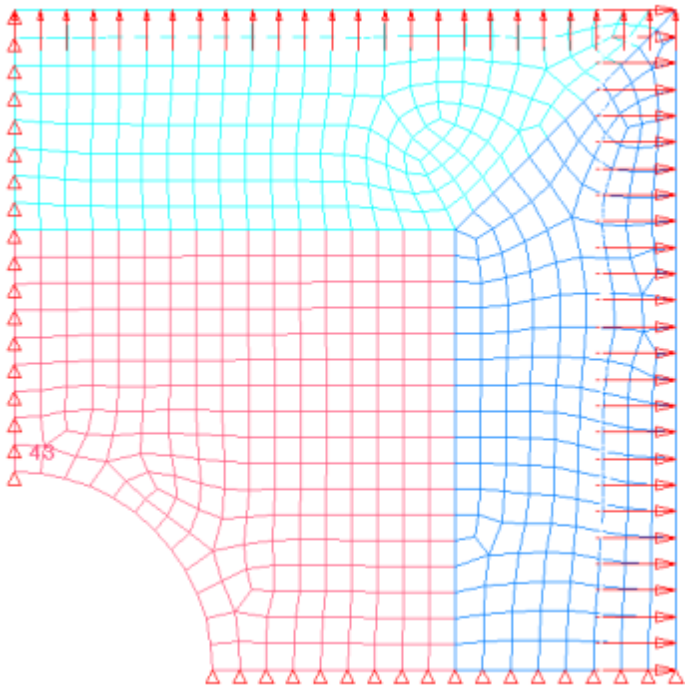
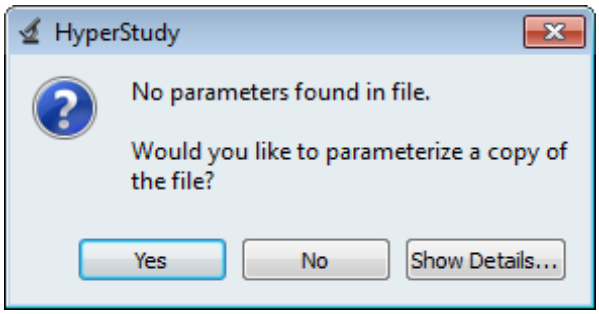


Figure 1: Double Symmetric Plate Model

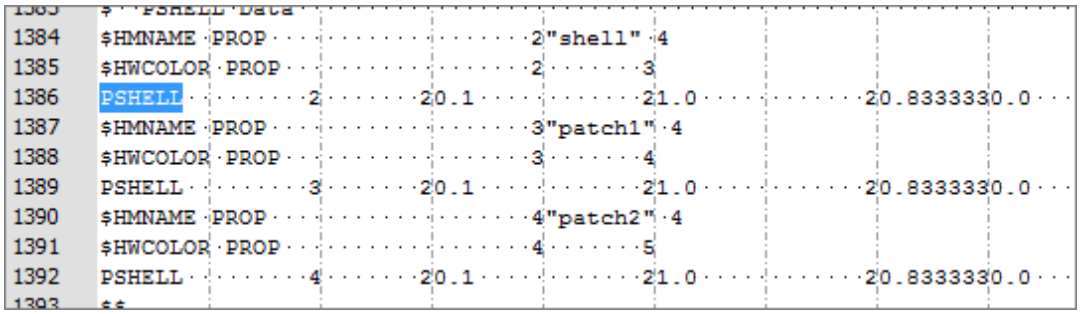
Step 1: Perform the Study Setup

1. Start HyperStudy.
2. To start a new study, click ***File*** > ***New*** from the menu bar, or click on the toolbar.
3. In the **HyperStudy – Add** dialog, enter a study name, select a location for the study, and click ***OK***.
4. Go to the **Define Models** step.
5. Click ***Add Model***.
6. In the **Add - HyperStudy** dialog, select ***Parameterized File*** and click ***OK***.
7. In the **Resource** column, click .
8. In the **Open File** dialog, open the `plate.fem` file. A HyperStudy dialog opens, informing you that there are no parameters found in the file.



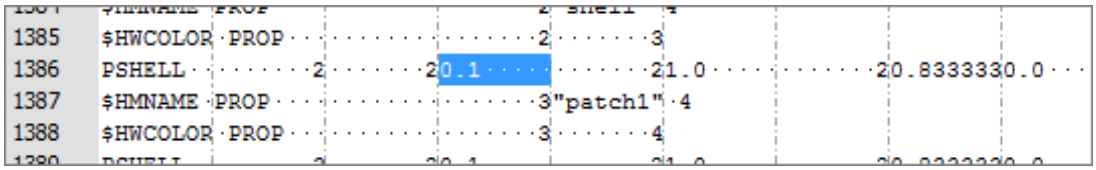
9. Click ***Yes*** to parameterize the file. The `plate.fem` file opens in the **Editor**.
10. In the **Find** area, enter `PSHELL`.
11. Click ▶ three times. Three PSHELL cards containing the shell thickness of each component in the fourth field are highlighted.

    Each field is eight characters long. The numbers need to be replaced with formatted Templex statements to mark an input variable.

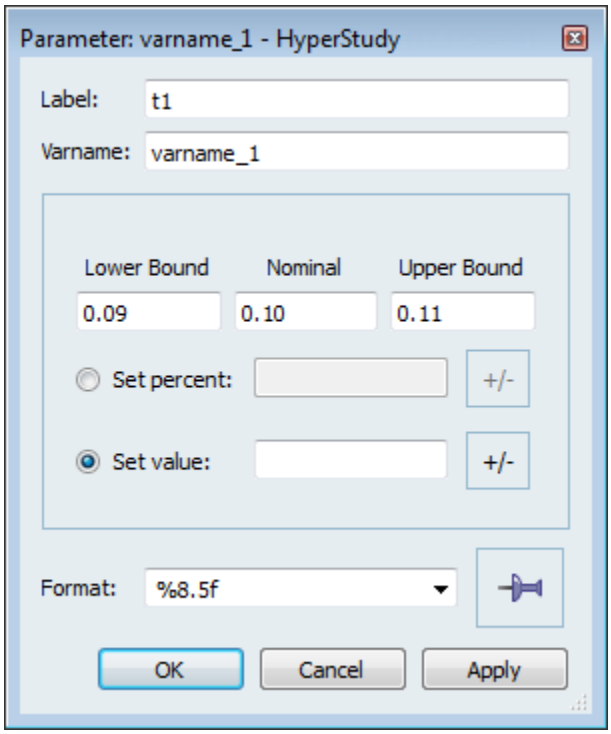


12. Starting at 0.1, highlight the first eight fields for thickness.

    Tip: Quickly highlight 8-character fields by pressing ***CTRL*** to activate the **Selector** (set to 8 characters) and then clicking the value.



13. Right-click on the highlighted fields and select ***Create Parameter*** from the context menu.
14. In the **Parameter - varname_1** dialog, **Label** field, enter `t1`.
15. In the **Format** field, enter `%8.5f`.
16. Click ***OK***.



17. Click ***OK*** to close the **Editor**.
18. In the **Save Template** dialog, save the template file as `plate.tpl`. HyperStudy automatically populates the **Resource** column with the `plate.tpl` file, and the **Solver input file** column with the solver input file name `plate.fem`.
19. In the **Solver execution script** column, select ***OptiStruct (os)***.

	Active	Label	Varname	Model Type	Resource	Solver input file	Solver execution script	Solver input argumer
1	☑	Model 1	m_1	{ } Parameterized File	C:/.../HS-4410/plate.tpl	plate.fem	OptiStruct (os)	${file}

20. Click ***Import Variables***. One input variables is imported from the `plate.tpl` resource file.
21. Go to the **Define Input Variables** step.
22. Review the input variable's lower and upper bound ranges.
23. Go to the **Specification** step.

Step 2: Perform the Nominal Run

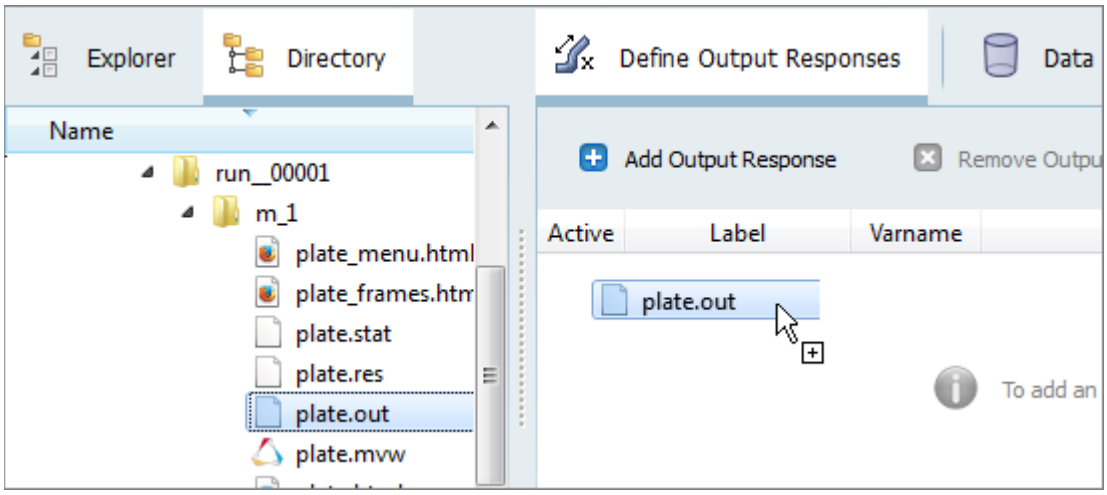
1. In the work area, set the **Mode** to ***Nominal Run***.
2. Click ***Apply***.
3. Go to the **Evaluate** step.
4. Click ***Evaluate Tasks***.

5. Go to the **Define Output Responses** step.

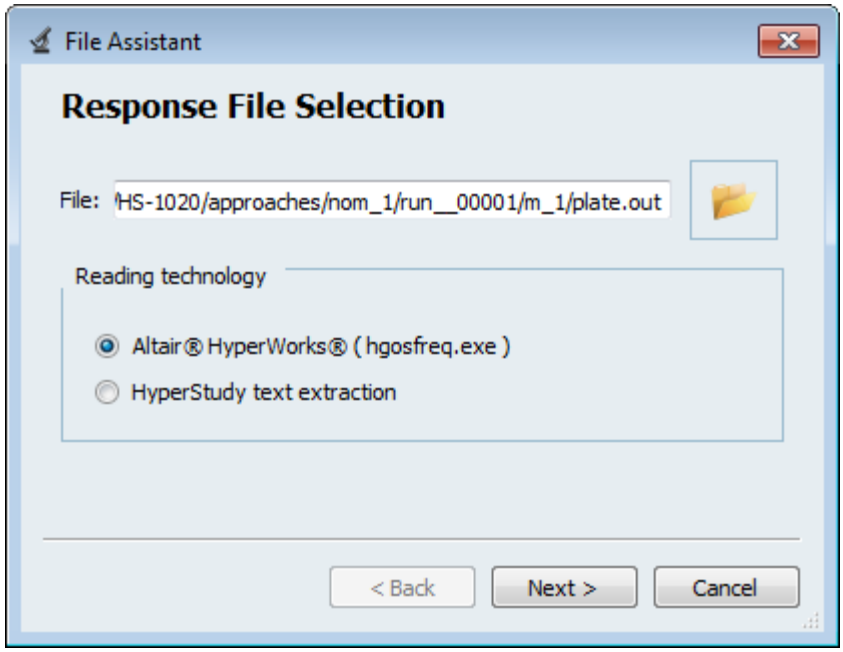
Step 3: Create and Define Output Responses

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.



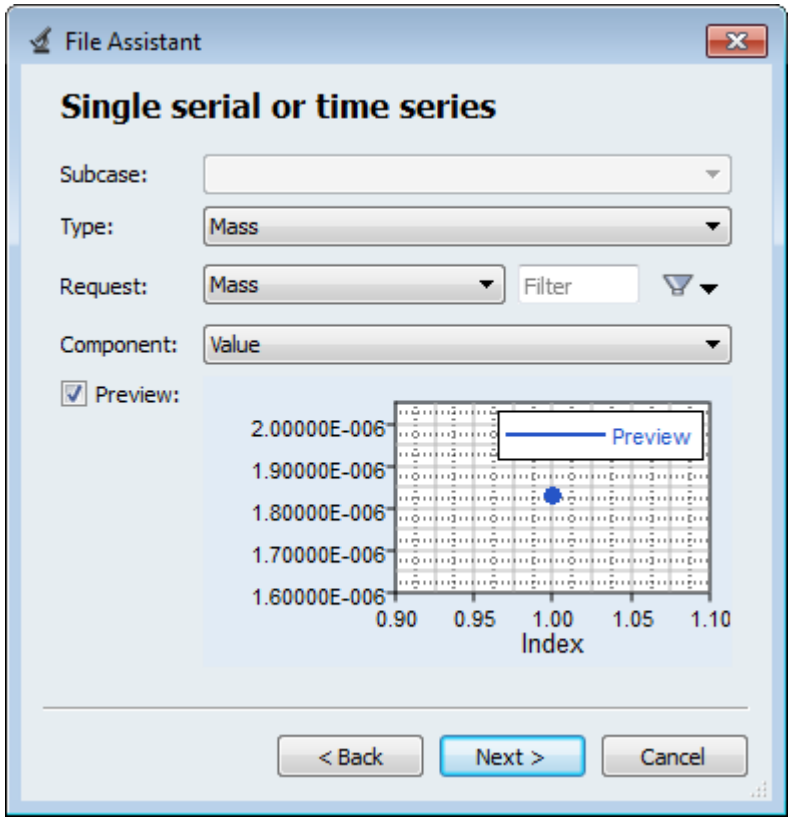
   b. In the **File Assistant** dialog, set the **Reading technology** to ***Altair® HyperWorks®*** and click ***Next***.



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

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

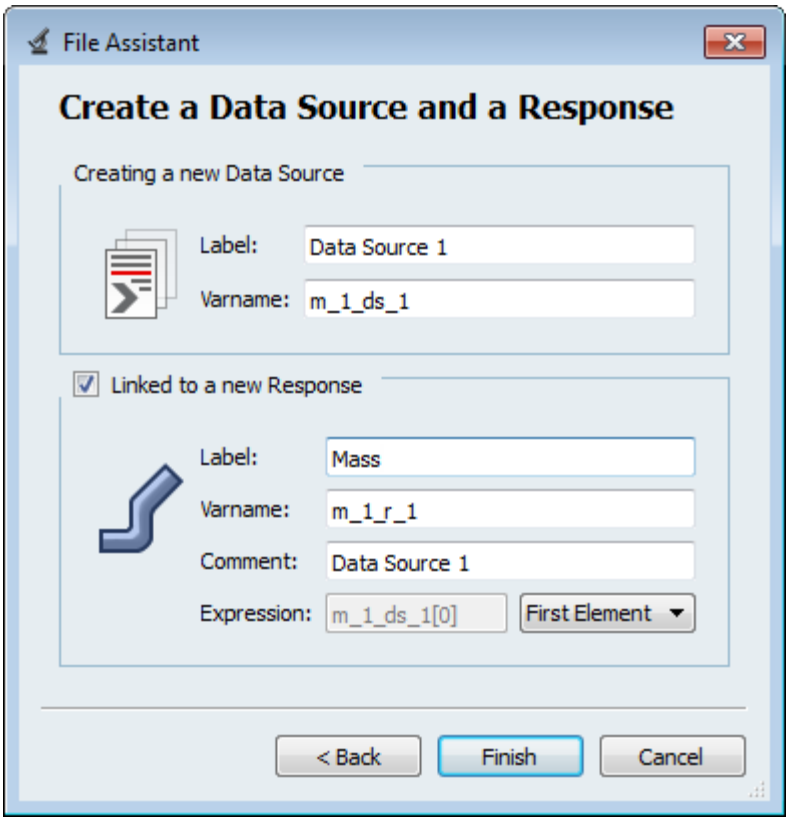
- Set **Type** to ***Mass***.
- Set **Request** to ***Mass***.
- Set **Component** to ***Value***.



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 displayed in 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***.

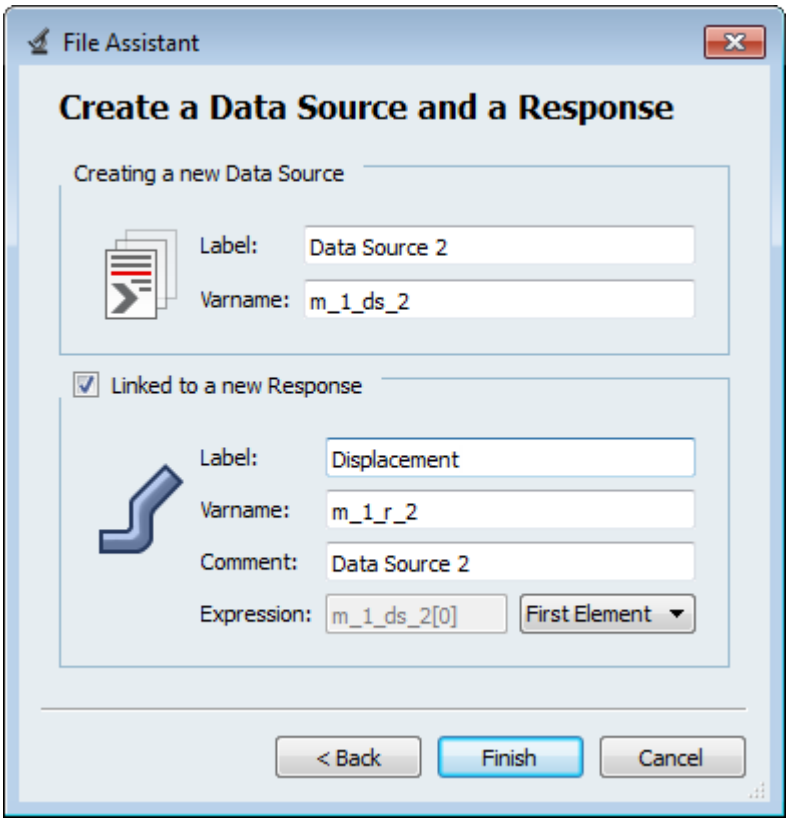
   c. Select ***Single item in a time series***, then click ***Next***.

   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)