Altair

HyperWorks

# Altair HyperStudy 2019 Tutorials

## HS-1090: Defining Discrete Size Variables with the Lookup Model

# HS-1090: Defining Discrete Size Variables with the Lookup Model

In this tutorial you will define discrete size input variables with the Lookup model. You will establish links between the input variables imported from a parameterized file with the output responses imported from a `.csv` file using the Lookup model.

## Model Files

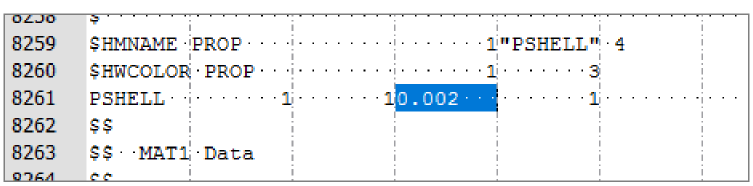
The files used in this tutorial can be found in `<hst.zip>/HS-1090/`. Copy the `beam.fem` file and the `material_prop.csv` file to your working directory.

## Exercise

### Step 1: Perform the Study Set Up

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. In the work area, click **Add Model**.
    b. In the **Add – HyperStudy** dialog, select **Parameterized File** and click **OK**.
    c. In the Resource column, click .
    d. In the **Open File** dialog, navigate to your working directory and open the `beam.fem` file.
    e. In the **HyperStudy** dialog, click **Yes** to parameterize the file.
    f. In the **Editor – HyperStudy** dialog, Find area, enter `PSHELL` and click until you find the PSHELL card.
    g. In the same line as PSHELL, highlight the value `0.002` for the PSHELL thickness.

    **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.



h. Right-click on the highlighted fields and select **Create Parameter** from the context menu.

i. In the **Parameter - varname_1** dialog define the parameter and click **OK**.

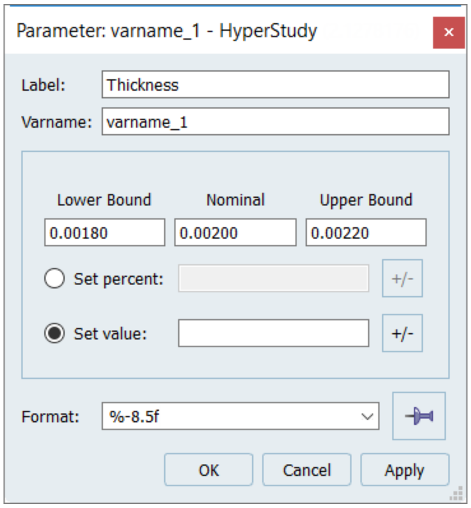
   i. In the Label field, enter `Thickness`.

   ii. For the Upper bound, enter `0.0022`.

   iii. For the Nominal value, enter `0.0020`.

   iv. For the Lower bound, enter `0.0018`.

   v. In the Format field, enter `%-8.5f`.




j. 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<br>Fields: 16 to 24 | 2.1E+11 & 1 space | Modulus | Lower Bound: 1.9E+011<br>Upper Bound: 2.3E+011 | %-8.1E |
| Poisson's Ratio | Line number: 8267<br>Fields: 32 to 40 | 0.3 + 5 spaces | Poisson | Lower Bound: 0.27<br>Upper Bound: 0.33 | %-8.2f |

| | | | | | |
|---|---|---|---|---|---|
| Density | Line number: 8267<br>Fields: 40 to 48 | 7820.0 + 2 spaces | Density | Lower Bound: 7038<br>Upper Bound: 8602 | %-8.2f |

k. Click **Save**.

l. In the **Save Template** dialog, save the file as `beam.tpl`.

m. Click **OK** to close the **Editor – HyperStudy** dialog.

The Resource column displays the `beam.tpl` file, and the Solver input file column displays `beam.fem`, which is the name of the solver input file HyperStudy writes during any evaluation.

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

5. Add a Lookup model.

   a. In the work area, click **Add Model**.

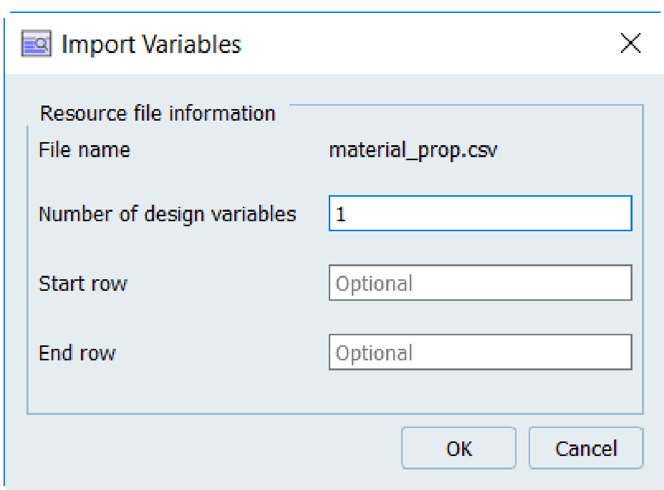
   b. In the **Add – HyperStudy** dialog, select **Lookup** and click **OK**.

   c. In the Resource column, click .

   d. In the **HyperStudy – Load model resource** dialog, navigate to your working directory and open the `material_prop.csv` file.

6. Click **Import Variables**.

7. In the **Import Variables** dialog, Number of design variables field, enter `1` and then click **OK**.




8. Go to the **Define Input Variables** step.

9. Review the four input variables that were imported from the `beam.tpl` file in the Parameterized File model, and the one input variable that was imported from the `material_prop.csv` file in the Lookup model.

**Notice:** The label of fifth input variable has the same label as the first column in the `material_prop.csv` file, that is Material.

The Lookup model automatically populates the input variables based on the number you provided, and you can now identify the material by strings.

| | Active | Label | Varname | Lower Bound | Nominal | Upper Bound |
|---|---|---|---|---|---|---|
| 1 | ☑ | Thickness | m_1_varname_1 | 0.0018000 ... | 0.0020000 ... | 0.0022000 ... |
| 2 | ☑ | Modulus | m_1_varname_2 | 1.90e+11 ... | 2.10e+11 ... | 2.30e+11 ... |
| 3 | ☑ | Poisson | m_1_varname_3 | 0.2700000 ... | 0.3000000 ... | 0.3300000 ... |
| 4 | ☑ | Density | m_1_varname_4 | 7038.0000 ... | 7820.0000 ... | 8602.0000 ... |
| 5 | ☑ | Material | var_5 | Aluminium | Steel | Steel |

10. Go to the **Specifications** step.

### Step 2: Perform the Nominal Run

1. In the work area, set the **Mode** to **Nominal Run**.

| | Mode | Label | Varname | Details |
|---|---|---|---|---|
| 1 | ◉ | Nominal Run | Nom | Run system at nominal values |
| 2 | ○ | System Bounds Check | Chk | Run system at nominal values, then lower and upper values |
| | | Show more ... | | |

2. Click **Apply**.
3. Go to the **Evaluate** step and click **Evaluate Tasks**.
4. Go to the **Define Output Responses** step.

### Step 3: Define Output Responses

1. Review the three output responses that were automatically added to the study from in Lookup model, which correspond to the nominal values from the `material_prop.csv` file.

   **Notice:** The output response labels are the same as the labels in the `material_prop.csv` file.

| | Active | Label | Varname | Expression | Value | Comment |
|---|---|---|---|---|---|---|
| 1 | ☑ | Modulus | r_1 | ds_1[0] ... | 2.10e+11 | ... |
| 2 | ☑ | Poisson | r_2 | ds_2[0] ... | 0.3000000 | ... |
| 3 | ☑ | Density | r_3 | ds_3[0] ... | 7820.0000 | ... |

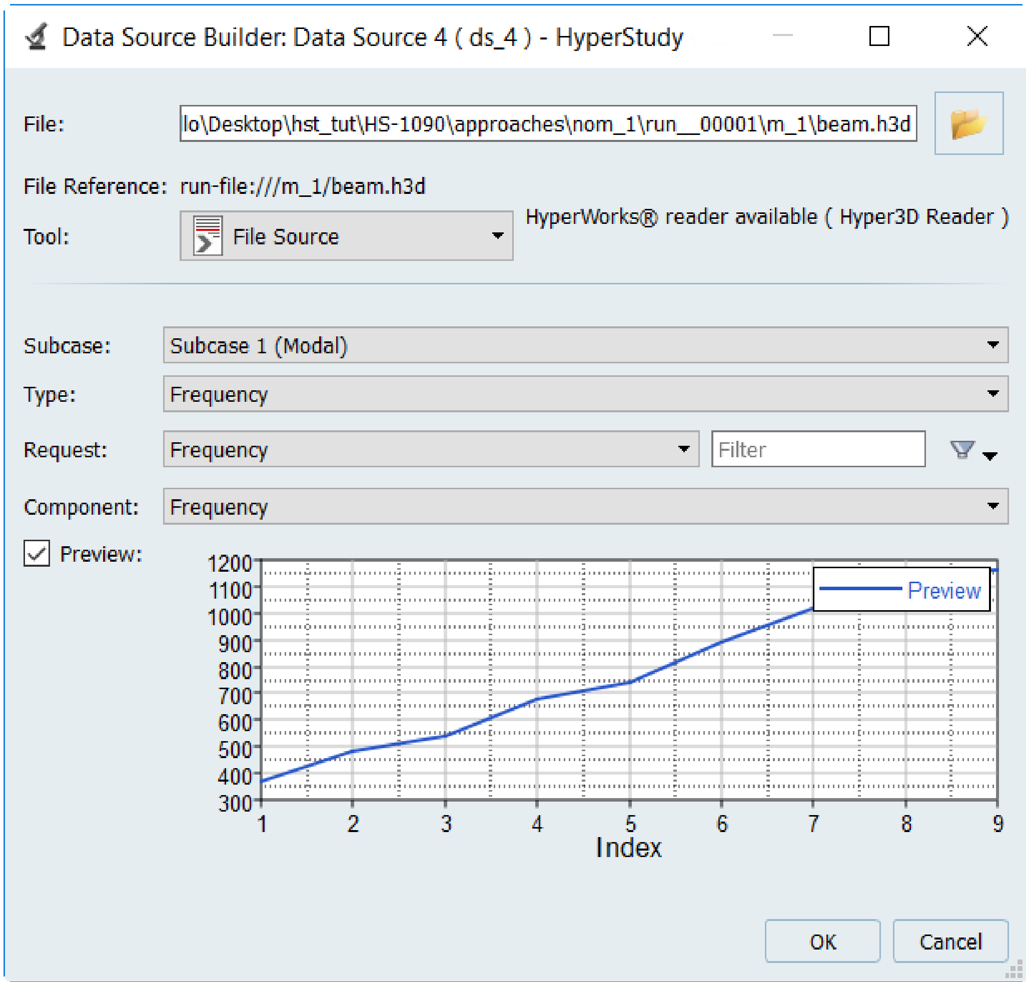
2. Click the **Data Sources** tab.
3. Click **Add Data Source** to add two data sources.
4. Define Data Source 4.
   a. In the File field for Data Source 4, click (**...**).
   b. In the **Data Source Builder** dialog, File field, navigate to the `approaches\nom_1\run__00001\m_1` directory inside your working directory and open the `beam.h3d` file.

c. Set Tool to **File Source**.

d. Set Subcase to **Subcase 1 (Modal)**.

e. Set Type, Request, and Component to **Frequency**.

f. Click **OK**.




5. Define Data Source 5.

   a. In the File field for Data Source 5, click (**...**).

   b. In the **Data Source Builder** dialog, File field, navigate to the `approaches\nom_1\run__00001\m_1` directory inside your working directory and open the `beam.h3d` file.

   c. Set Tool to **Read Simulation**.

   d. Set Subcase to **Subcase 2 (Static)**.

   e. Set Type to **Displacement (Grids)**.

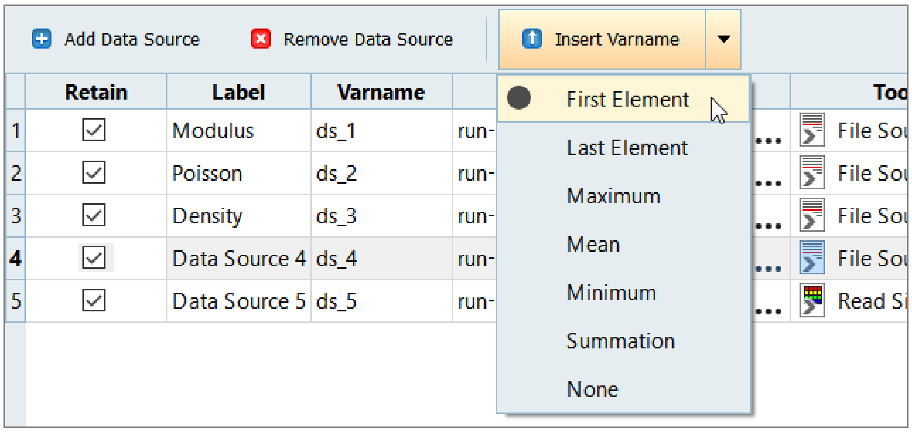
   f. For Request, set Start to **First Request** and End to **Last Request**.

   g. For Components, select **MAG**.

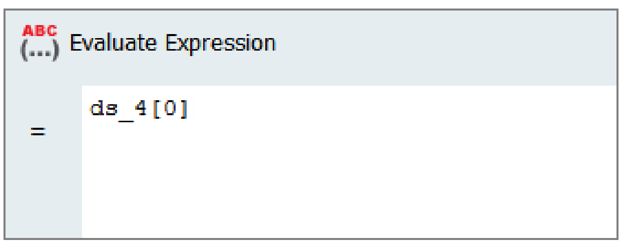
   h. Set Time to **All**.

   i. Click **OK**.

6. Click the **Define Output Responses** tab.
7. Define the 1st_natural_freq output response.
    a. Click **Add Output Response**.
    b. In the Label field, enter `1st_natural_freq`.
    c. In the Expression field, click (**...**).
    d. In the **Expression Builder** dialog, click the **Data Sources** tab.
    e. From the list of data sources, click **Data Source 4**.
    f. Next to Insert Varname, click ▼ and select **First Element**.




    g. Click **Insert Varname**.
    The Evaluate Expression field displays ds_4[0].




    h. Click **OK**.
8. Define the Max_disp output response.
    a. Click **Add Output Response**.
    b. In the Label field, enter `Max_disp`.
    c. In the Expression field, click (**...**).
    d. In the **Expression Builder** dialog, click the **Data Sources** tab.
    e. From the list of data sources, click **Data Source 5**.
    f. Next to Insert Varname, click ▼ and select **Maximum**.
    g. Click **Insert Varname**.
    The Evaluate Expression field displays max(ds_5).
    h. Click **OK**.

9. Click **Evaluate** to extract the response values.

| | Active | Label | Varname | Expression | Value | Comment |
|---|---|---|---|---|---|---|
| 1 | ☑ | Modulus | r_1 | ds_1[0] ... | 2.10e+11 | ... |
| 2 | ☑ | Poisson | r_2 | ds_2[0] ... | 0.3000000 | ... |
| 3 | ☑ | Density | r_3 | ds_3[0] ... | 7820.0000 | ... |
| 4 | ☑ | 1st_natural_freq output | r_4 | ds_4[0] ... | 368.83109 | ... |
| 5 | ☑ | Max_disp | r_5 | max(ds_5) ... | 0.0027399 | ... |

### Step 4: Link the Input Variables and Output Responses

In this step you will establish links between the input variables imported from the `beam.tpl` file in the Parameterized File model with the output responses imported from the `material_prop.csv` file in the Lookup model.

1. Go to the **Define Input Variables** step.
2. Click the **Links** tab.
3. In the Expression field for Modulus, click (**...**).
4. In the **Expression Builder**, click the **Output Responses** tab.
5. From the list of output responses, select **Modulus**.
6. Click **Insert Varname**.
7. Click **OK** to close the **Expression Builder**.

   The Modulus input variable is now linked to the Modulus output response.

8. Link the Poissons input variable to the Poissons output response and the Density input variable to the Density output response.

| | Active | Label | Varname | Expression |
|---|---|---|---|---|
| 1 | ☑ | Thickness | m_1_varname_1 | ... |
| 2 | ☑ | Modulus | m_1_varname_2 🔗 | r_1 ... |
| 3 | ☑ | Poisson | m_1_varname_3 🔗 | r_2 ... |
| 4 | ☑ | Density | m_1_varname_4 🔗 | r_3 ... |
| 5 | ☑ | Material | var_5 | ... |

9. Go to the **Specifications** step and click **Apply**.
10. In the **HyperStudy** dialog, click **Yes** to overwrite the run matrix.
11. Go to the **Evaluate** step and click **Evaluate Tasks** to re-evaluate the setup.

### Step 5: Run a DOE Study

1. In the Explorer, right-click and select **Add** from the context menu.
2. In the **Add – HyperStudy** dialog, select **DOE** and click **OK**.
3. Go to the **Specifications** step.

4. Verify that the Mode is set to **Modified Extensible Lattice Sequence (Mels)** and that the Number of Runs is set to 6.
5. Click **Apply**.
6. Go to the **Evaluate** step and click **Evaluate Tasks**.
7. Go to the **Post-Processing** step.
8. Click the **Summary** tab.

**Notice:** The output responses (material property numbers) from the `.csv` file are linked to the input variables (material property set in the FEA deck), and are now controlled in the categorical input variable Material.

Any number of material data can be added using a library, without requiring you to explicitly create "if" conditions in a `.tpl` file. This is the advantage of using Lookup model in this case.

| | Thickness | Modulus | Poisson | Density | Material | Modulus | Poisson | Density | |
|---|---|---|---|---|---|---|---|---|---|
| 1 | 0.0018800 | 7.00e+10 | 0.3300000 | 2700.0000 | Aluminium | 7.00e+10 | 0.3300000 | 2700.0000 | 361 |
| 2 | 0.0019600 | 2.10e+11 | 0.3000000 | 7820.0000 | Steel | 2.10e+11 | 0.3000000 | 7820.0000 | 368 |
| 3 | 0.0020400 | 7.00e+10 | 0.3300000 | 2700.0000 | Aluminium | 7.00e+10 | 0.3300000 | 2700.0000 | 362 |
| 4 | 0.0021200 | 2.10e+11 | 0.3000000 | 7820.0000 | Steel | 2.10e+11 | 0.3000000 | 7820.0000 | 369 |
| 5 | 0.0018160 | 2.10e+11 | 0.3000000 | 7820.0000 | Steel | 2.10e+11 | 0.3000000 | 7820.0000 | 366 |
| 6 | 0.0018960 | 7.00e+10 | 0.3300000 | 2700.0000 | Aluminium | 7.00e+10 | 0.3300000 | 2700.0000 | 361 |