

altairhyperworks.com

HS-1025: Working with a HyperMesh and HyperMorph Model

In this tutorial you will learn how to import size and shape variables to HyperStudy from HyperMesh. The input variables are three shape variables; xtrans, ytrans and radius. Each of these shapes are created by perturbing the mesh in the corresponding directions by 1 unit.

The sample base input template used in this tutorial can be found in <hst.zip>/HS-1025/. Copy the tutorial files from this directory to your working directory.



Figure 1: Double Symmetric Plate Model

Figure 2: Double Symmetric Plate Model with Shape Vectors

Step 1: Perform the Study Setup

- 1. Start HyperStudy.
- 2. To start a new study, click *File* > *New* from the menu bar, or click *on the toolbar*.
- 3. In the **HyperStudy Add** dialog, enter a study name, select a location for the study, and click **OK**.
- 4. Go to the **Define Models** step.
- 5. Add a HyperMesh model.
 - a. From the **Directory**, drag-and-drop the plate_with_shapes.hm into the work area.



E	Explorer	Directory	,		s	Define Mo	odels		
Nam	ie 👘		Size	Туре	_				
۵ 鷆 ۵	C:\HS-102	5			Ð	Add Mode	I (🗶 Rer	nove Model
	🝝 study_l	lock.xml	858 bytes	Study Lock	Active	Label	Varnan	ne	Model Type
	🛆 plate_v	vith_shapes.hm	118 KB	hm File		Caber			model type
	📄 plate_v	vith_shapes.fem	101 KB	fem File					_
⊳	📗 _usr			Settings Fold	4	💪 plate_v	vith_sha	pes.hm	<u> </u>
	Study_	1.xml	3 KB	xml File					۰ ۲

- b. In the **Solver input file** column, enter plate.fem. This is the name of the solver input file HyperStudy writes during any evaluation.
- c. In the **Solver execution script** column, select **OptiStruct (os)**.

A	ctive	Label	Varname	Model Type	Resource	Solver input file	Solver execution script	Solver input arguments
1	V	Model 1	m_1	HyperMesh	C:/HS-1025/plate_with_shapes.hm 📂	plate.fem	OptiStruct (os)	\${file}

- 6. Click *Import Variables*.
- 7. In the **Model Parameters** dialog, select parameters to import into HyperStudy.
 - a. Select the thickness and shape variables. A total of 6 parameters should be selected.
 - b. Click Add.
 - c. Click OK.

🛆 Model Paramete	rs		
Variable name: Initial value: Lower bound: Upper bound: HyperMesh Model P - Model - Thickness - Shape - Shape - Shape - Shape - Ytrans. - Vytrans. - FORCE	radius.S 0.0 -1.0 1.0 arameters	 Apply to all selected iten Apply to all selected iten HyperStudy Parameters shell.T.1 patch1.T.1 patch2.T.1 xtrans.S ytrans.S radius.S 	ns Add ns Remove
		OK	Cancel



- 8. Go to the **Define Input Variables** step.
- 9. Review the input variable's lower and upper bound ranges.
- 10. Go to the **Specifications** step.

Step 2: 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*.
- 5. Go to the **Define Output Responses** step.

Step 3: Create and Define Output Responses

In this step you will create two output responses: Mass and Displacement.

- 1. Create the Mass output response.
 - a. From the **Directory**, drag-and-drop the plate.out 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, and then click **Next**.
 - Set **Type** to *Mass*.
 - Set **Request** to *Mass*.
 - Set Component to Value.



🔮 File Assistant								
Single serial or time series								
Subcase:								
Type:	Mass							
Request:	Mass Filter V							
Component:	Value 🔹							
✓ Preview:	2.00000E-006 1.90000E-006 1.80000E-006 1.70000E-006 1.60000E-006 0.90 0.95 1.00 1.05 1.10 Index							
	<back next=""> Cancel</back>							

e. Label the output response Mass.

f. Set Expression to *First Element*.

Note: Because there is only a single value in this data source, [0] is inserted after m_1_ds_1, thereby choosing the first (and only) entry in the data source.

1	File Assistar	ıt		×				
	Create a Data Source and a Response							
	Creating a new Data Source							
	Label: Data Source 1							
	2	Varname: n	n_1_ds_1					
	🔽 Linked t	o a new Resp	onse					
		Label:	Mass					
		Varname:	m_1_r_1					
		Comment:	Data Source 1					
		Expression:	m_1_ds_1[0] First Element					
_								
	< Back Finish Cancel							



- g. Click *Finish*. The Mass output response is added to the work area.
- 2. Create the Displacement output response.
 - a. From the **Directory**, drag-and-drop the plate.h3d 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, and then click **Next**.
 - Set Subcase to Subcase 1 (Load).
 - Set Type to *Displacement (Grids)*.
 - Set **Request** to **N298**.
 - Set **Component** to **MAG**.
 - e. Label the output response Displacement.
 - f. Set Expression to *First Element*.

₫	File Assistar	nt		×				
	Create a Data Source and a Response							
	Creating a new Data Source							
		Label: [Data Source 2					
	> -	Varname: n	n_1_ds_2					
	✓ Linked t	o a new Resp	onse					
		Label:	Displacement					
		Varname:	m_1_r_2					
		Comment:	Data Source 2					
		Expression:	m_1_ds_2[0] First Element 🔻)				
_			< Back Finish Cance	ـــــــــــــــــــــــــــــــــــــ				

- g. Click *Finish*. The Displacement output response is added to the work area.
- 3. Click *Evaluate* to extract the output response values.



	Active	Label	Varname	Expression	Value	Comment
1	V	Mass	m_1_r_1	m_1_ds_1[0]	1.83e-06	Data Source 1
2	V	Displacement	m_1_r_2	m_1_ds_2[0]	0.0024906	Data Source 2

4. Proceed to the desired study type (DOE, Optimization, of Stochastic study).

Last modified: v2017.2 (12.1156684)

