

altairhyperworks.com

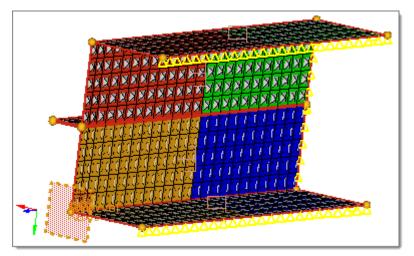
# HS-1560: Study Setup Using LS-DYNA Model Parameters in HyperMesh

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 <code>boxbeam\_morphed\_noDV.hm</code> 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.



		•
Name	Value	-
Solver Key	word *MAT_PLASTIC_KINEMATIC	
Name	Material	
ID	1	
Color		
Include Fil	[Master Model]	
Card Imag	MATL3	
User Comr	ents Hide In Menu/Export	=
Title		
Rho	7.85e-006	
E	210.0	
NU	Select Parameter / Parameterize	
SIGY	Unparameterize	
ETAN	Create and Assign Parameter	
BETA	Edit Parameter	
SRC		
SRP	XRef Parameter	-

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

🛆 Create Parameters	×	
Name	Value	
Solver Keyword	*PARAMETER	
Name	E	
ID	1	
Include File	[Master Model]	
Card Image	PARAMETER	
Parameter usage	General	
Parameter type	double	
Double value	210.0	
Name in CAD system		
7	Close	

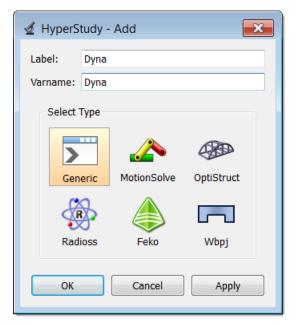
### Step 2: Register LD-DYNA as a Solver

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 *P*.
- 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 <u>Solver Script Files</u>.

	🗄 Add Solver Script 🛛 🛛	Remove Solver Scrip	ot			
	Label	Varname	Path		Argumen	t
1	RADIOSS	radioss	C:/Program Files/Altair/14.0.0.63/hwsolvers/scripts/radioss.bat	<u>-</u>		
2	OptiStruct	os	C:/Program Files/Altair/14.0.0.63/hwsolvers/scripts/optistruct.bat	<u>6</u>		6
3	Templex	templex	C:/Program Files/Altair/14.0.0.63/hw/bin/win64/templex.exe	<u>:</u>		6
4	HyperXtrude	hx	C:/Program Files/Altair/14.0.0.63/hwsolvers/scripts/hx.bat	<u>:</u>		6
5	Python	ру	C:/Program Files/Altair/14.0.0.63/hw/python/python27/win64/p	<u>6</u>		6
6	TCL	tcl	C:/Program Files/Altair/14.0.0.63/hw/tcl/tcl8.5.9/win64/bin/tclsh	<u>:</u>		6
7	HyperMath	hmath	C:/Program Files/Altair/14.0.0.63/hwx/hypermath.bat	<u>6</u>		
8	MotionSolve - standalone	ms	C:/Program Files/Altair/14.0.0.63/hwsolvers/scripts/motionsolve	<u>-</u>		
9	None	HstSolver_None		<u>-</u>		
10	Dyna	Dyna	Path to LS-DYNA solver			



- 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 *p* 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 <code>boxbeam\_morphed\_noDV.hm</code> file into the work area.

Explorer Directory		\$	Define Mode	els	
Name	Date Modifie	•	Add Model	Remove M	Iodel
4 📗 C:\HS-1560					
study_lock.xml	11/22/2015 4		Active	Label	Varname N
Southeam_morphed_noDV.hm	11/11/2015 5				
boxbeam_morphed.k	11/11/2015 5		🛆 boxbe	am_morphed_	noDV.htm
▷ 📙 _usr	11/22/2015 4	+			Ē
Study_1.xml	11/22/2015 4				

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

🛆 Model Paramete	ers	_ • •
Variable name: Initial value: Lower bound: Upper bound: HyperMesh Model P B-Model B-Parameter B-Thickness B-Shape shape shape B-ControlCar	s 1 <u>h.S</u> 1 <u>w.S</u> 1_LS	<ul> <li>Apply to all selected items</li> <li>Apply to all selected items</li> <li>HyperStudy Parameters</li> <li>Shape1_I.S</li> <li>F</li> <li>shape1_w.S</li> <li>shape1_h.S</li> </ul>
		OK Cancel

- 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**

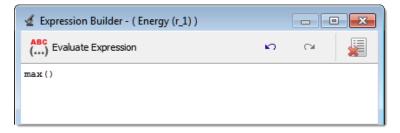
- 1. In the work area, set the **Mode** to **Nominal Run**.
- 2. Click *Apply*.



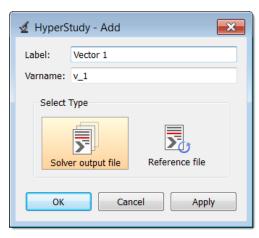
- 3. Go to the **Evaluate** step.
- 4. Click Evaluate Tasks. The run is executed using LS-DYNA, and an approach/nom\_1/ directory is created inside the study directory. The approaches/nom\_1/run\_00001/m\_1 directory contains the glstat (for the strain energy), binout0000 (for the reaction force), and 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.

🚽 Vector Sou	urce - ( Vector 1 (v_1) )	
File:	C:/approaches/nom_1/run00001/m_1/glstat	📂 »
Subcase:	<b></b>	Filter
Type:	Energy 👻	Filter
Request:	Internal Energy 🗸	Filter
Component:	Energy 👻	Filter
		OK Cancel

- 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.

🚽 Vector So	urce - ( Vector 2 (v_2) )	
File:	C:/approaches/nom_1/run00001/m_1/binout0000	) 📂 »
Subcase:	spcforc 💌	Filter
Type:	spcforc 💌	Filter
Request:	Resultant 👻	Filter
Component:	z_resultant	Filter
		OK Cancel



- 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.

🔮 Extract file -	· ( FileParser 1 (f_1) )	
C:\approaches	s\nom_1\run00001\m_1\d3hsp	<b>1</b>
x-coc y-coc z-coc inert row1= row2=	properties of body mass of body = 0.44642943E+00 ordinate of mass center = 0.41698353E+02 ordinate of mass center = 0.00000000E+00 ordinate of mass center = 0.11750002E+03 tia tensor of body = 0.2699E+04 0.0000E+00 0.2441E-03 = 0.0000E+00 0.2201E+04 0.0000E+00 = 0.2441E-03 0.0000E+00 0.7757E+03	
i11 =	cipal inertias of body = 0.2699E+04 = 0.2201E+04	
Keyword:	total mass of body	ext Previous
Offset:	12	
Length:	14	
		ок



- 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.

