Altair

HyperWorks

Altair HyperStudy 2019 Tutorials

HS-1560: Study Setup Using LS-DYNA Model Parameters in HyperMesh

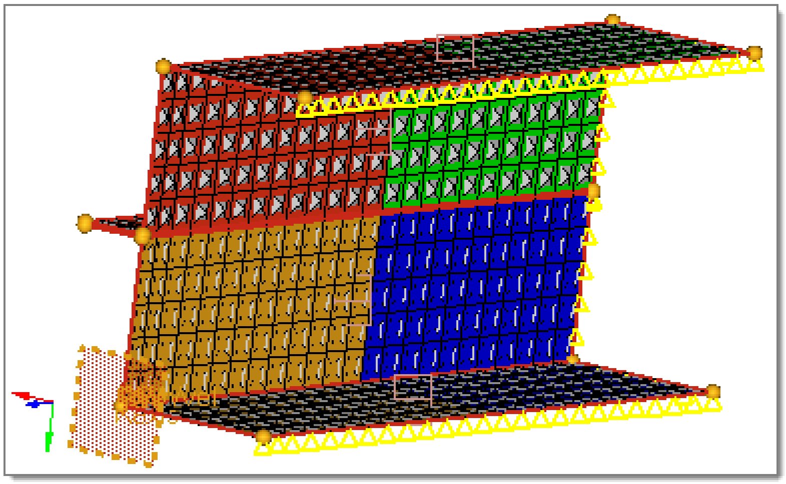
HS-1560: Study Setup Using LS-DYNA Model Parameters in HyperMesh

This exercise outlines the procedure for setting up a study for a LS-DYNA model in HyperMesh. This model has shape variables created using HyperMorph. In addition to the shape variables a HyperMesh parameter is created for material property, E. These variables are then imported to HyperStudy as input variables. For this study, you will start HyperStudy from within HyperMesh.

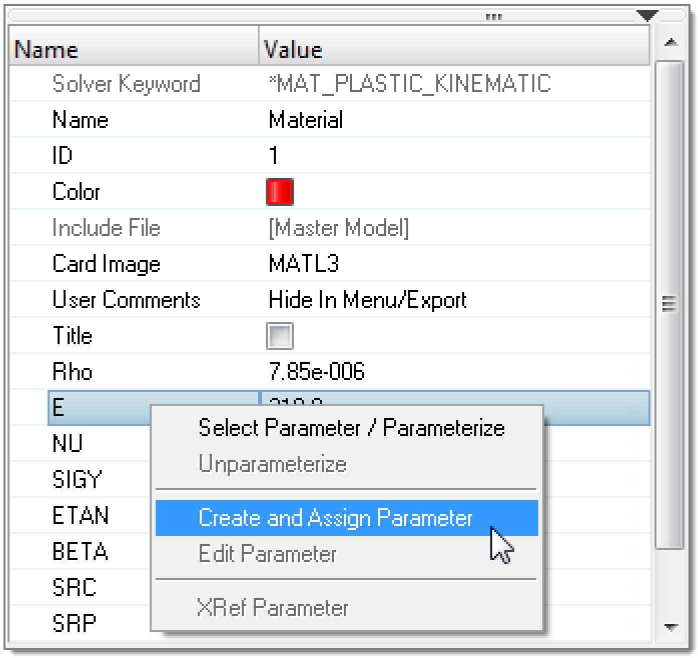
The files used in this tutorial can be found in `<hst.zip>/HS-1560/`. Copy the tutorial files from this directory to your working directory.

Step 1: Importing a HyperMesh Parameter into HyperStudy

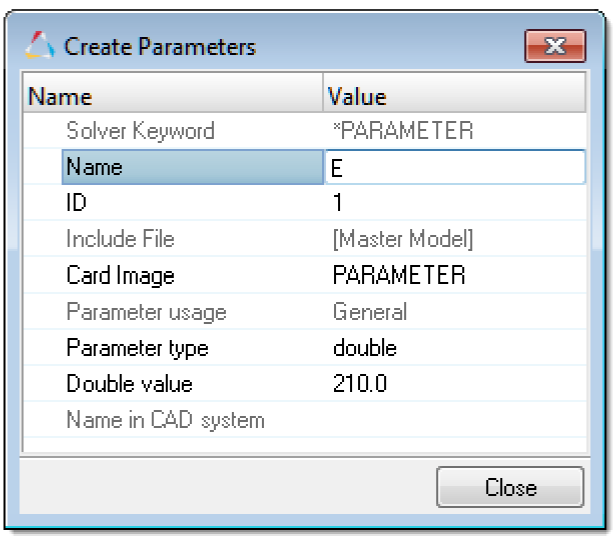
1. Start HyperMesh Desktop.
2. In the **User Profiles** dialog, set the user profile to ***LS-DYNA***.
3. From the menu bar, click ***File*** > ***Open*** > ***Model***.
4. In the **Open Model** dialog, open the `boxbeam_morphed_noDV.hm` file. A model appears in the graphics area.



5. In the **Model** browser, **Material** folder, click ***Material***. The **Entity Editor** opens and displays the material's corresponding data.
6. Right-click on ***E*** and select ***Create and Assign Parameter*** from the context menu.



7. In the **Create Parameters** dialog, change the **Name** to `E` and then click ***Close***.

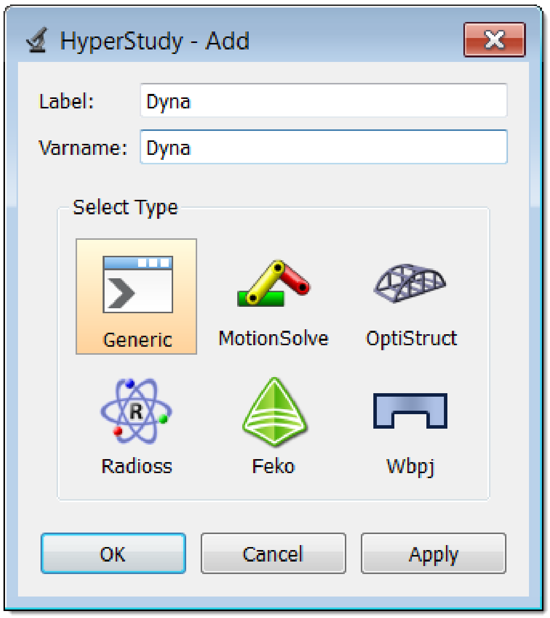


Step 2: Register LD-DYNA as a Solver

In order to use LS-DYNA as a solver, you need to register it in the preference file for HyperStudy.

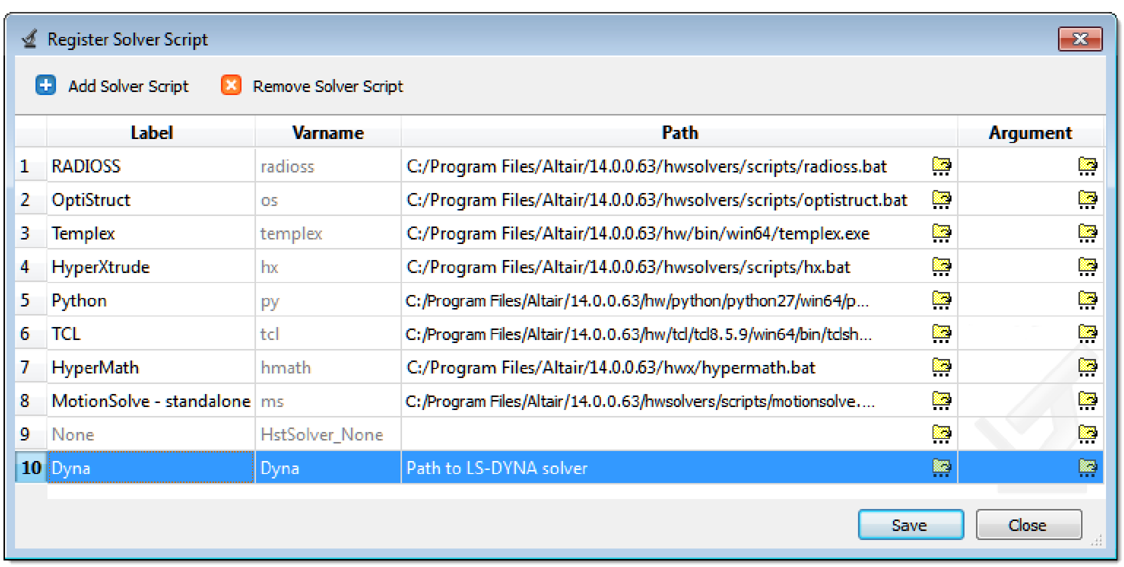
1. Start HyperStudy.
2. From the menu bar, click ***Edit*** > ***Register Solver Script***.

3. In the **Register Solver Script** dialog, click ***Add Solver Script***.
4. In the **Add - HyperStudy** dialog, enter `Dyna` in the **Label** and **Varname** fields.
5. From the list of solver script types, select ***Generic***.
6. Click ***OK***.



7. In the **Path** column of the script **Dyna**, click .
8. In the **Open** dialog, navigate to the local installation of the LS-DYNA solver, and then have it point to the LS-DYNA solver executable without any spaces in the file path.

   Note: If LS-DYNA is not installed locally, write a solver script to call it properly. For more information on this process, refer to Solver Script Files.



9. Click ***Save***.
10. In the **Save Preferences** dialog, navigate to your working directory.

    Note: On UNIX, the preference file can also be saved in your home directory or in the working directory from which you launched HyperStudy.

11. In the **File name** field, enter a label for the new user preference file (example: `userprefs.mvw`).
12. Click ***Save***.

    Note: Do not overwrite the system preferences file, which is located in <install_directory>/hw by default.

13. Click ***Close***.

    When you start a new HyperStudy session, load your preference file by clicking ***File*** > ***Set Preference File*** from the menu bar. The default preferences file in the installation directory will be read, followed by the preference file that you specify. This ensures that all solvers, readers, and import templates are available.

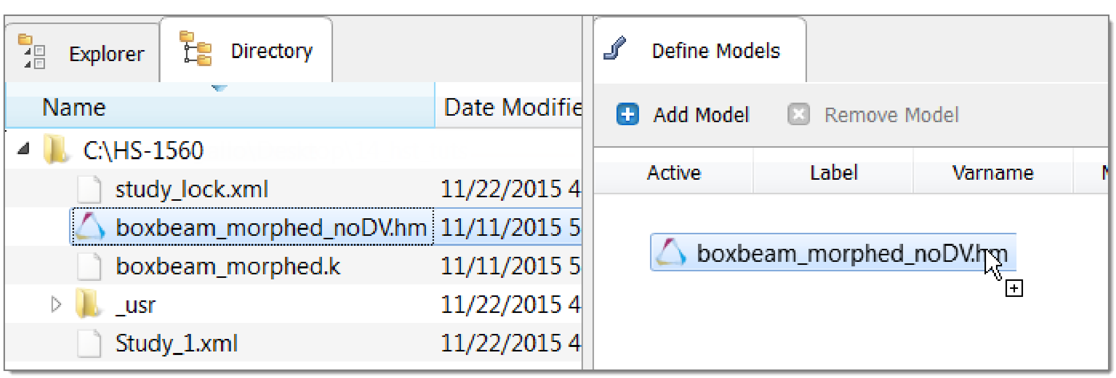
    Append the current user preference file by clicking ***Append***, or exit solver registration by clicking ***Close***. In the last case, the solver will only be registered for the current study.

Step 3: Perform the Study Setup

During this step, you will import the input variables that you created in Step 1: Importing a HyperMesh Parameter into HyperStudy.

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 HyperMesh model.

    a. From the **Directory**, drag-and-drop the `boxbeam_morphed_noDV.hm` file into the work area.

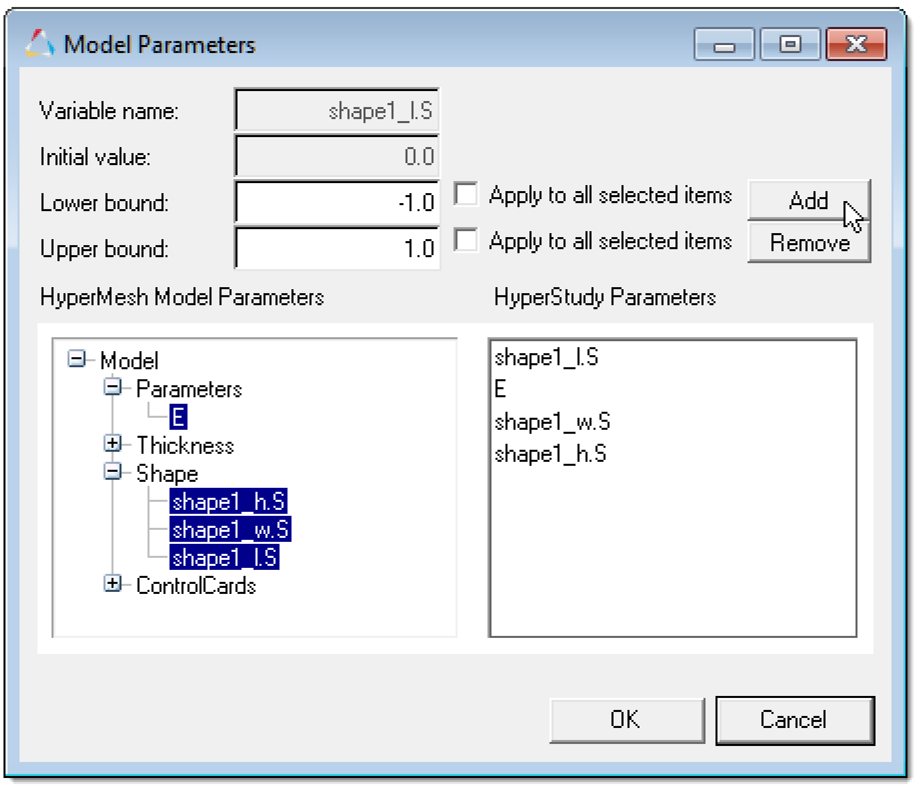
    

    b. In the **Solver input file** column, enter `boxbeam_morphed.kas`. This is the name of the solver input file HyperStudy writes during any evaluation.

c. In the **Solver execution script** column, select ***Dyna (Dyna)***.

d. In the **Solver input arguments** column, enter `i=` before `$file`.

5. Click ***Import Variables***.
6. In the **Model Parameters** dialog, select parameters to import into HyperStudy.
   a. Select the following parameters: ***E***, ***shape1_h.S***, ***shape1_w.S***, and ***shape1_l.S***.
   b. Click ***Add***.
   c. Click ***OK***



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.

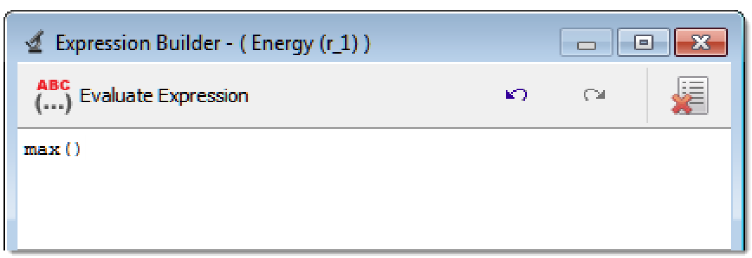
Step 4: Perform the Nominal Run

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

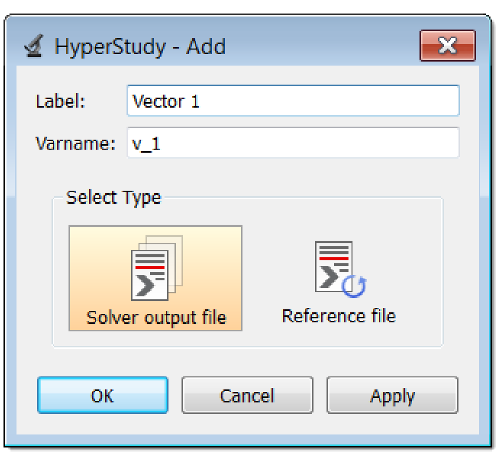
3. Go to the **Evaluate** step.
4. Click ***Evaluate Tasks***. The run is executed using LS-DYNA, and an `approach/nom_1/` directory is created inside the study directory. The `approaches/nom_1/run__00001/m_1` directory contains the `glstat` (for the strain energy), `binout0000` (for the reaction force), and `d3hsp` (for the structural mass) files, which are the result of the nominal run.
5. Go to the **Define Output Responses** step.

Step 5: Create and Define Output Responses

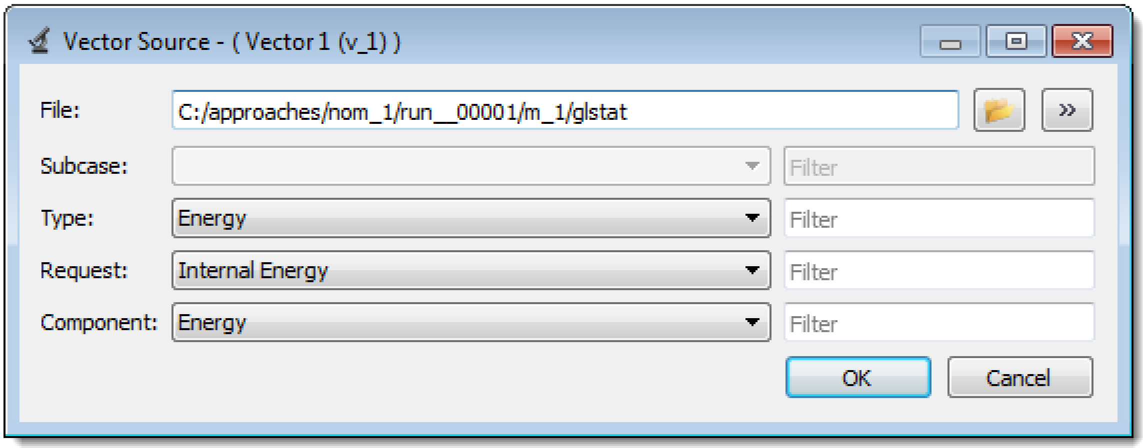
1. Click ***Add Output Response***.
2. In the **Add - HyperStudy** dialog, add three output responses and label them Energy, Force, and Mass.
3. In the **Expression** column of the output response **Energy**, click **...**.
4. In the **Expression Builder**, click the ***Functions*** tab.
5. From the list of available functions, select ***max***.
6. Click ***Insert Varname***. The function `max()` appears in the **Evaluate Expression** field.



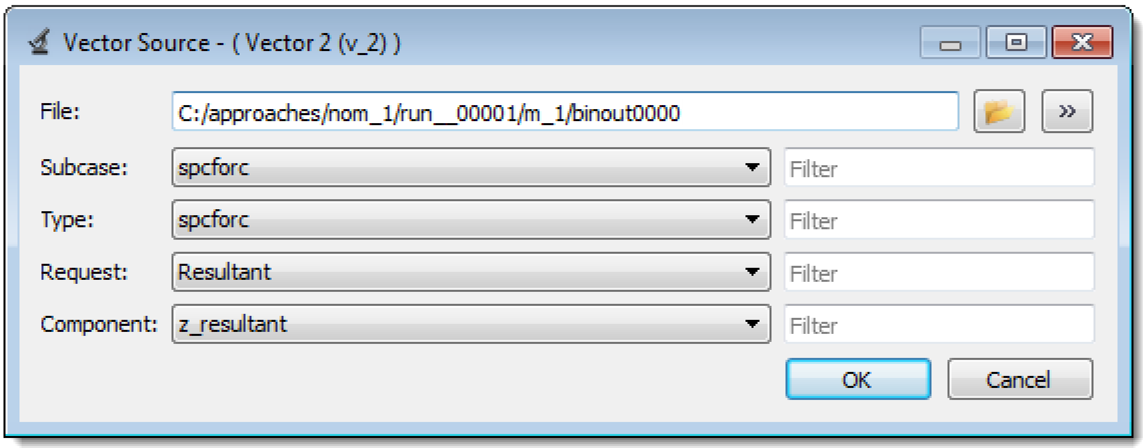
7. Click the ***File Sources*** tab.
8. Click ***Add File Source***.
9. In the **Add - HyperStudy** dialog, add one **Solver output file**.



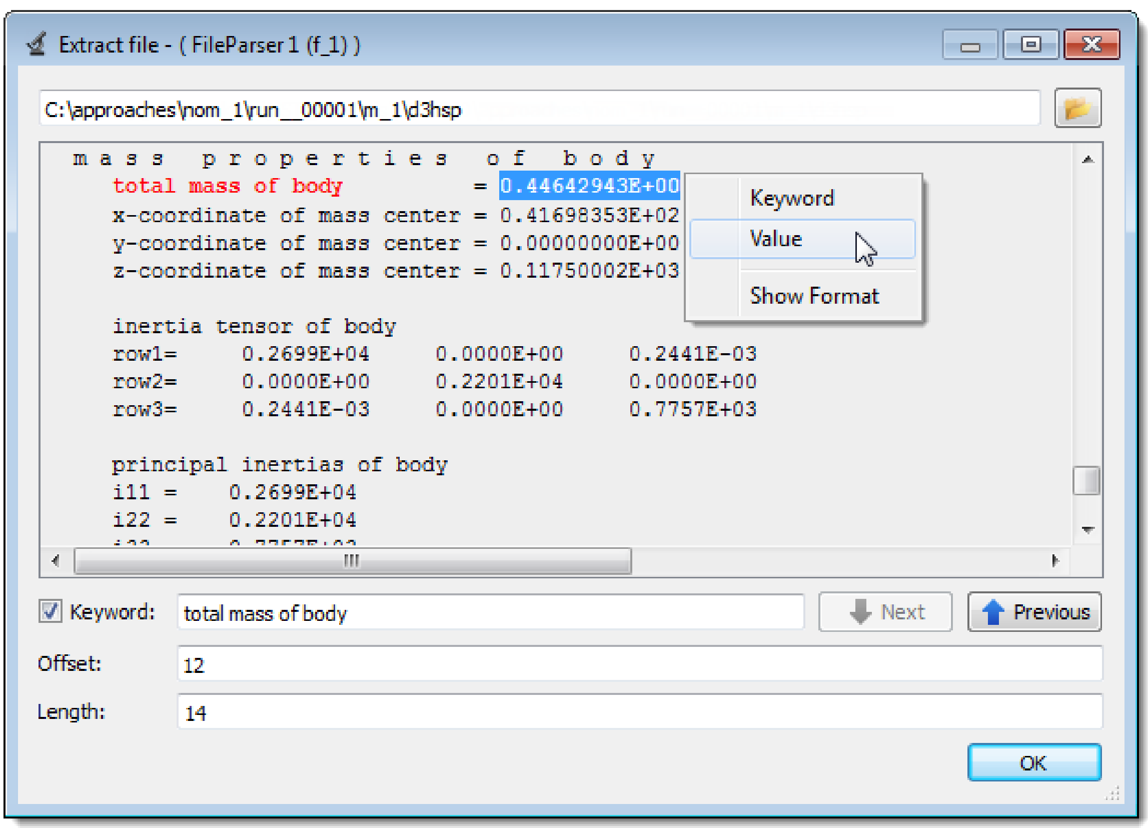
10. In the **File** column of **Vector 1**, click ***.
11. In the **Vector Source** dialog, navigate to the `approaches/nom_1/run__00001/m_1` directory and open the `glstat` file.
12. From the **Type**, **Request**, and **Component** fields, select the options indicated in the image below.



13. Click ***OK***.
14. Click ***Insert Varname***. The expression `max(m_1_ds_1[0])` appears in the **Evaluate Expression** field.
15. Remove `[0]` from the expression, so that it reads `max(m_1_ds_1)`. This expression produces the value of the energy value extracted from the nominal run.
16. Click ***OK***.
17. Repeat steps 3 through 10 for the **Force** output response.
18. In the **Vector Source** dialog, navigate to the `approaches/nom_1/run__00001/m_1` directory and open the `binout0000` file.
19. From the **Subcase**, **Type**, **Request**, and **Component** fields, select the options indicated in the image below.



20. Click ***OK***.
21. Click ***Insert Varname***. The expression `max(v_2[0])` appears in the **Evaluate Expression** field.
22. Remove `[0]` from the expression, so that it reads `max(v_2)`. This expression produces the value of the Force value extracted from the nominal run.
23. Click ***OK***.
24. In the **Expression** column of the output response **Mass**, click **...**.
25. In the **Expression Builder**, click the ***ASCII Extracts*** tab.
26. Click ***Add Extract Source***.
27. In the **Add - HyperStudy** dialog, add one extract source.
28. In the **File Path** column of **FileParser1**, click **...**.
29. In the **Extract File** dialog, navigate to the `approaches/nom_1/run__00001/m_1` directory and open the `d3hsp` file.
30. To search for certain keywords within the `d3hsp` file, select the ***Keyword*** checkbox.
31. In the **Keyword** field, enter `total mass of body`.
32. Click ***Next***. HyperStudy locates **total mass of body** in the file.
33. Highlight the value for total mass of body.
34. Right-click on the highlighted fields, and select ***Value*** from the context menu.



35. Click ***OK***.
36. Click ***Insert Varname***. The expression `f_1[0]` appears in the **Evaluate Expression** field. This expression produces the value of the Mass value extracted from the nominal run.
37. Click ***OK***.
38. Click ***Evaluate Expressions*** to extract the output response values.
39. Go to the **Post processing** step.
40. Click the ***Scatter 2D*** tab to view the values of the output responses, and to check that your output responses have similar values.

    Use the **Channel** selector to set the **X Axis** to **Energy** and the **Y Axis** to **Force**.

    Note: The version and architecture of LS-DYNA that you are using may produce slightly different output response values.