Altair

HyperWorks

# HS-4200: Material Calibration Using System Identification

The purpose of this tutorial is to introduce a method for characterizing parameters of a RADIOSS material law used for modeling elasto-plastic material. The characterization of a ductile aluminum alloy is studied. A RADIOSS simulation is performed to replicate an experimental tensile test. The parameters of the material law are determined to fit the experimental results.

HS-1506: Material Calibration with a Curve Difference Integral provides an alternative method to setup this problem using a HyperMath or Python function to measure the difference 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.
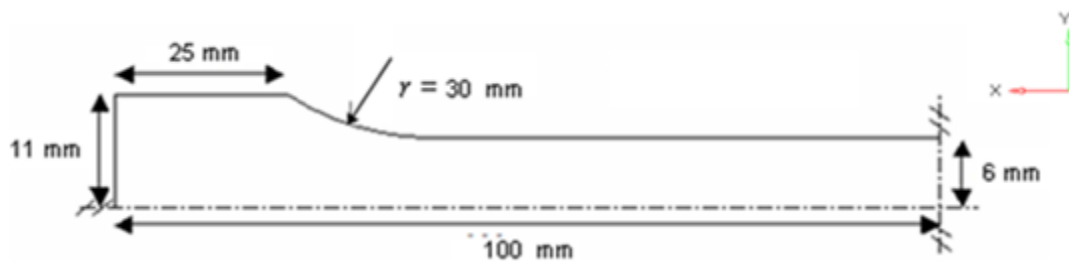
Geometry of the Tensile Specimen (One Quarter of the Specimen is Modeled)


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:

$$\sigma = \underbrace{\left(a + b\varepsilon_p^n\right)}_{\substack{\text{Influence} \\ \text{of} \\ \text{plastic} \\ \text{strain}}} \underbrace{\left(1 + c\ln\tfrac{\varepsilon}{\varepsilon_0}\right)}_{\substack{\text{Influence} \\ \text{of} \\ \text{strain} \\ \text{rate}}} \underbrace{\left(1 - T^{*m}\right)}_{\substack{\text{Influence} \\ \text{of} \\ \text{temperature} \\ \text{change}}}$$

$\sigma =$	Stress level	$n =$	Hardening Exponent
$\varepsilon_p =$	Plastic strain	$c =$	Strain rate coefficient
$a =$	Yield Stress	$\varepsilon =$	Strain rate
$b =$	Hardening modulus	$\varepsilon_0 =$	Reference strain rate

In this study we define, as input variables, the parameters a, b, n, σmax (maximum stress) and the Young modulus. The stress-strain curve obtained by the experimental test is shown in the following image.
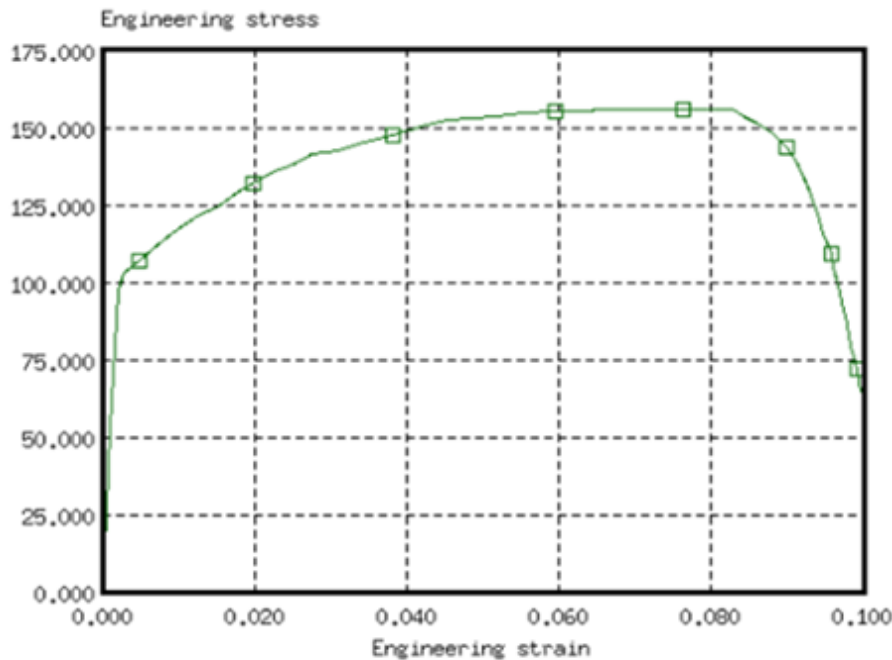
1 □ Experimental results Aluminum AL 6063 T7

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).

Altair
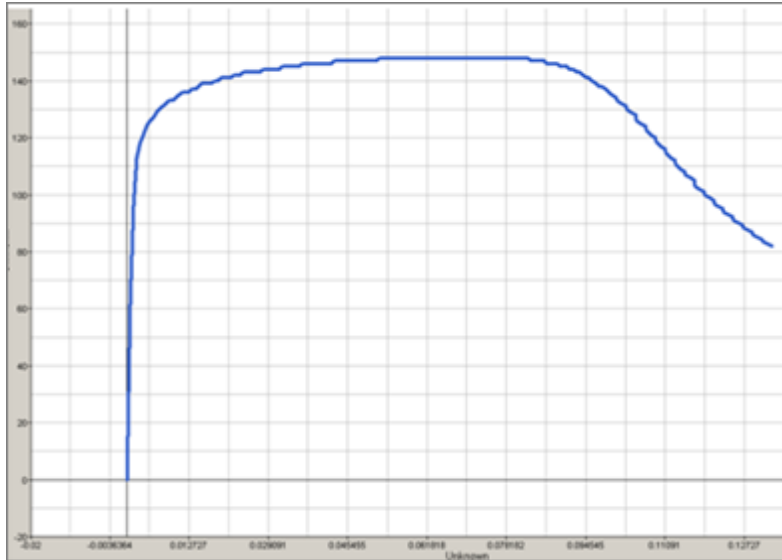
Engineering Stress Versus Strain Curve (Simulation Results)

In this tutorial, you will:

- Create an input template from a RADIOSS data file using the HyperStudy Template Editor
- Set up a study
- Run a system identification optimization study

The sample base input template used in this tutorial can be found in `<hst.zip>/HS-4200/`. Copy the files `TENSILE_TEST_0000.rad`, `TENSILE_TEST_0001.rad`, and `exper.xy` from this directory to your working directory.

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, navigate to your working directory and open the `TENSILE_TEST_0000.rad` file.

    Note: RADIOSS uses fixed fields of 20 characters for properties.

4. In the **Find** area, enter `/MAT/PLAS_JOHNS/1` and click ▶. HyperStudy highlights `/MAT/PLAS_JOHNS/1` in the `TENSILE_TEST_0000.rad` file.

△ Altair

5. Select **E** by starting at the beginning of row 51 and highlighting the first 20 fields.

    Tip: To assist you in selecting 20-character fields, press **CTRL** to activate the **Selector** (set to 20 characters) and then click the value.

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

7. In the **Parameter Varname_1** dialog, **Label** field, enter `E_Young`.

8. Change the **Lower Bound** to `50000`, the **Initial Bound** to `60400`, and the **Upper Bound** to `70000`.

9. In the **Format** field, enter `%20.5f`.
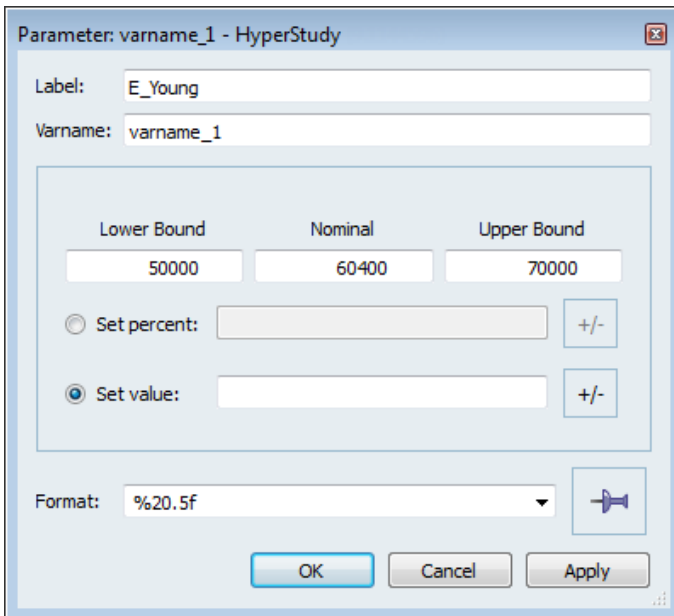
10. Click **OK**.

11. Define four more variables using the information provided in the table below.

    Note: Some of the initial values are different from the values in the original file.

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

12. Click **Save**.

13. In the **Save Template** dialog, navigate to your working directory and save the file as `TENSILE_TEST_0000.tpl`.

14. Close the **Editor** dialog.

## 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 `TENSILE_TEST_0000.tpl` file into the work area.

△ Altair
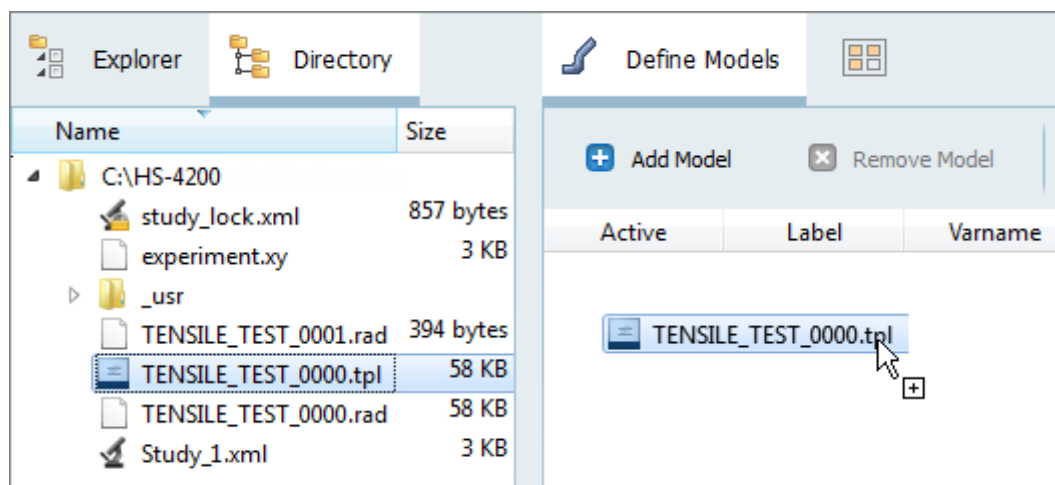
b. In the **Solver input file** column, enter `TENSILE_TEST_0000.rad`. This is the name of the solver input file HyperStudy writes during any evaluation.

c. In the **Solver input file** column, select **RADIOSS (radioss)**.

d. Verify that the **Solver input arguments** field reads `$(file)`. This argument runs the Starter, and the Engine of RADIOSS for the crash analysis. It also prevents the creation of the `.h3d` result file from animation files. 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
1	☑	Model 1	m_1	{} Parameterized File	C:/.../HS-4200/TENSILE_TEST_0000.tpl ⁵	TENSILE_TEST_0000.rad	RADIOSS (radioss)	${file} ❷

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**.
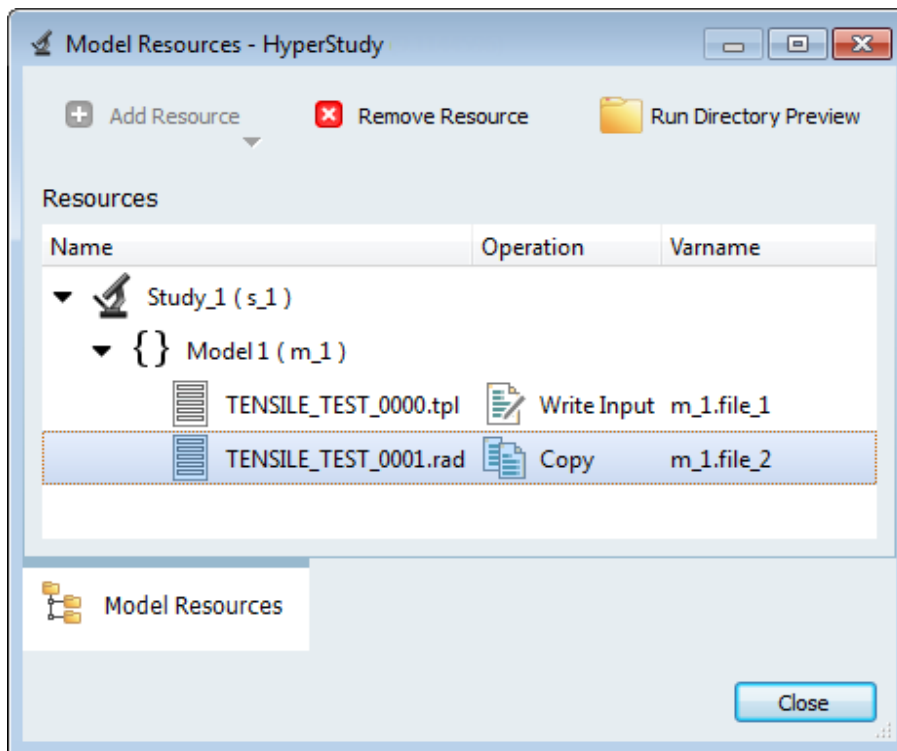
6. Click **Import Variables**. Five input variables are imported from the `TENSILE_TEST_0000.tpl` resource file.

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

8. Check the input variable's lower and upper bound ranges.

9. Go to the **Specifications** step.

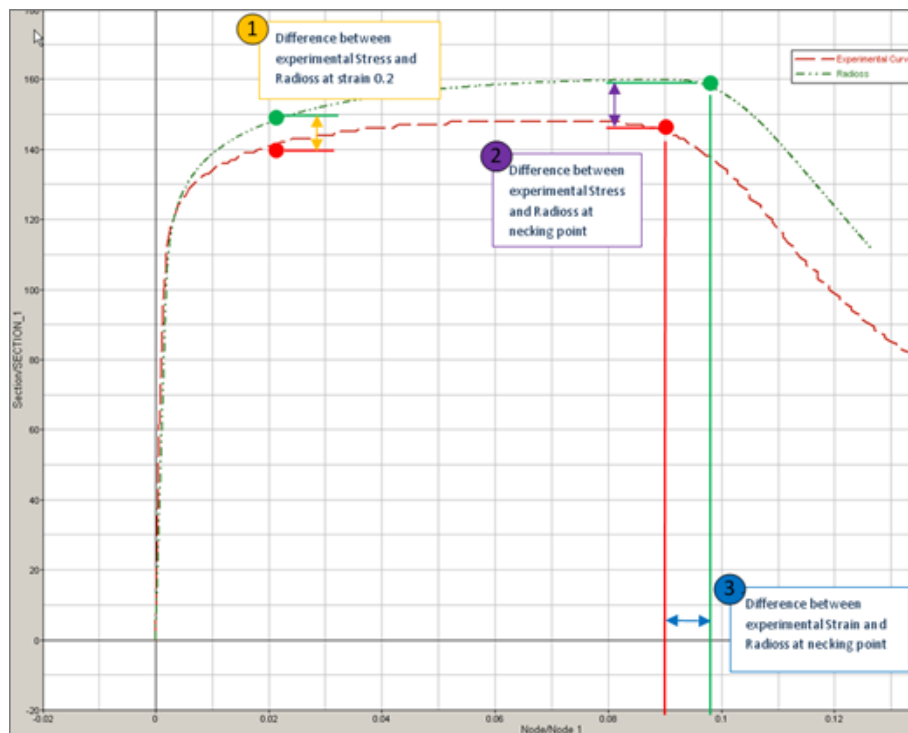△ Altair

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***. An `approaches/nom_1/` directory is created inside the study directory. The `approaches/nom_1/run__00001/m_1` directory contains the `TENSILE_TESTT01` file, which stores the time history results of the simulation.

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

Step 4: Create and Define Output Responses

In order to fit the RADIOSS stress-strain curve to the experimental data we need to find a way to compare the two curves. In this tutorial we will only focus on three specific points on the curve.

Since damage is not modeled with this law, the comparison is not needed after the necking point.

- Difference between experimental stress and RADIOSS at Strain equal 0.02 (1)

- Difference between experimental strain and RADIOSS at Necking point (2)

- Difference between experimental stress and RADIOSS at Necking point (3)

△ Altair

1. 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*.
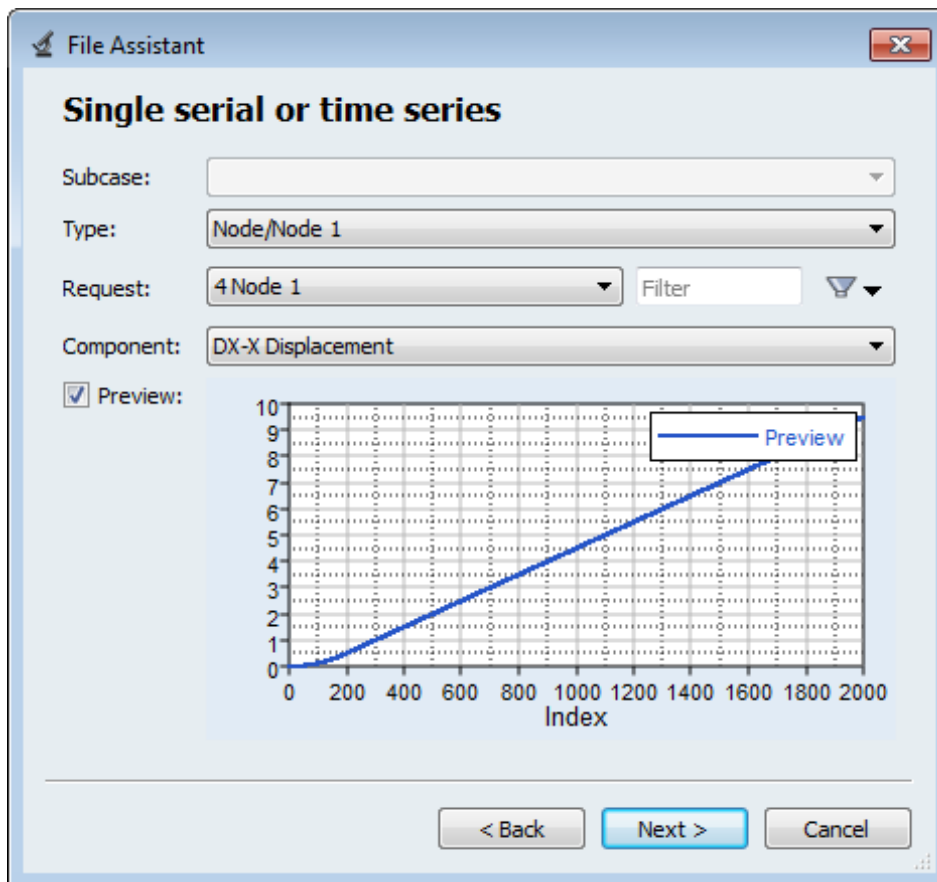
    e. Under **Creating a new Data Source**, enter Disp_sim in the **Label** field.

   f. Clear the **Linked to a new Response checkbox**.

    g. Click *Finish*
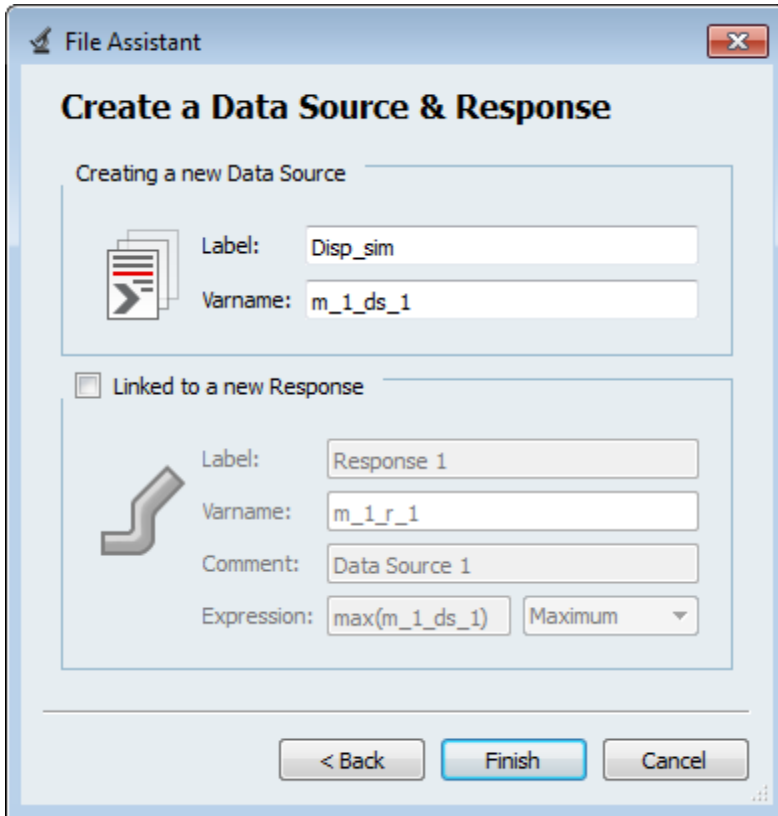
2. Create a second data source labeled `Force_sim` by repeating step 1, except define the following options:

    - Set **Type** to *Section/SECTION_2*.

    - Set **Request** to *2 section 1*.

    - Set **Component** to *FT-Resultant Tangent Force*.

3. Create three output responses.

    a. Click **Add Output Response** three times.

    b. In the work area, **Label** column, change the labels for the three output responses to Radioss_Strain_0_2, Radioss_Stress_Necking, and Radioss_Strain_Necking.

|	Active	Label	Varname	Expression	Value
1	✓	Radioss_Strain_0_2	r_1	•••	Not Extracted
2	✓	Radioss_Stress_Necking	r_2	•••	Not Extracted
3	✓	Radioss_Strain_Necking	r_3	•••	Not Extracted

4. Define the Radioss_Strain_0_2 output response.

    a. In the **Expression** column of the output response **Radioss_Strain_0_2**, click •••.

    b. Click the **Function** tab.

    c. From the list of functions, select **lininterp**.

d. Click *Insert Varname*. The function `lininterp(,,)` appears in the **Evaluate Expression** field.

e. In the **Evaluate Expression** field, enter `(m_1_ds_1/75,m_1_ds_2/10.2,0.02)` in the **lininterp** function.

   This expression computes the Stress with respect to the Strain, at Strain equals `0.02`.
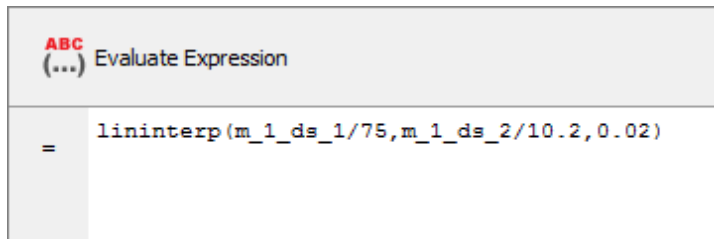
f. Click *Evaluate Expression*.

g. Click *OK* to close the **Expression Builder**.

5. Define the Radioss_Stress_Necking output response.

a. In the **Expression** column of the output response **Radioss_Stress_Necking**, click **...**.

b. In the **Expression Builder**, click the *Data Sources* tab.

c. From the list of data sources, select *Force_sim*.

d. From the **Insert Varname** drop-down, select *Maximum*.
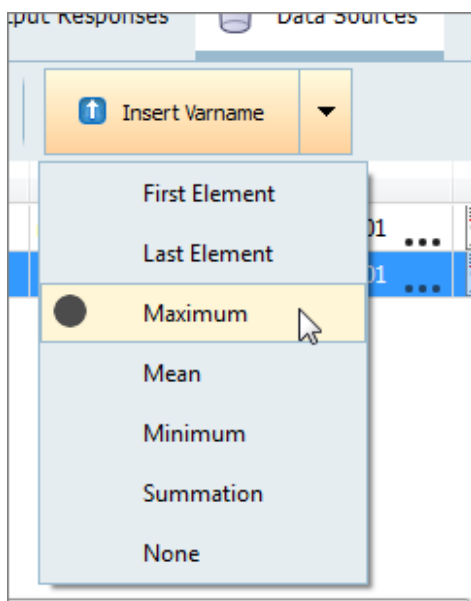
e. Click *Insert Varname*. The function `max(m_1_ds_2)` appears in the *Evaluate Expression* field.

f. In the **Evaluate Expression** field, edit the function so that it reads `max(m_1_ds_2)/10.2`.

This is the maximum of the force (`m_1_ds_2`), which is trimmed between the min strain and the strain at the max value of Force, divided by `10.2` (surface area) to obtain the stress.
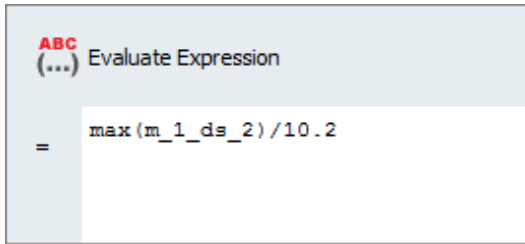

```
ABC
(...) Evaluate Expression

=    max(m_1_ds_2)/10.2
```

> **Note:** In some similar curve fitting problems it may be required to trim the data after the ultimate load, the expression to do this is:
> `max(m_1_ds_2[subrange(m_1_ds_1,min(m_1_ds_1),m_1_ds_1[indexofma x(m_1_ds_2)])])/10.2`.

   g. Click **Evaluate Expression.**

   h. Click **OK** to close the **Expression Builder**.

6. Define the Radioss_Strain_Necking output response.

   a. In the **Expression** column of the output response **Radioss_Strain_Necking**, click **...**.

   b. In the **Evaluate Expression** field, enter `m_1_ds_1[indexofmax(m_1_ds_2)]/75`.

   This is the displacement (`m_1_ds_1`) at the max value of the force value, divided by 75 (reference length) to obtain strain.
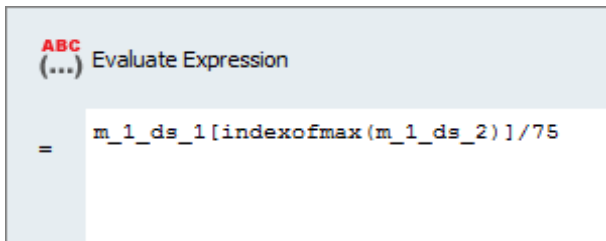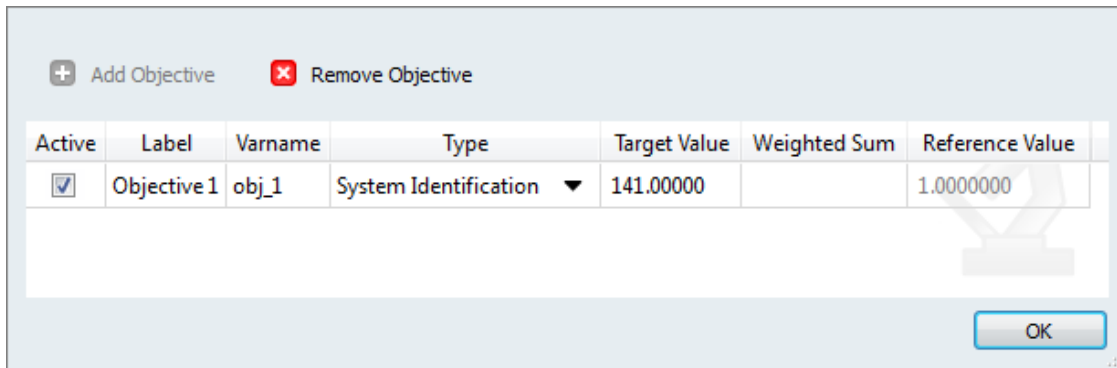
```
ABC
(...) Evaluate Expression

=    m_1_ds_1[indexofmax(m_1_ds_2)]/75
```

> **Note:** In some similar curve fitting problems it may be required to trim the data after the ultimate load, the expression to do this is:
> `m_1_ds_1[maxindex(subrange(m_1_ds_1,min(m_1_ds_1),m_1_ds_1[indexof max(m_1_ds_2)]))]/75`.

   c. Click **Evaluate Expression**.

   d. Click **OK** to close the **Expression Builder**.

Step 5: 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**.

△ Altair

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 Radioss_Strain_0_2 output response.

    a. In the **Objectives** column for **Radioss_Strain_0_2**, click ⊕.

    b. In the pop-up window, define the following and click **OK**.

        •   Set **Type** to *System Identification*.

        •   For **Target Value**, enter `141.00000`.

| Active	Label	Varname	Type	Target Value	Weighted Sum	Reference Value
☑	Objective 1	obj_1	System Identification ▼	141.00000		1.0000000

7. Apply an objective on the Radioss_Stress_Necking and Radioss_Strain_Necking output responses.

    a. Radioss_Stress_Necking

        •   Type = System Identification

        •   Target Value = 148.00000

    b. Radioss_Strain_Necking

        •   Type = System Identification

        •   Target Value = 0.0800000

8. Click **Apply**.

9. Go to the **Specifications** step.

10. 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.

11. Click **Apply**.

12. Go to the **Evaluate** step.

13. Click **Evaluate Tasks** to launch the Optimization.

14. Click the **Iteration Plot** tab.

    Using the **Channel** selector, select the three objectives from and Objective Function. Activate multi-plot to see the each channel in its own plot.

    The first three selections are the actual values used in the system identification optimization problem. Observe their objective history to see that their values indeed

approach their respective target values. The final plot is the scalar objective which is used in the system identification problem; a normalized sum of the squares difference between the actual and target objective values. Note that the value of this combined function has been reduced through the optimization.
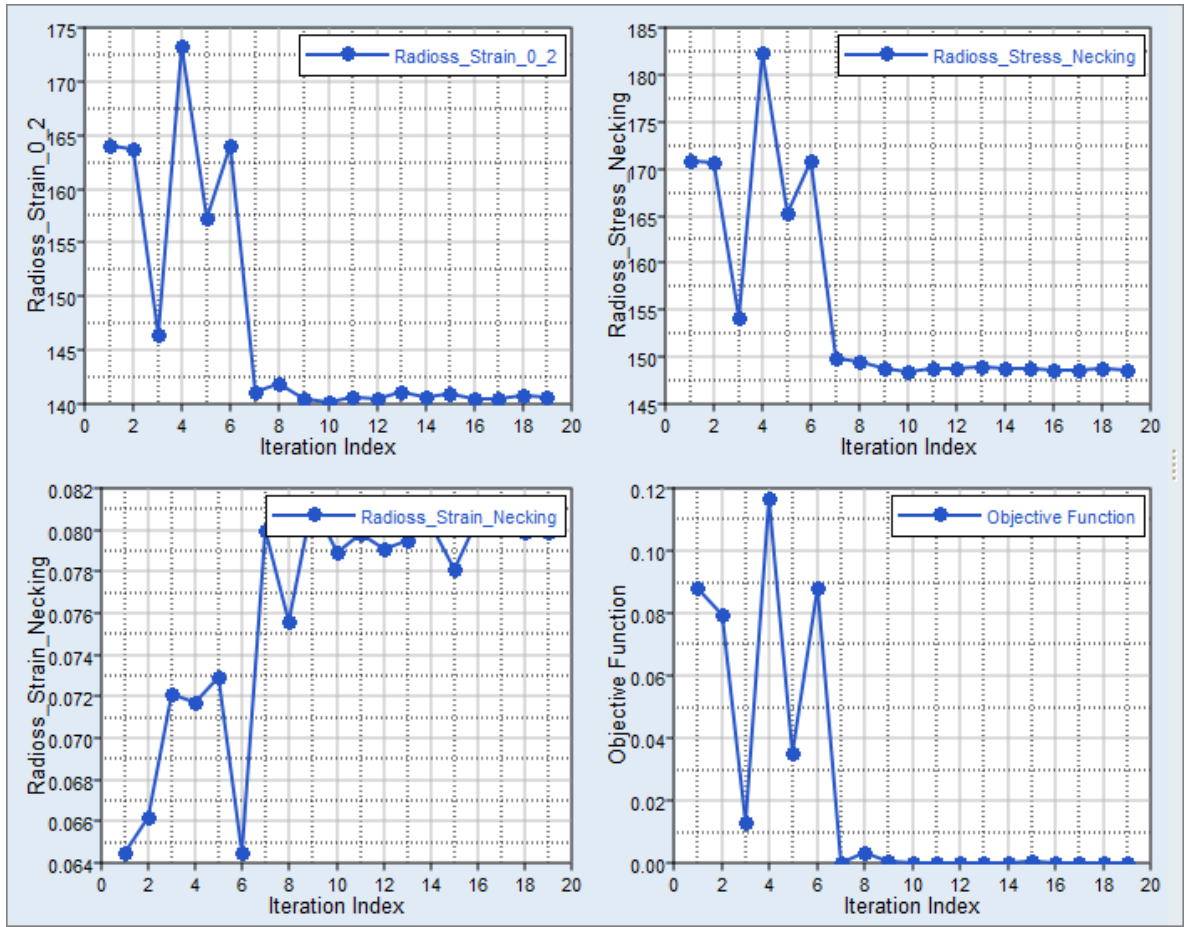
Last modified: v2017.2 (9.1144596)

Altair