



Altair

HyperWorks

HS-1506: Material Calibration with a Curve Difference Integral

In this tutorial you will learn how to register a Compose, Python, or Templex function in HyperStudy. The objective of this tutorial is to find the RADIOSS material parameter values so that the stress-strain curve of the tensile test simulation matches the tensile test experimental curve. HS-4200: Material Calibration Using System Identification provides an alternative method to setup this problem using system identification.

By the end of this tutorial, you will know how to:

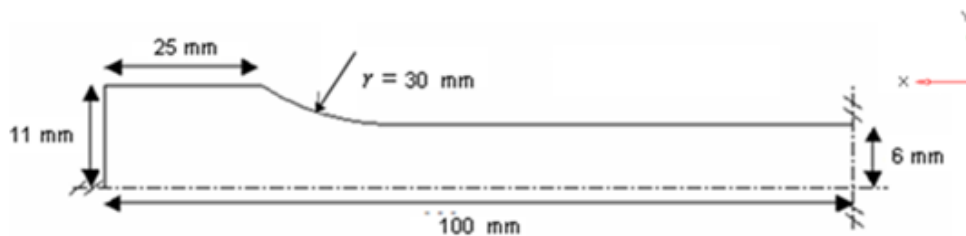
- Create an input template from a RADIOSS file using the HyperStudy - Editor
- Register a Compose, Python, or Templex function
- Setup a study
- Run a system identification optimization study

The sample base input template used in this tutorial can be found in `<hst.zip>/HS-1506/`. Copy all of the files from this directory to your working directory. The `.py` and `.oml` file contain a function that calculates the area between two curves.

Description of the Model

A quarter of a standard tensile test specimen is modeled using symmetry conditions. A traction is applied to a specimen via an imposed velocity at the left-end.

The units are: mm, ms, g, N, MPa.



Geometry of the Tensile Specimen (One Quarter of the Specimen is Modeled)



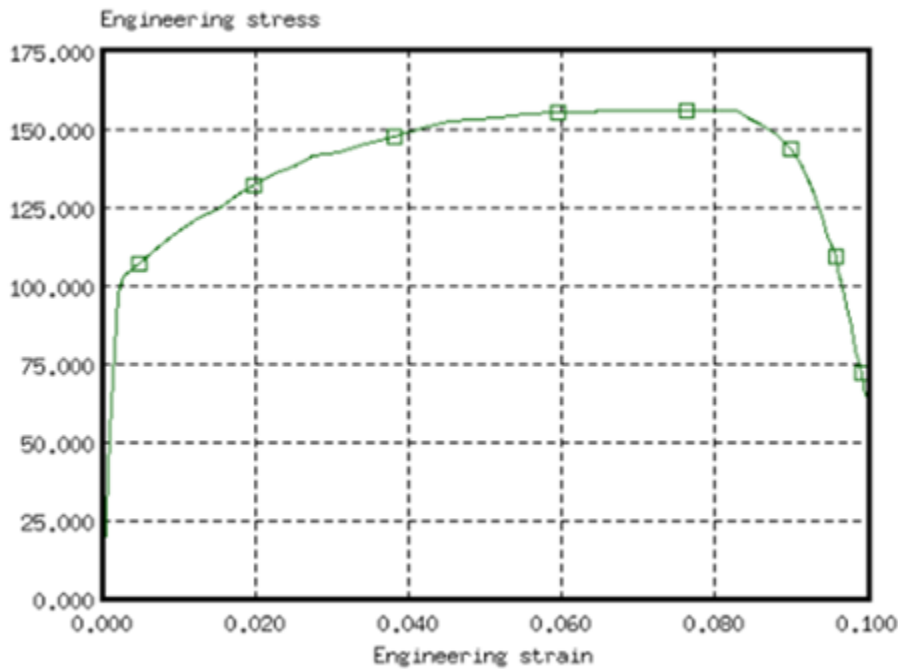
Sections of Node Saved for Time History

The material to be characterized is a 6063 T7 Aluminum. It has an isotropic elasto-plastic behavior which can be reproduced by a Johnson-Cook model without damage (RADIOSS Block Law2), defined as follows:

$$\sigma = \underbrace{(a + b\varepsilon_p^n)}_{\text{Influence of plastic strain}} \underbrace{\left(1 + c \ln \frac{\dot{\varepsilon}}{\dot{\varepsilon}_0}\right)}_{\text{Influence of strain rate}} \underbrace{(1 - T^{*m})}_{\text{Influence of temperature change}}$$

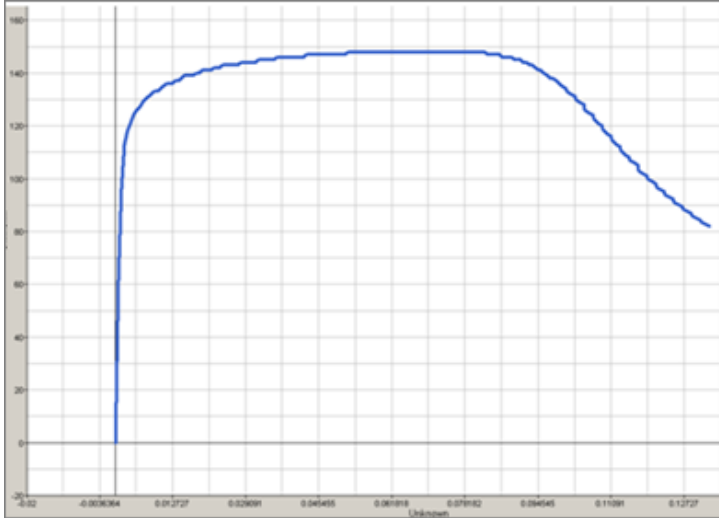
- σ = Stress level n = Hardening Exponent
- ε_p = Plastic strain c = Strain rate coefficient
- a = Yield Stress $\dot{\varepsilon}$ = Strain rate
- b = Hardening modulus $\dot{\varepsilon}_0$ = Reference strain rate

In this study, the parameters a , b , n , σ_{max} (maximum stress), and the Young modulus are defined as input variables. The stress-strain curve obtained by the experimental test is shown in the following image.



1 ■ Experimental results Aluminum AL 6063 T7
 Engineering Stress Versus Engineering Strain Curve (Experimental Data)

For the simulation results, engineering strains will be obtained by dividing the displacement of node 1 by the reference length (75 mm), and engineering stresses will be obtained by dividing the force in section 1 by its initial surface (10.2 mm²).



Engineering Stress Versus Strain Curve (Simulation Results)

Step 1: Create the Base Input Template in HyperStudy

In this step, you can create the base input template in HyperStudy or use the base input template in the study Directory.

1. Start HyperStudy.
2. From the menu bar, click **Tools > Editor**. The **Editor** opens.
3. In the **File** field, open the TENSILE_TEST_0000.rad file.
4. In the **Find** area, enter /MAT/PLAS_JOHNS/1.
5. Click **▶**. HyperStudy highlights /MAT/PLAS_JOHNS/1.

```

44 #---2---MATERIALS:
45 #---1---|---2---|---3---|---4---|---5---|---6---|---7---|---8---
46 /MAT/PLAS_JOHNS/1
47 Law_2
48 #.....RHO_I
49 .....0027.....0027
50 #.....E.....Nu
51 .....60400.....33
    
```

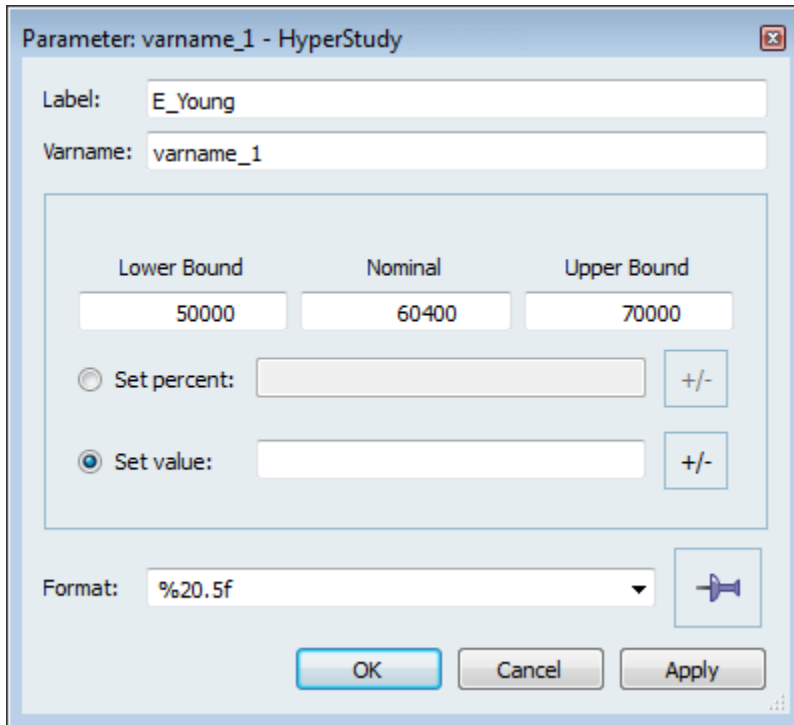
6. Select variable **E** by highlighting the first 20 fields in row 51.

Tip: Quickly highlight 20-character fields by pressing **CTRL** to activate the **Selector** (set to 20 characters) and then clicking the value.

```

46 /MAT/PLAS_JOHNS/1
47 Law_2
48 #.....RHO_I
49 .....0027.....0027
50 #.....E.....Nu
51 .....60400.....33
52 #.....a.....b.....n.....EF
53 .....110.....120.....15.....
54 #.....c.....EPS_0.....ICC.....Fsmooth.....
55 .....0.....0.....0.....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 E_Young.
9. Set the **Lower Bound** to 50000, the **Initial Bound** to 60400, and the **Upper Bound** to 70000.
10. In the **Format** field, enter %20.5f.



11. Click **OK**.
12. Define four more variables using the information provided in the table below.

Variable	Label	Lower Bound	Initial Value	Upper Bound	Format
a	a_PlasticityYieldStress	90	110	120	%20.5f
b	b_HardeningCoeff	100	125	160	%20.5f
n	n_HardeningExpo	0.1	0.2	0.3	%20.5f
sigmax	Sigma_Max	250	280	290	%20.5f

13. Click **Save**.
14. In the **Save Template** dialog, save the file as TENSILE_TEST_0000.tpl.
15. Close the **Editor**.

Step 2: Register the Curve Difference Function

The function `curve_difference` calculates the integral of the absolute value of the difference between two curves over the common domain of the supplied functions. This is a robust function that tends to zero only as the two functions become equal.

Register the `curve_difference` function using:

- Compose

$$\int_a^b |f(x) - g(x)| dx$$

1. Start Compose.
2. From the menu bar, click **File > Open**.
3. In the **Open File** dialog, open the file `Curve_Difference.oml`.
4. Highlight the function **`curve_difference(...)`**.
5. Right-click on the highlighted text and select **Register Function** from the context menu.
6. In the **Register Function** dialog, click **OK**.
7. Close Compose.

Python

1. In your favorite text editor, open the `hst_pref_py.mvw` file.
2. Replace `<path>` with the actual location of the `curve_difference.py` file.


```
*Id("HyperStudy v11.0")
  *BeginDefaults()
    *BeginPlotDefaults()
      *RegisterPythonFunction("curve_difference",
        "<path>/curve_difference.py", 1)
    *EndPlotDefaults()
  *EndDefaults()
```

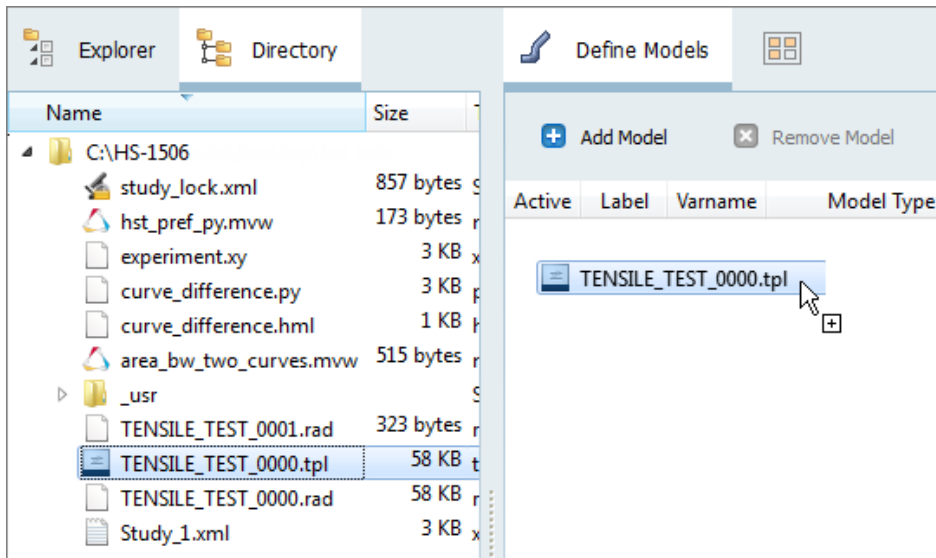
3. Save the changes you made to the preference file.
4. Close the text editor.
5. In HyperStudy, from the menu bar, click **File > Set Preferences File**.
6. In the **HyperStudy - Set Preferences** dialog, navigate to your working directory and open the `hst_pref_py.mvw` file.

Templex

1. From the menu bar, click **File > Use Preferences File**.
2. In the **HyperStudy - Set Preference File** dialog, open the `curve_difference.mvw` file.

Step 3: 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 `TENSILE_TEST_0000.tpl` file into the work area.

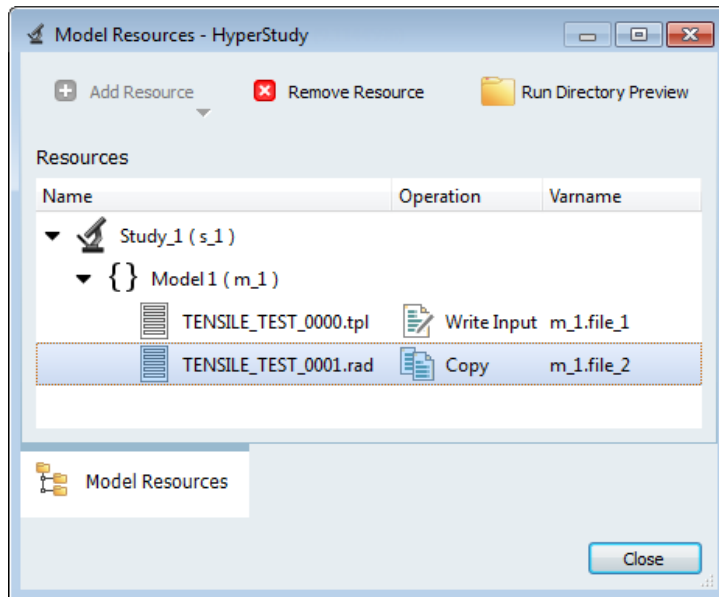


- b. In the **Solver input file** column, enter `TENSILE_TEST_0000.rad`. This is the name of the solver input file HyperStudy writes during any evaluation.
- c. In the **Solver execution script** column, select **RADIOSS (radioss)**.
- d. Verify that the **Solver input arguments** field reads `$file`. This solver input argument runs the Starter and the Engine of RADIOSS for the crash analysis, and also prevents the creation of the `.h3d` result file from animation files.

Note: X is the number of CPUs to use for the simulation.

Active	Label	Varname	Model Type	Resource	Solver input file	Solver execution script	Solver input arguments
<input checked="" type="checkbox"/>	Model1	m_1	{ } Parameterized File	C:/.../HS-1506/TENSILE_TEST_0000.tpl	TENSILE_TEST_0000.rad	RADIOSS (radioss)	{file}

5. Define a model dependency.
 - a. Click **Model Resources**.
 - b. In the **Model Resource** dialog, click **Add Resource** > **Add Input Resource**.
 - c. In the **Select File** dialog, navigate to your working directory and open the TENSILE_TEST_0001.rad file.
 - d. Set **Operation** to **Copy**.
 - e. Click **Close**.



6. Click **Import Variables**. Five input variables are imported from the TENSILE_TEST_0000.tpl resource file.
7. Go to the **Define Input Variables** step.
8. Review the input variable's lower and upper bound ranges.
9. Go to the **Specifications** step.

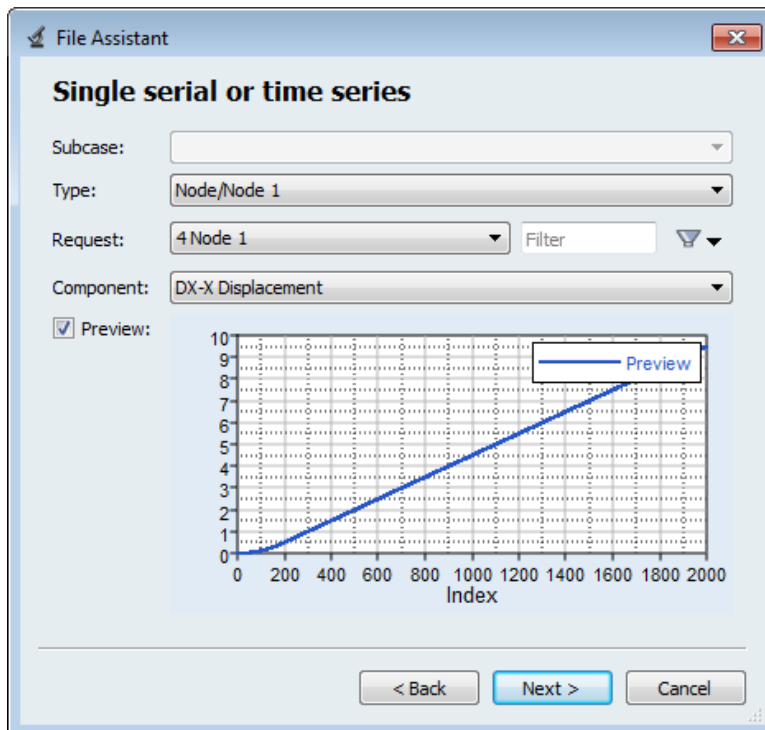
Step 4: Perform the Base Run

1. In the work area, set the **Mode** to **Nominal Run**.
2. Click **Apply**.
3. Go to the **Evaluate** step.
4. Click **Evaluate Tasks**. An approaches/nom_1/ directory is created inside the study directory. The approaches/nom_1/run__00001/m_1 directory contains the TENSILE_TEST01 file, which consist of the time history results of the simulation.
5. Go to the **Define Output Responses** step.

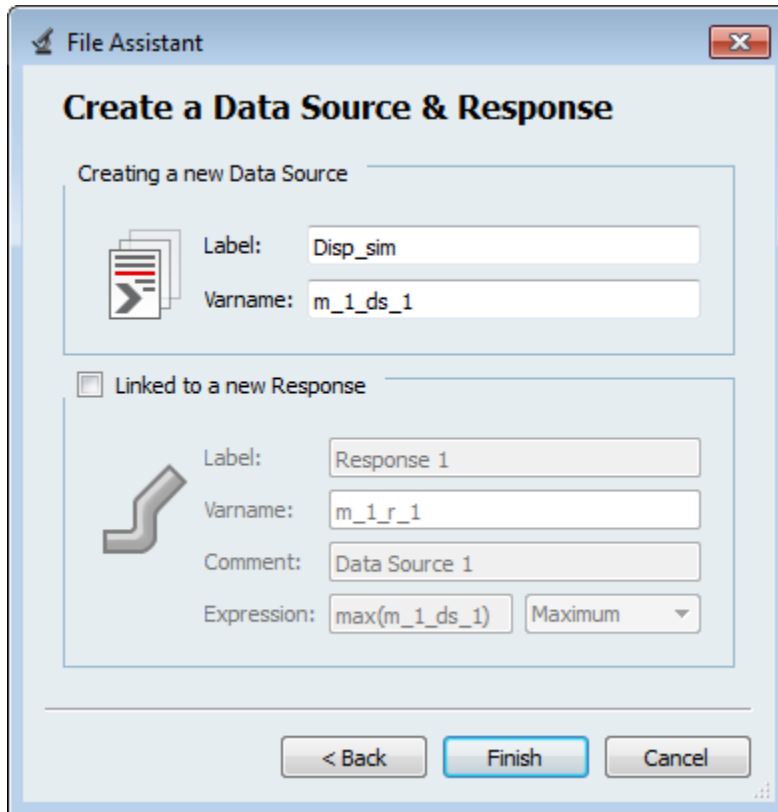
Step 5: Create and Define Output Responses

In order to fit the RADIOSS stress-strain curve to the experimental data, you must compare the two curves. In this step, you will use the curve_difference function.

1. Create the Area Between Two Curves output response.
 - a. Click **Add Output Response**.
 - b. In the work area, **Label** field, enter Area Between Two Curves.
2. Create a data source labeled Disp_sim.
 - a. From the **Directory**, drag-and-drop the TENSILE_TEST01 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 **None/Node 1**.
 - Set **Request** to **4 Node 1**.
 - Set **Component** to **DX-X Displacement**.

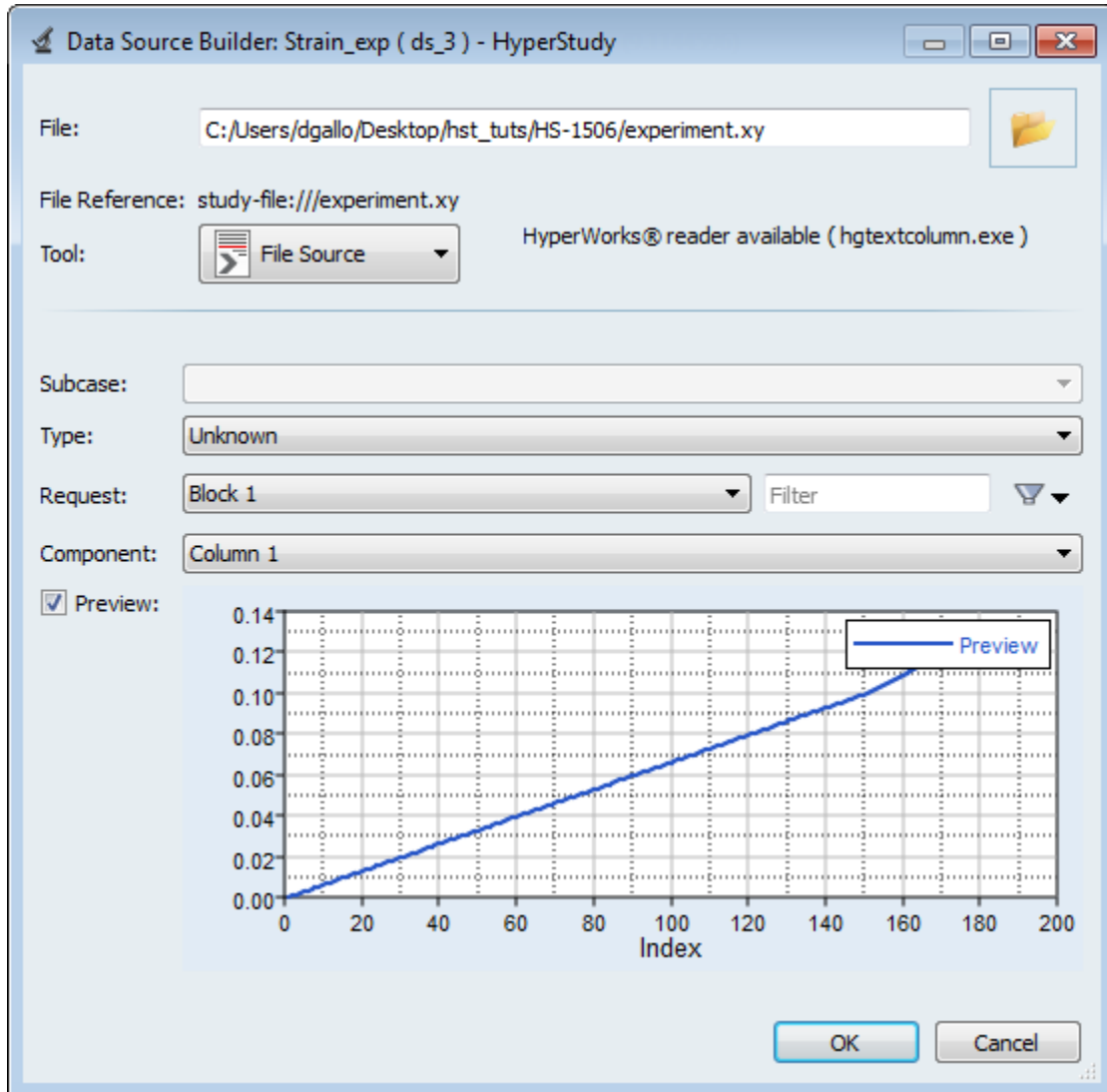


- e. Under **Creating a new Data Source**, enter Disp_sim in the **Label** field.
- f. Clear the **Linked to a new Response** checkbox.
- g. Click **Finish**

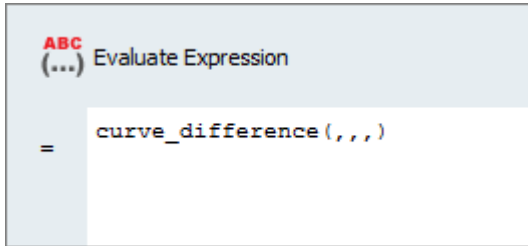


3. Create a second data source labeled Force_sim by repeating step 2, except select the following options:
 - Set **Type** to **Section/SECTION_2**.
 - Set **Request** to **2 section 1**.
 - Set **Component** to **FT-Resultant Tangent Force**.
4. Create a third data source labeled Strain_exp.
 - a. In the **Expression** column of the Area Between Two Curves output response, click **...**.
 - b. In the **Expression Builder**, click the **Data Sources** tab.
 - c. Click **Add Data Source**. A new data source is added to the work area.
 - d. In the **Label** field for the new data source, enter Strain_exp.
 - e. In the **File** field, click **...**.
 - f. In the **Data Source Builder**, define the data source and click **OK**.
 - In the **File** field, navigate to your working directory and open the experiment.xy file.
 - Set **Tool** to **File Source**.
 - Set **Type** to **Unknown**.
 - Set **Request** to **Block1**.

- Set **Component** to **Column 1**.



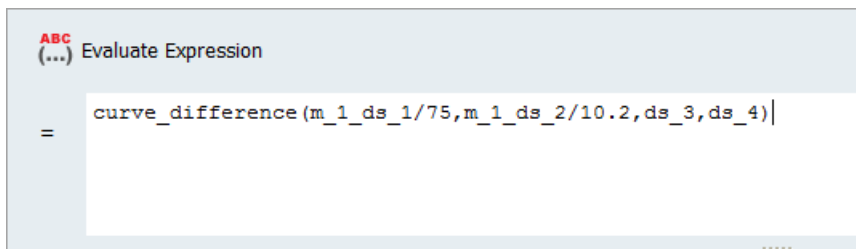
5. Create a fourth file source labeled Stress_Exp by repeating step 4, except select the following options:
 - Set **Type** to **Unknown**.
 - Set **Request** to **Block1**.
 - Set **Component** to **Column 2**.
6. Define the Area Between Two Curves output response.
 - a. In the **Expression Builder**, click the **Functions** tab.
 - b. From the list of available functions, select **curve_difference**.
 - c. Click **Insert Varname**. The function `curve_difference()` appears in the **Evaluate Expression** field.



- d. In the **Evaluate Expression** field, edit the expression so that it reads `curve_difference(m_1_ds_1/75,m_1_ds_2/10.2,ds_3,ds_4)`.

Note: In the expression, you will notice that the four data sources are entered differently. The varname for the first two data sources created using the File Assistant are labeled as `m_1_d_#`, whereas the varname for the last two data sources created in the Expression Builder are labeled as `ds_#`.

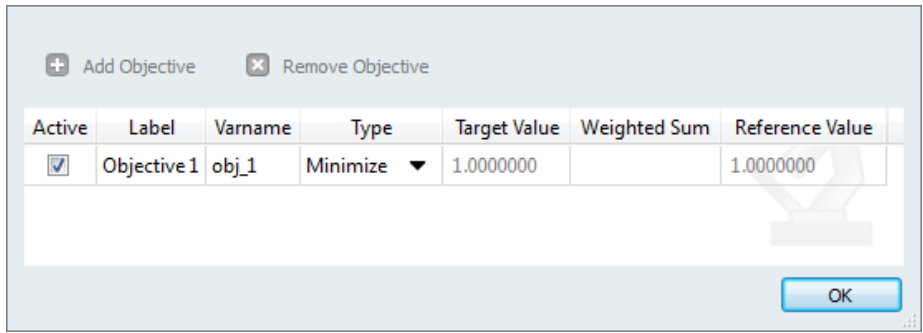
The displacements and forces are read from the simulation, whereas from the experiment you have strains and stresses. In order to convert the displacement and forces to strains and stresses, you need to divide the displacements by the length (75) and forces by the area (10.2).



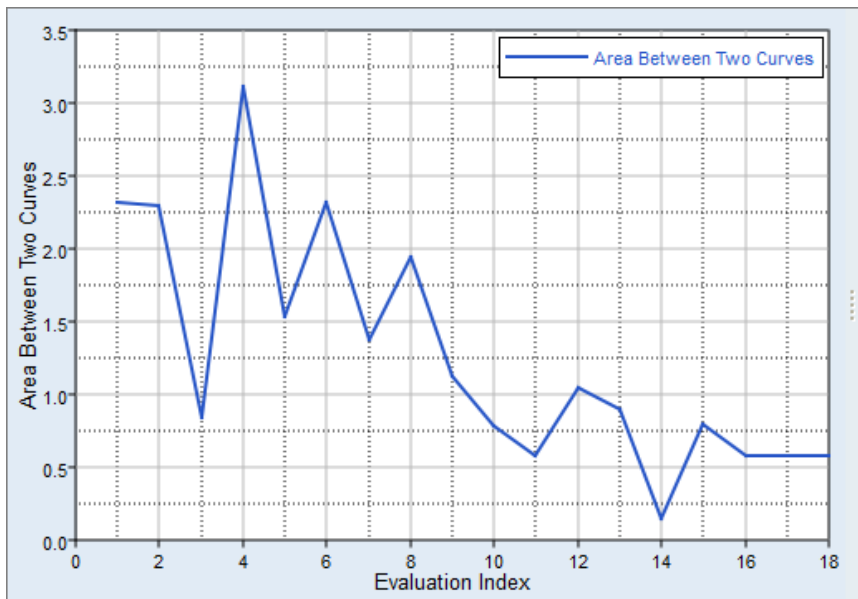
7. Click **OK** to close the **Expression Builder**.
8. Click **Evaluate** to extract the output response value.

Step 6: Run an Optimization Study

1. In the **Explorer**, right-click and select **Add** from the context menu.
2. In the **Add - HyperStudy** dialog, select **Optimization** and click **OK**.
3. Go to the **Select Input Variables** step.
4. Review the input variable's lower and upper bound ranges.
5. Go to the **Select Output Responses** step.
6. Apply an objective on the Area Between Two Curves output response.
 - a. In the **Objectives** column for **Area Between Two Curves**, click **+**.
 - b. In the pop-up window, set **Type** to **Minimize** and click **OK**.



7. Click **Apply**.
8. Go to the **Specifications** step.
9. In the work area, set the **Mode** to **Adaptive Response Surface Method (ARSM)**.
Note: Only the methods that are valid for the problem formulation are enabled.
10. Click **Apply**.
11. Go to the **Evaluate** step.
12. Click **Evaluate Tasks**.
13. **Optional.** Click the **Evaluation Plot** tab to plot the optimization iteration history of the objective.



14. **Optional.** Click the **Iteration History** tab to review the iteration history in a table.

Last modified: v2017.2 (9.1144596)