Altair

HyperWorks

Altair HyperStudy 2019 Tutorials

HS-1070: Defining Discrete Size Variables with Conditional Linking for use in HyperStudy

HS-1070: Defining Discrete Size Variables with Conditional Linking for use in HyperStudy

This tutorial outlines the procedure for defining discrete input variables that are conditionally linked. In this tutorial, beam thickness and material model values are used as variables. Material model values are linked to the selection of the material to be used. The variables you will use in this tutorial include: Shell Thickness, Young's Modulus, Poisson's, Ratio, and Density. The output responses you will use in this tutorial includes: Mass.

The `beam.fem` model file that you will use in this tutorial can be found in `<hst.zip>/HS-1070/`. Copy the tutorial files from this directory to your working directory.

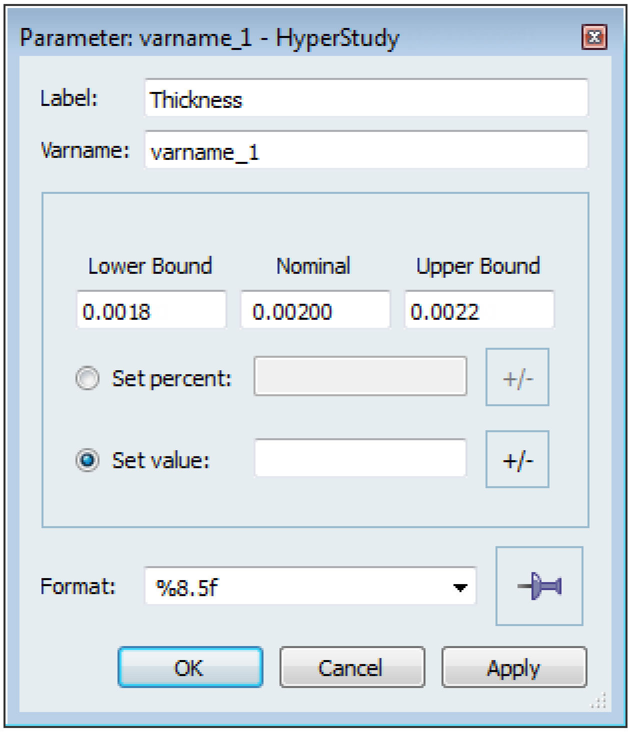
Step 1: Create the Base Import Template in HyperStudy

1. Start HyperStudy.
2. From the menu bar, click ***Tools*** > ***Editor***. The **HyperStudy - Editor** opens.
3. In the **File** field, open the `beam.fem` file.
4. In the **Find** area, enter `PSHELL`.
5. Click ▶ until you find the PSHELL card.
6. In the same line as PSHELL, highlight the value 0.002 for the PSHELL thickness as indicated in the image below.

   Note: In an OptiStruct deck, each field within a card is 8 characters long. Properly select the value for the PSHELL thickness by selecting 0.002 and the three spaces that follow.

```
$$
$$  PSHELL Data
$$
$
$HMNAME PROP                  1"PSHELL" 4
$HWCOLOR PROP                 1       3
PSHELL         1       10.002          1               1       0.0
$$
$$  MAT1 Data
$$
$HMNAME MAT                   1"MAT1" "MAT1"
$HWCOLOR MAT                  1       3
MAT1           12.1E+11        0.3     7820.0
$$
```

7. Right-click on the highlighted fields and select ***Create Parameter*** from the context menu.
8. In the **Parameter - varname_1** dialog, **Label** field, enter `Thickness`.
9. Set the **Upper bound** to `0.0022`, the **Nominal** to `0.0020`, and the **Lower bound** to `0.0018`.
10. Set the **Format** to `%8.5f`.
11. Click ***OK***.



12. Add three more input variables to the template file from the MAT1 card using the information provided in the table below.

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

Parameter	Selection Notification	Selection	Input Variable Label	Lower Bound & Upper Bound	Format
Young's Modulus	Line number: 8267 Fields: 16 to 24	2.1E+11 & 1 space	Young	Lower Bound: 7E+10 Upper Bound: 2.1E+11	%8.1e
Poisson's Ratio	Line number: 8267 Fields: 32 to 40	0.3 + 5 spaces	Poisson	Keep default values	%8.5f
Density	Line number: 8267 Fields: 40 to 48	7820.0 + 2 spaces	Density	Keep default values	%8.3f

13. Click ***Save***.
14. In the **Save Template** dialog, save the file as `beam.tpl`.
15. Close the **HyperStudy - Editor** dialog.

16. In a text editor, open the `beam.tpl` file.
17. Replace:

```
{parameter(varname_3, "Poisson",  0.30000,  0.27000,
0.33000)}
{parameter(varname_4, "Density", 7820.000, 7038.000,
8602.000)}
```

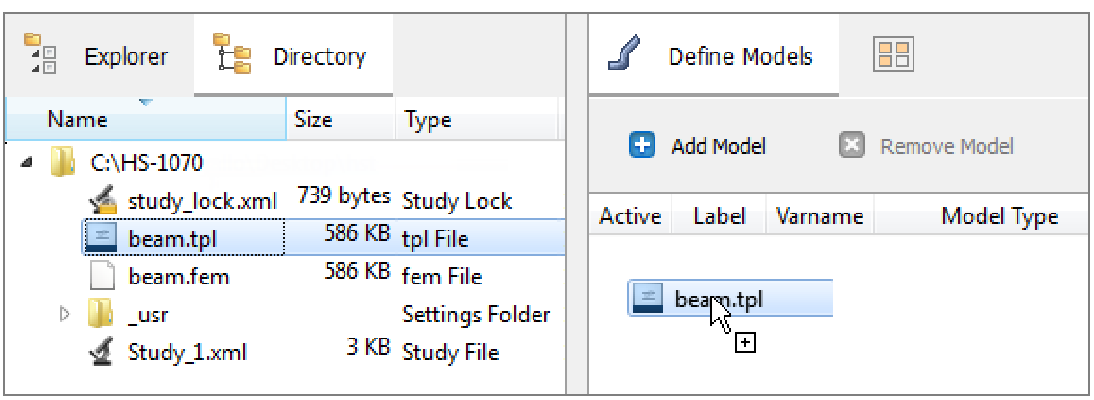
with:

```
{if (varname_2==2.1E+11)}
            {varname_3=0.3}
            {varname_4=7820}
{elseif (varname_2==7E+10)}
            {varname_3=0.33}
            {varname_4=2700}
{endif}
```

18. Save the `beam.tpl` file and close the text editor.

Step 2: Perform the Study Setup

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 `beam.tpl` file into the work area.

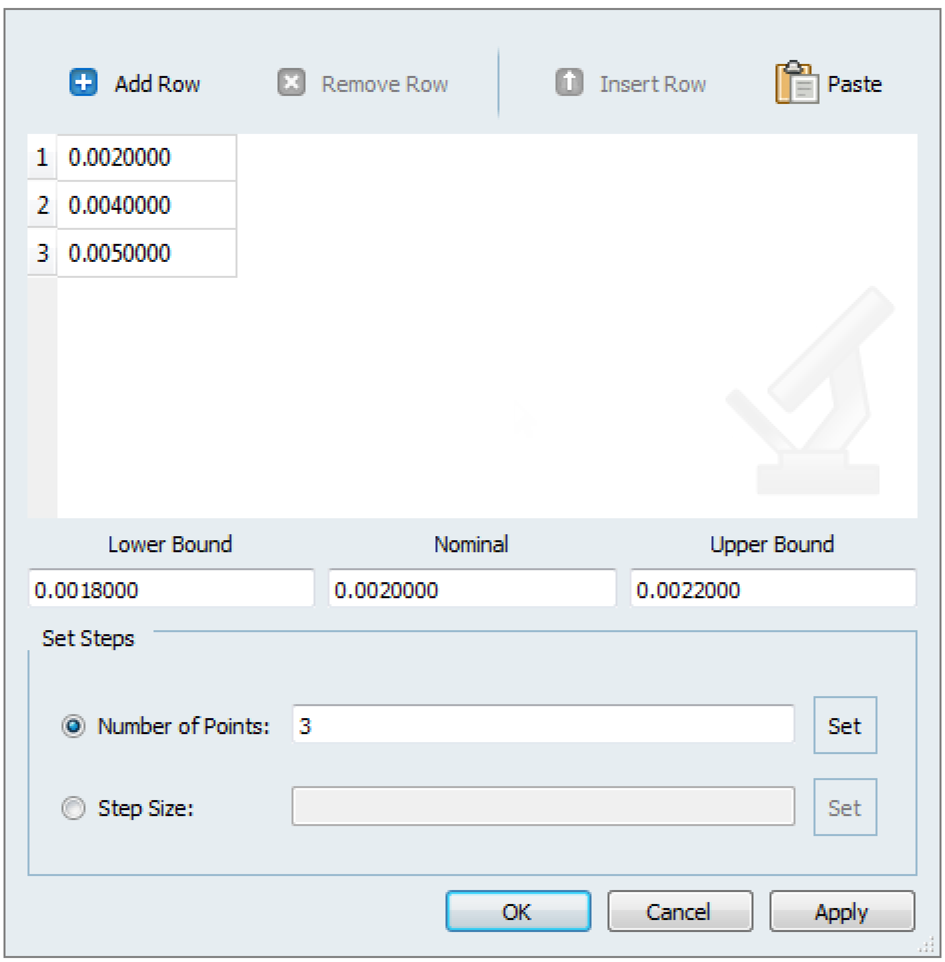


   b. In the **Solver input file** column, enter `beam.fem`. This is the name of the solver input file HyperStudy writes during any evaluation.

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-1070/beam.tpl	beam.fem	OptiStruct (os)	${file}

5. Click **Import Variables**. Two input variables are imported from the `beam.tpl` resource file.
6. Go to the **Define Input Variables** step.
7. Click the **Modes** tab.
8. For the input variable **Thickness**, set **Mode** to ***Discrete***.
9. In the **Values** column for the input variable **Thickness**, click **...**. A pop-up window opens.
10. In the **Number of Points** field, enter `3`.
11. Click ***Set***.
12. Change the values in the table to the following: `0.002`, `0.004`, and `0.005`.



13. Click ***OK***.

14. For the input variable **Young**, set the **Mode** to ***Discrete***.
15. In the **Value** column of the input variable **Young**, click **...**. A pop-up window opens.
16. In the **Number of Points** field, enter `2`.
17. Click ***Set***.
18. Change the values in the table to `7E+10` and `2.1E+11`.
19. Click ***OK***.
20. Go to the **Specifications** step.

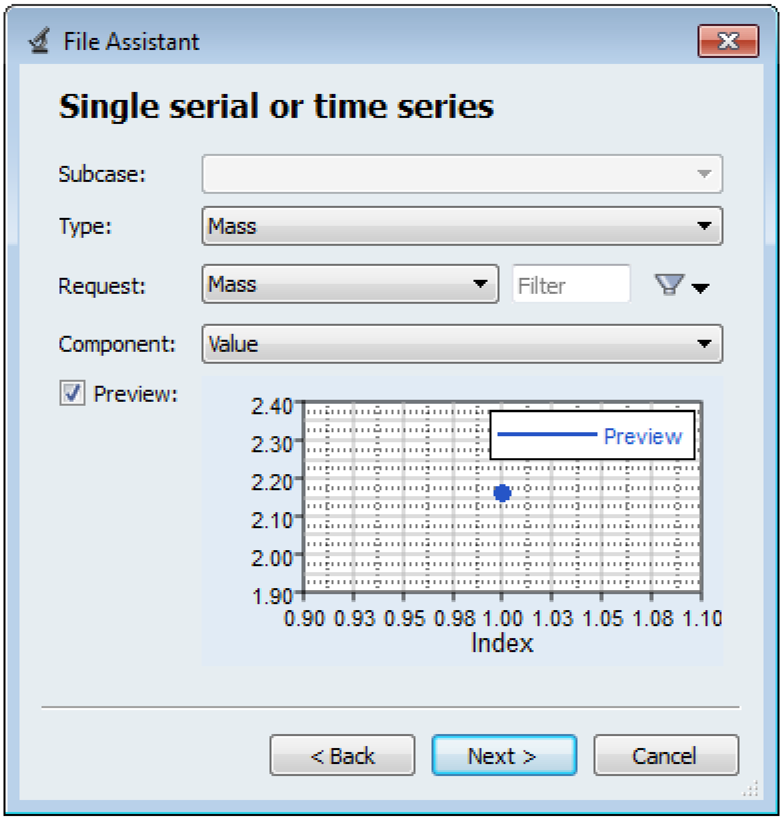
Step 3: Perform the Nominal Run

1. In the work area, set the **Mode** to ***Nominal Run***.
2. Click ***Apply***.
3. Go to the **Evaluate** step.
4. Click ***Evaluate Tasks***. The run is executed using OptiStruct, and all input files are written to the study directory.
5. Go to the **Define Output Responses** step.

Step 4: Create and Define Output Responses

In this step you will create one output response, Mass.

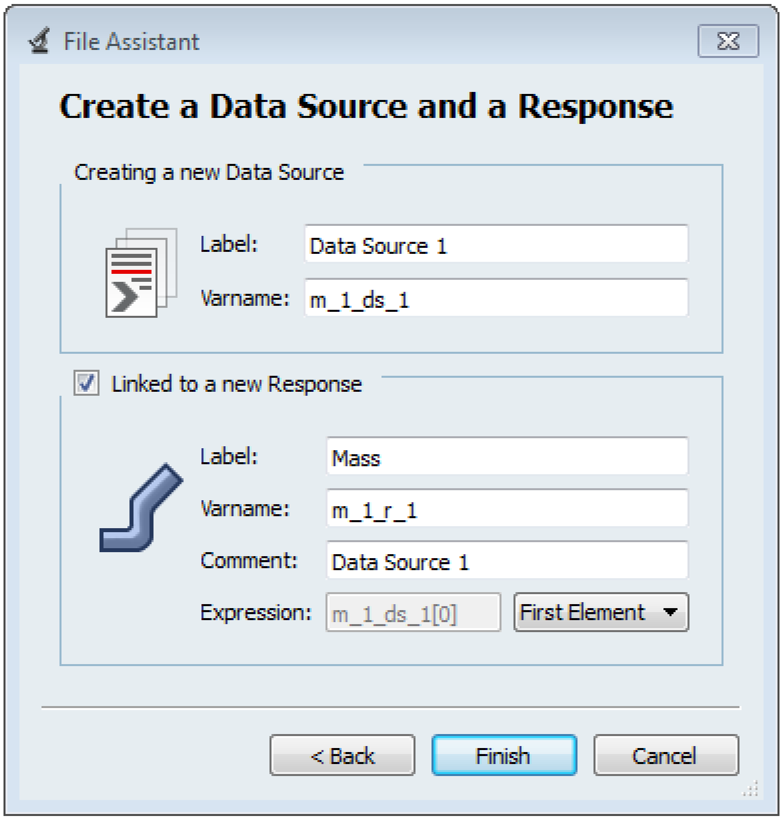
1. From the **Directory**, drag-and-drop the `beam.out` file, located in `approaches/nom_1/run_00001/m_1`, into the work area.
2. In the **File Assistant** dialog, set the **Reading technology** to ***Altair® HyperWorks®*** and click ***Next***.
3. Select **Single item in a time series**, then click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



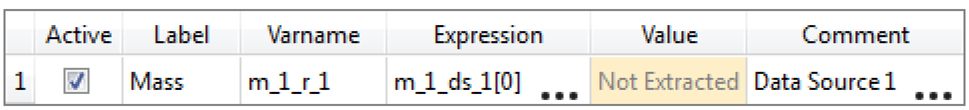
5. Label the output response Mass.
6. Set **Expression** to ***First Element***.

Note: Since the mass is a scalar quantity, the only component of the data source is index zero, therefore a scalar mass value is properly written in the **Expression** field as `m_1_ds_1[0]`. Multi-axis vectors may be accessed through each individual axis by using the corresponding index in the vector array.

The mass value can be accessed directly out of the `beam.out` file and does not require any mathematical operations before being used as an output response for your study. The **Expression Builder** is capable of performing a full complement of mathematical functions on an arbitrary number of vector solutions in order to build an output response.



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



	Active	Label	Varname	Expression	Value	Comment
1	☑	Mass	m_1_r_1	m_1_ds_1[0] ...	Not Extracted	Data Source 1 ...

8. Click ***Evaluate*** to extract the output response value.

Last modified: v2017.2 (12.1156684)