Altair

HyperWorks

Altair HyperStudy 2019 Tutorials

HS-1080: Setting Up an Operator Model

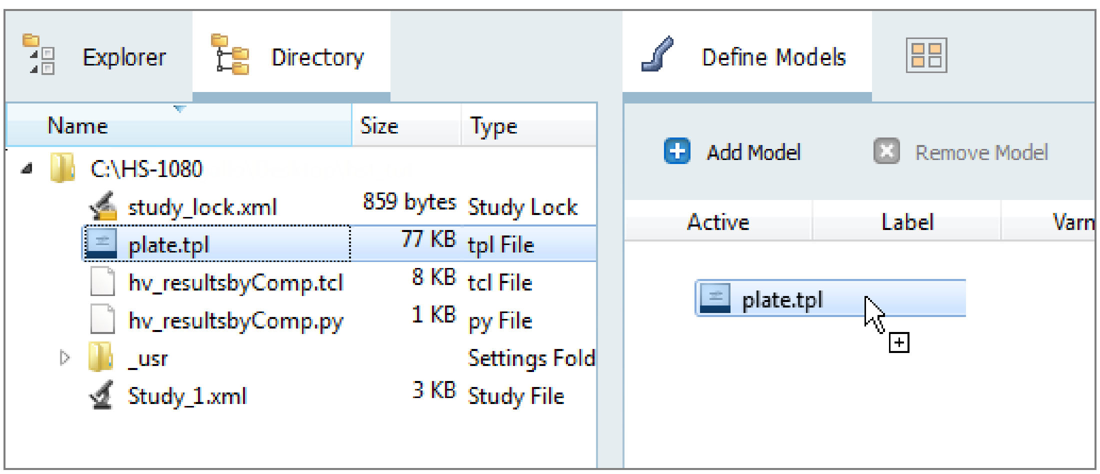
HS-1080: Setting Up an Operator Model

In this tutorial you will use the Operator model type to run a script that uses a combination of HyperView and HVTrans to split the solver result file in multiple result files, one for each component in the model. This tutorial uses a model which consists of a plate with a hole which is loaded in plane. The design has three thickness variables; one for each zone. The output responses of interest are the maximum stress in each of the three zones.

The files used in this tutorial can be found in `<hst.zip>/HS-1080/`. Copy the tutorial files from this directory to your working directory.

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 Parameterized File model.
   a. From the **Directory**, drag-and-drop the `plate.tpl` file into the work area.



   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)***.

6. Click ***Import Variables***. Three input variables are imported from the `plate.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 **Specification** 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***.

Step 3: Setup the Operator Model

This step requires the `plate.h3d` file generated in step 2, which is why you cannot setup the operator model until after the nominal run.

1. Go back to the **Define Models** step.
2. Add an Operator model.
   a. Click ***Add Model***.
   b. In the **Add - HyperStudy** dialog, select ***Operator*** and click ***OK***.
   c. Set the **Solver execution script** to ***Python (py)***.
   d. In the **Solver input arguments** field, enter `${m_2.file_1} ${m_2.file_3}`.

      The input arguments are references to the model resources' varnames. The first argument (m_2.file_1) is a reference to the model resource's varname, and tells python which script to run. The second argument (m_2.file_3) is the varname to the target result file to split, and will be the first argument to the python script.
3. Define model dependencies.
   a. Click ***Model Resources***. The **Model Resources** dialog opens.
   b. Define a model dependency that references the python script that will be used as the solver script. This is a reference to a file that is not generated during a solver run, therefore it is of type Normal. This file does not need to be in the run directory.
      i. Verify that **Model 2 (m2)** is selected.
      ii. Click ***Add Resource*** > ***Add Input Resource***.
      iii. In the **Select File** dialog, navigate to your working directory and open the `hv_resultsbyComp.py` file.
      iv. Set **Operation** to ***None***.
   c. Define a model dependency that references the tcl script that will be used to run HyperView and Hvtrans in batch. This is a reference to a file that is called by the python script and not by a solver, therefore it is of type Normal. This file is required to be in the run directory.

i. Verify that **Model 2 (m2)** is selected.

ii. Click ***Add Resource*** > ***Add Input Resource***.

iii. In the **Select File** dialog, navigate to your working directory and open the `hv_resultsbyComp.tcl` file.

iv. Set **Operation** to ***Copy***.

d. Define a model dependency that will be used as a link to the result file from the first model. This file is the target file for the tcl script, and it is required to be in the run directory. It is a file that changes for each run, therefore it is a linked file and it is copied in the run directory. Note that the file can be moved.

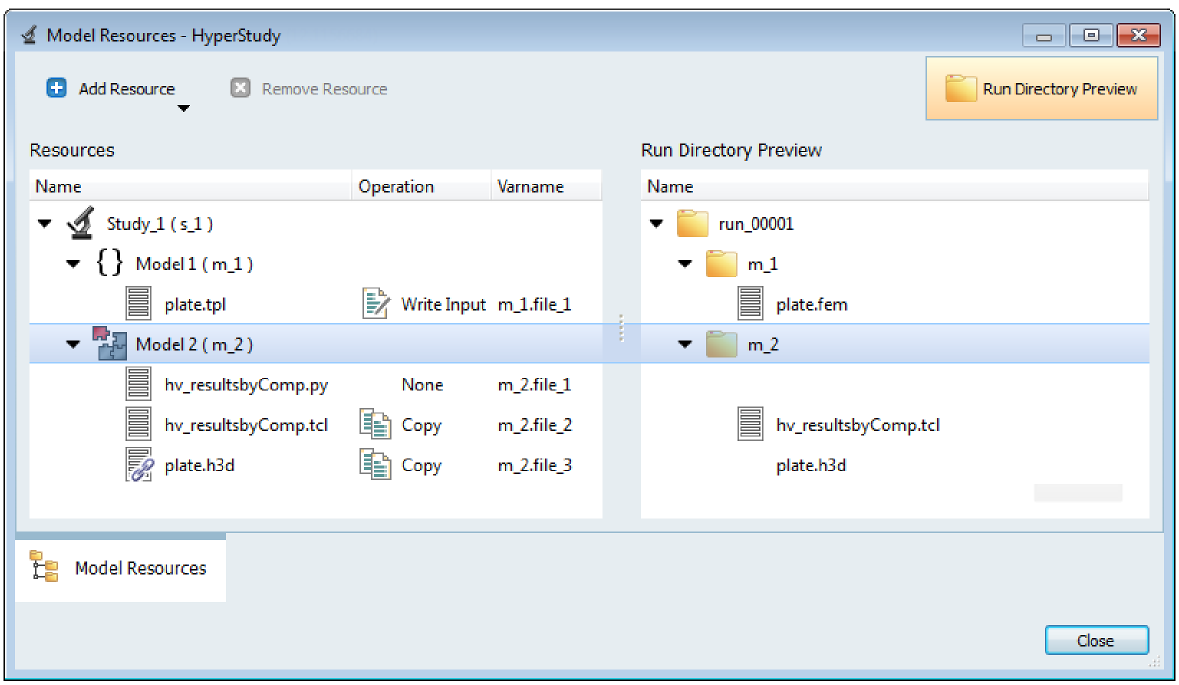
i. Verify that **Model 2 (m2)** is selected.

ii. Click ***Add Resource*** > ***Add Link Resource***.

iii. In the **Select File** dialog, navigate to the `approaches/nom_1/run__00001/m_1` directory and open the `plate.h3d` file.

iv. Set **Operation** to ***Copy***.

e. Click ***Close*** to exit the **Model Resources** dialog.



4. Go to the **Specifications** step.

Step 4: Perform the Nominal Run

In this step you will perform the same steps as Step 2, except during this nominal run the Operator model will also be run.

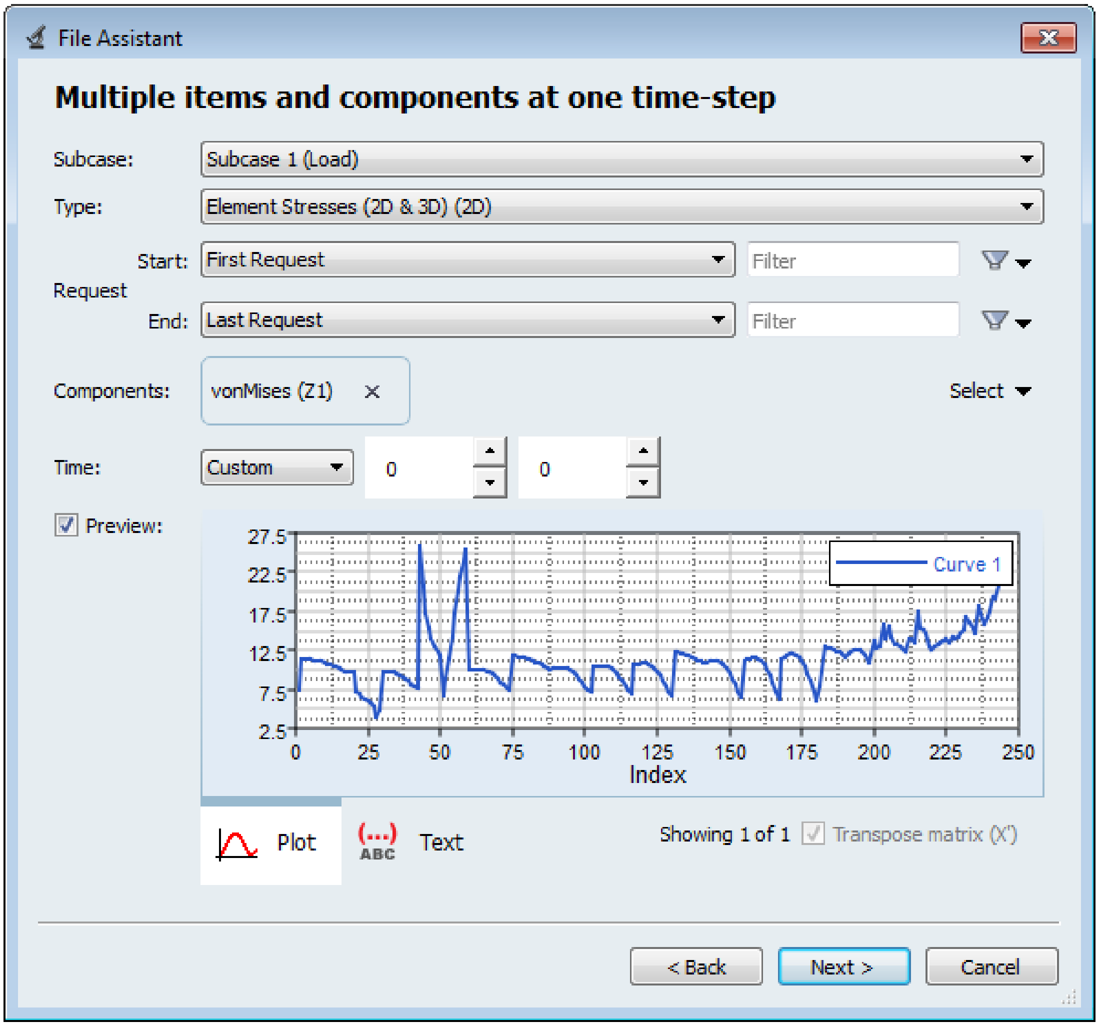
1. In the work area, set the **Mode** to ***Nominal Run***.

2. Click ***Apply***.
3. In the **HyperStudy** dialog, click ***Yes*** to overwrite the run matrix.
4. Go to the **Evaluate** step.
5. Click ***Evaluate Tasks***.
6. In the **HyperStudy** dialog, click ***Yes*** to overwrite files.

Step 5: Define Output Responses

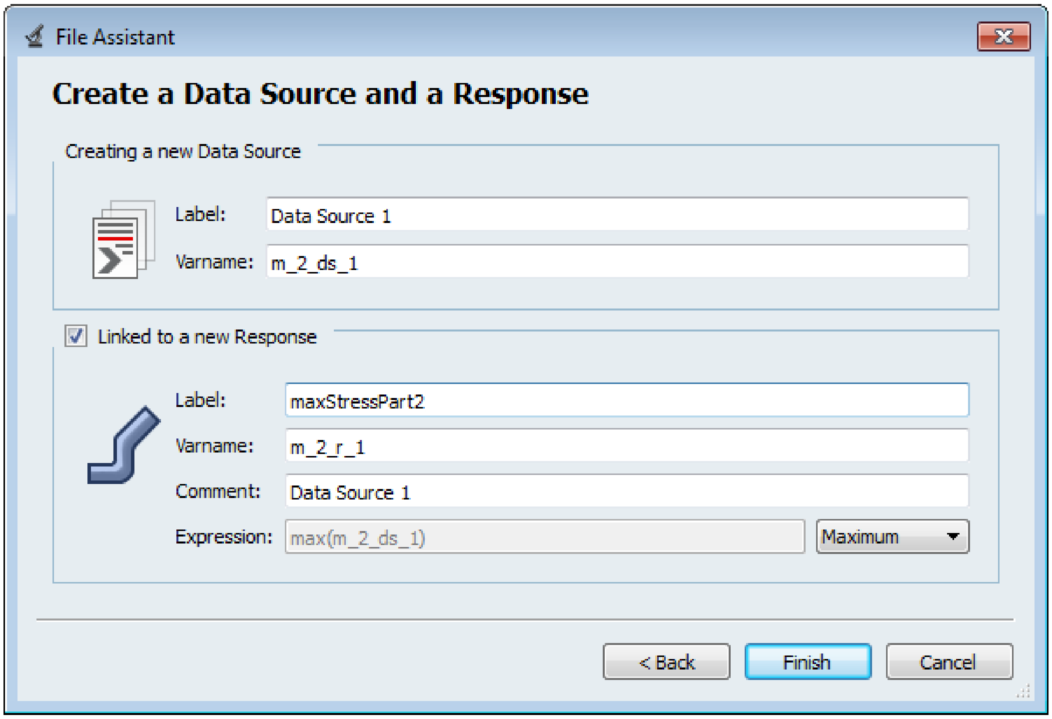
In this step you will create four output responses: maxStressPart2, maxStressPart4, maxStress3, Volume.

1. Go to the **Define Output Responses** step.
2. Create the maxStressPart2 output response.
   a. From the **Directory**, drag-and-drop the `plate_2_shell.h3d` file, located in `approaches/nom_1/run_00001/m_2`, into the work area.
   b. In the **File Assistant** dialog, set the **Reading technology** to ***Altair® HyperWorks® (Hyper3D Reader)*** and click ***Next***.
   c. Select **Multiple items at multiple time steps**, then click ***Next***.
   d. Define the following options, and then click ***Next***.
      - Set **Subcase** to ***Subcase 1(Load)***.
      - Set **Type** to ***Element Stresses (2D & 3D) (2D)***.
      - For **Request**, set **Start** to ***First Request*** and **End** to ***Last Request***.
      - For **Components**, select ***vonMises (Z1)***.
      - For **Timestep**, select ***Custom*** and enter `0`.



e. Label the output response maxStressPart2.

f. Set **Expression** to ***Maximum***.



g. Click ***Finish***. The maxStressPart2 output response is displayed in the work area.

3. Create the maxStressPart4 output response.

   a. From the **Directory**, drag-and-drop the `plate_4_patch2.h3d` file, located in `approaches/nom_1/run_00001/m_2`, into the work area.

   b. In the **File Assistant** dialog, set the **Reading technology** to ***Altair® HyperWorks® (Hyper3D Reader)*** and click ***Next***.

   c. Select **Multiple items at multiple time steps (readsim)**, then click ***Next***.

   d. Define the following options, and then click ***Next***.

      - Set **Subcase** to ***Subcase 1(Load)***.
      - Set **Type** to ***Element Stresses (2D & 3D) (2D)***.
      - For **Request**, set **Start** to ***First Request*** and **End** to ***Last Request***.
      - For **Components**, select ***vonMises (Z1)***.
      - For **Timestep**, select ***Custom*** and enter `0`.

   e. Label the output response maxStressPart4.

   f. Set **Expression** to ***Maximum***.

   g. Click ***Finish***. The maxStressPart4 output response is added to the work area.

4. Create the maxStress3 output response.

   a. From the **Directory**, drag-and-drop the `plate_3_patch1.h3d` file, located in `approaches/nom_1/run_00001/m_2`, into the work area.

   b. In the **File Assistant** dialog, set the **Reading technology** to ***Altair® HyperWorks® (Hyper3D Reader)*** and click ***Next***.

   c. Select **Multiple items at multiple time steps (readsim)**, then click ***Next***.
   d. Define the following options, and then click ***Next***.
      - Set **Subcase** to ***Subcase 1(Load)***.
      - Set **Type** to ***Element Stresses (2D & 3D) (2D)***.
      - For **Request**, set **Start** to ***First Request*** and **End** to ***Last Request***.
      - For **Components**, select ***vonMises (Z1)*** and ***vonMises (Z2)***.
      - For **Timestep**, select ***Custom*** and enter `0`.
   e. Label the output response maxStress3.
   f. Set **Expression** to ***Maximum***.
   g. Click ***Finish***. The maxStress3 output response is added to the work area.
5. Create the Volume 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® (osmass.tpl)*** 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 ***OptiStruct Analysis***.
      - Set **Request** to ***Out File***.
      - Set **Component** to ***Volume***.
   e. Label the output response Volume.
   f. Set **Expression** to ***Maximum***.
   g. Click ***Finish***. The Volume output response is added to the work area.
6. Click ***Evaluate*** to extract the output response values.

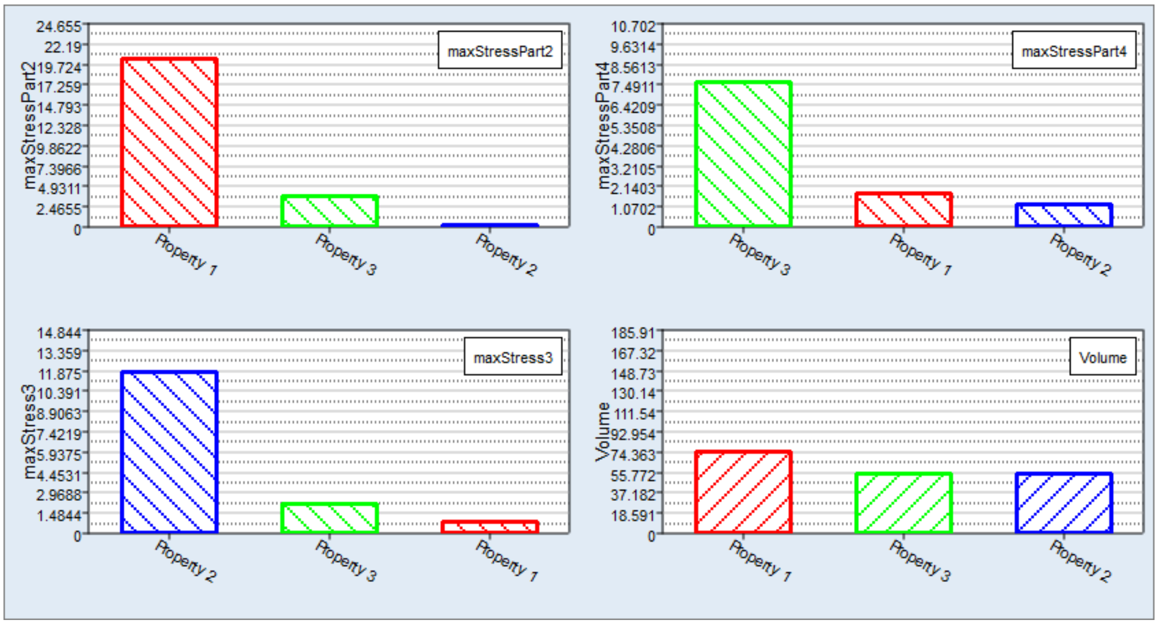
	Active	Label	Varname	Expression	Value	Comment
1	☑	maxStressPart2	m_2_r_1	max(m_2_ds_1) ...	25.931837	Data Source 1 ...
2	☑	maxStressPart4	m_2_r_2	max(m_2_ds_2) ...	11.523326	Data Source 2 ...
3	☑	maxStress3	m_2_r_3	max(m_2_ds_3) ...	15.796097	Data Source 3 ...
4	☑	Volume	m_1_r_1	max(m_1_ds_1) ...	232.38400	Data Source 4 ...

Step 6: Define the DOE

1. In the **Explorer**, right-click and select ***Add*** from the context menu.
2. In the **Add - HyperStudy** dialog, select ***Doe*** and click ***OK***.

3. Go to the **Specifications** step.
4. In the work area, set the **Mode** to ***Modified Extensible Lattice Sequence***.
5. Click ***Apply***.
6. Go to the **Evaluate** step.
7. Click ***Evaluate Tasks***.
8. Go to the **Post-Processing** step.
9. Click the ***Pareto Plot*** tab to plot the effects of variables on output responses in hierarchical order (highest to lowest).

   Each variable contributes nearly equally to volume. A positive hashing indicates that the relationship is positive: as the variable increases, mass increases. For the three stress output responses, the maximum stress in each zone is dominated by the thickness of that zone.



Last modified: v2017.2 (12.1156684)