Altair

HyperWorks

Altair HyperStudy 2019 Tutorials

HS-4450: Multi-Objective Optimization of a Cantilever Ibeam using an Inclusion Matrix

altairhyperworks.com

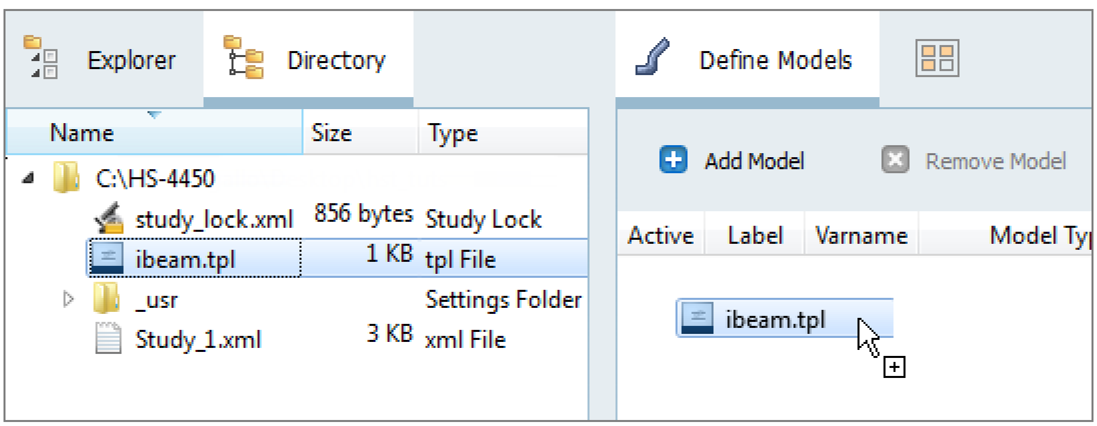
HS-4450: Multi-Objective Optimization of a Cantilever Ibeam using an Inclusion Matrix

The inclusion matrix feature passes an already existing set of data to the running process. In this tutorial, the data created from a DOE is passed to an optimization problem which re-uses the data. This promotes efficient design exploration practices: an optimization using a direct solver call can still be done in combination with a DOE to study the system without any loss of data. This example focuses on the competing objectives in the design of a cantilever ibeam.

The files used in this tutorial can be found in `<hst.zip>/HS-4450/`. Copy the 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 `ibeam.tpl` file into the work area.



   b. In the **Solver input file** column, enter `ibeam.py`. This is the name of the solver input file HyperStudy writes during any evaluation.
   c. In the **Solver execution script** column, select ***Python (py)***.

	Active	Label	Varname	Model Type	Resource	Solver input file	Solver execution script	Solver input arguments
1	☑	Model 1	m_1	{ } Parameterized File	C:/.../HS-4450/ibeam.tpl	ibeam.py	Python (py)	${file}

6. Click ***Import Variables***. Four input variables are imported from the `ibeam.tpl` resource file.
7. Go to the **Define Input Variables** step.

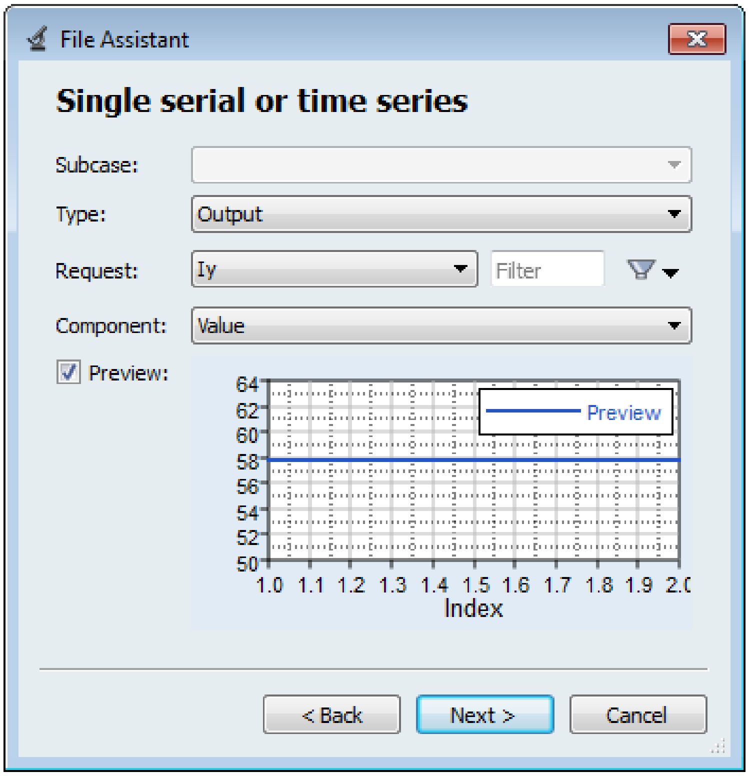
8. Review the input variable's lower and upper bounds ranges.
9. Go to the **Specifications** step.

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***. An `approaches/nom_1/` directory is created inside the study directory. The `approaches/nom_1/run__00001/m_1` sub-directory contains the `output.hstp` file, which is the result of the nominal run, and will be used during the Optimization.
5. Go to the **Define Output Responses** step.

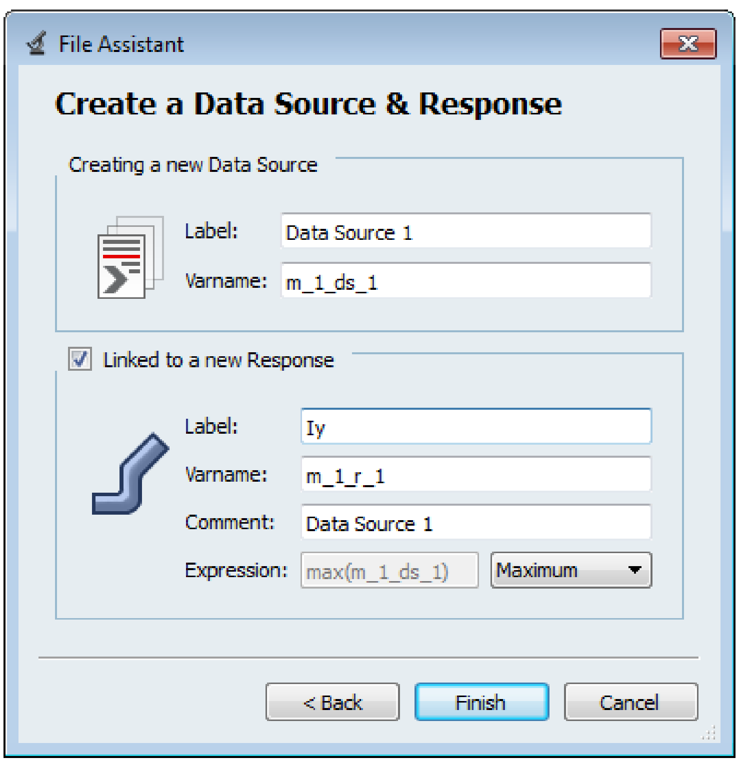
Step 3: Create and Define Output Responses

1. Create the Iy output response for the y-axis moment of inertia.
   a. From the **Directory**, drag-and-drop the `output.hstp` 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® (HstReaderPdd)*** 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 ***Output***.
      - Set **Request** to ***Iy***.
      - Set **Component** to ***Value***.



e. Label the output response Iy.

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



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

2. Create four more output responses by repeating step 1, except change the **Request** assigned to each output response to the following:

Output Response	Request
Volume	Vol
IZ	Iz
Displacement	d
Frequency1	Freq1

3. Click ***Evaluate*** to extract output response values.

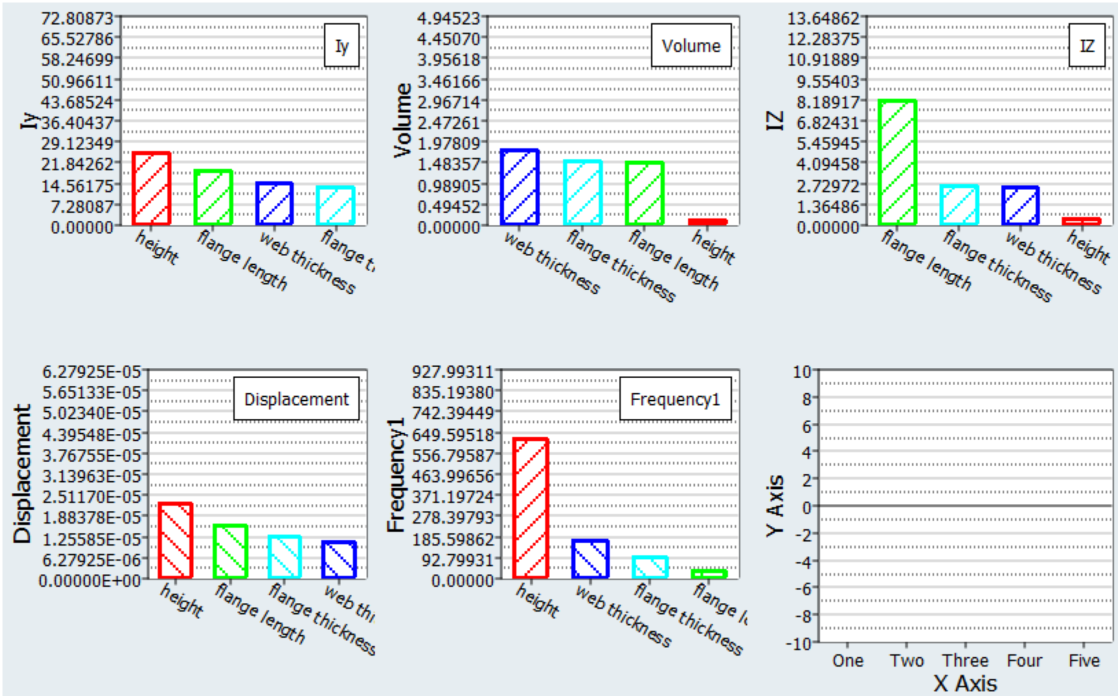
	Active	Label	Varname	Expression	Value	Comment
1	☑	Iy	m_1_r_1	max(m_1_ds_1) ...	64.034100	Data Source 1 ...
2	☑	Volume	m_1_r_2	max(m_1_ds_2) ...	5.4400000	Data Source 2 ...
3	☑	IZ	m_1_r_3	max(m_1_ds_3) ...	8.3381300	Data Source 3 ...
4	☑	Displacement	m_1_r_4	max(m_1_ds_4) ...	5.21e-05	Data Source 4 ...
5	☑	Frequency1	m_1_r_5	max(m_1_ds_5) ...	3205.7200	Data Source 5 ...

Step 4: Run a Hammersley DOE Study

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 ***Hammersley***.
5. In the **Settings** tab, verify that the **Number of runs** is 17.
6. Click ***Apply***.
7. Go to the **Evaluate** step.
8. Click ***Evaluate Tasks***.
9. Go to the **Post-Processing** step.
10. Click the ***Pareto Plot*** tab.

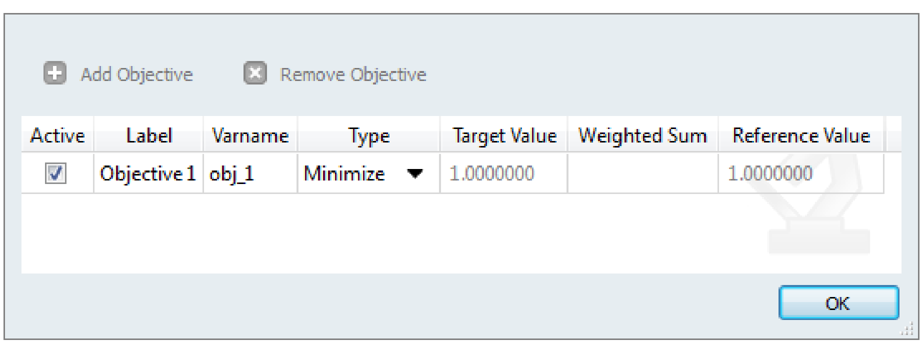
    Enable **multi-plot** and select all of the output responses from the **Channel** selector. In the options menu, ensure that **Linear effects** is enabled.

    A Pareto Plot shows the ranked influence of the input variables on the output response. For example, for the y-axis moment of the inertia, height has the largest influence and web thickness has the least. In contrast, for the z-axis moment of inertia, the flange length and flange thickness are the most influential variables. The size of the bar indicates the magnitude of the influence, and the hashed line's slope indicates the sign of the effect: positive or negative. For example, increasing the height will increase Iy, but it will decrease displacement.



Step 5: Run an Optimization Study

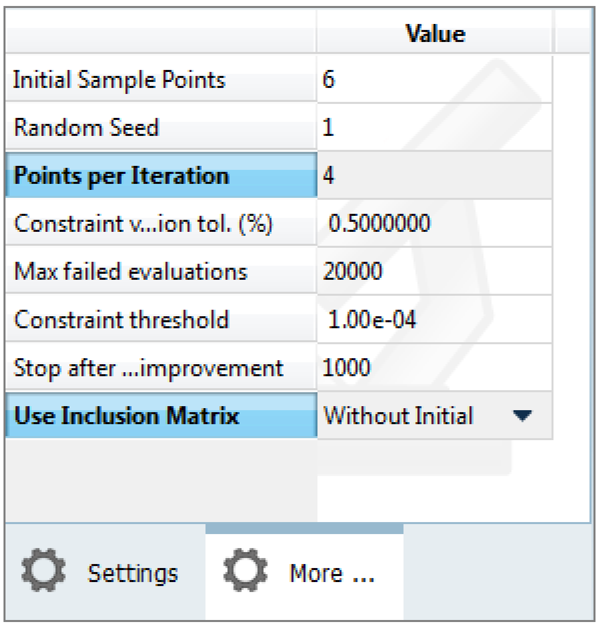
1. In the **Explorer**, right-click and select ***Add*** from the context menu.
2. In the **Add - HyperStudy** dialog, select ***Optimization*** and click ***OK***.
3. Go to the **Select Input Variables** step.
4. Review the input variable's lower and upper bound ranges.
5. Go to the **Select Output Responses** step.
6. Apply an objective on the Volume output response.
   a. In the **Objectives** column for **Volume**, click .
   b. In the pop-up window, set **Type** to ***Minimize*** and click ***OK***.



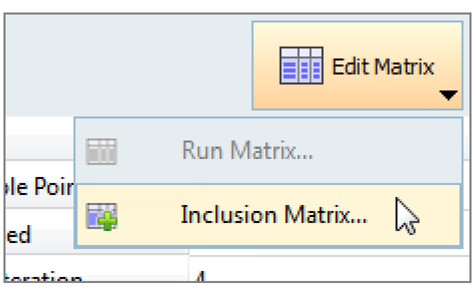
7. Apply an objective on the Iy output response.
   a. In the **Objectives** column for Iy, click .
   b. In the pop-up window, set **Type** to ***Maximize*** and click ***OK***.
8. Click ***Apply***.
9. Go to the **Specifications** step.
10. In the work area, set the **Mode** to ***Global Response Search Method (GRSM)***.

    Note: Only the methods that are valid for the problem formulation are enabled.
11. Click the ***More*** tab and define the following settings:
    - Set **Points per Iteration** to 4.
    - Set **Use Inclusion Matrix** to **Without Initial**.

      GRSM performs a global search, therefore the initial values of the variables are not important and do not have to be used within the optimization.

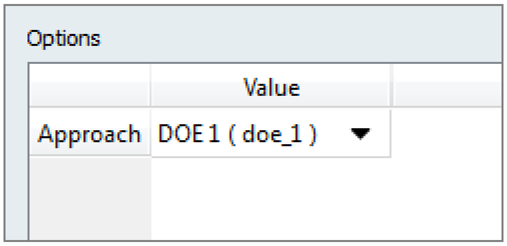


12. Import run data from the DOE using an Inclusion Matrix.
    a. Click ***Edit Matrix*** > ***Inclusion Matrix*** from the top, right corner of the work area.



    b. In the **Edit Inclusion Matrix** dialog, click ***Import Values***.
    c. In the **Import Values** dialog, select ***Approach evaluation data*** and click ***Next***.

d. Set the approach to ***DOE 1*** and click ***Next***.

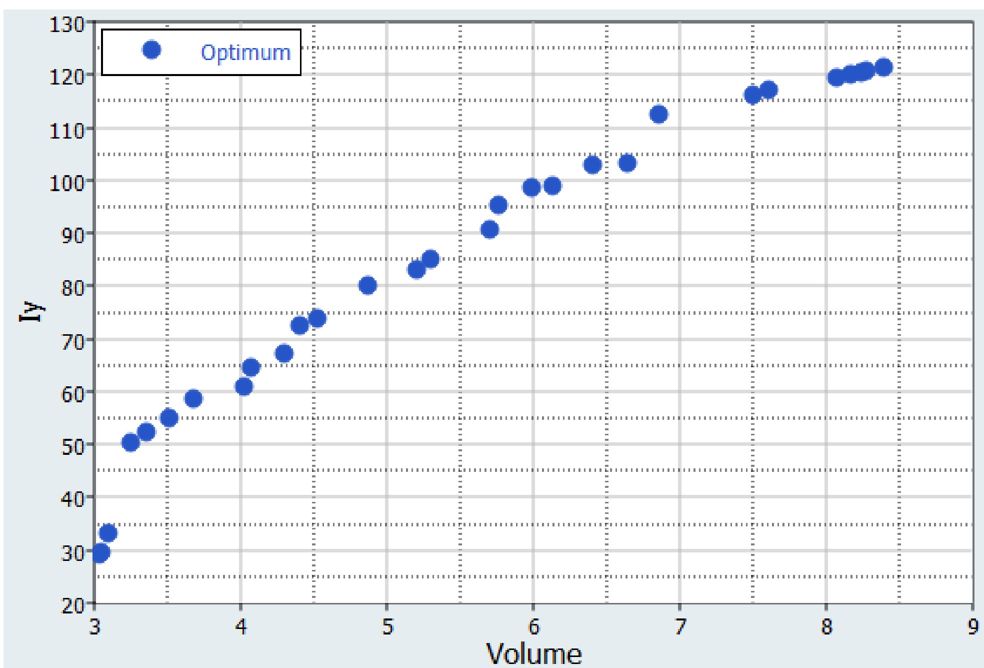


e. Click ***Finish***.

f. Review the imported run data and click ***OK***.

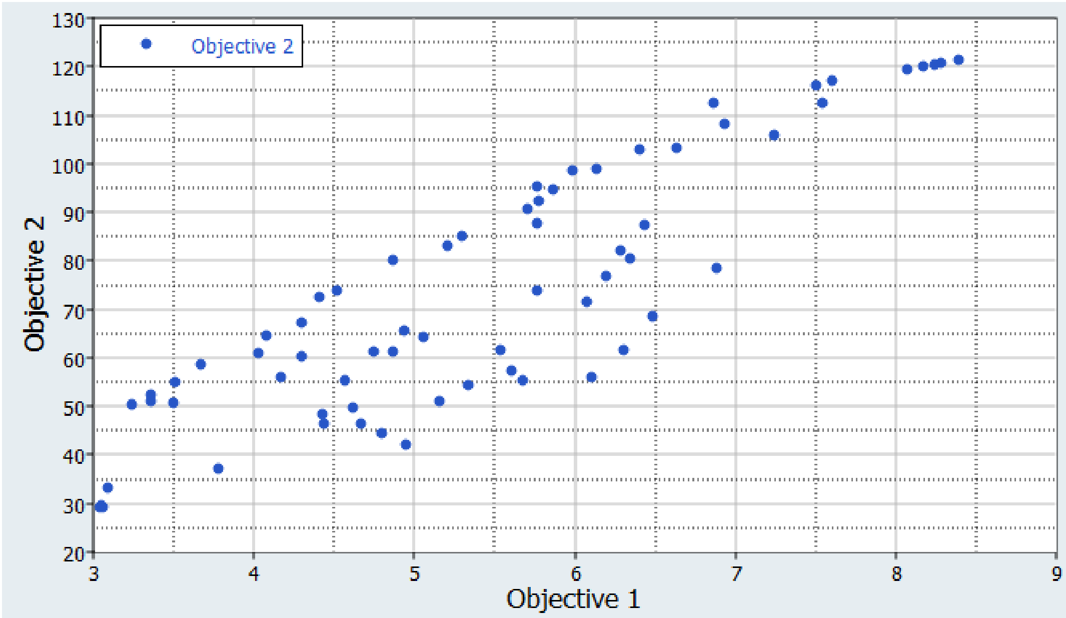
13. Click ***Apply***.
14. Go to the **Evaluate** step.
15. Click ***Evaluate Tasks*** to launch the Optimization.
16. Go to the **Post-Processing** step.
17. Click the ***Optima*** tab.

    Observe the non-dominated front of designs. These points represent the trade-off between the objective of minimizing volume and maximizing the y-axis moment of inertia. In the plot, it is evident that as the moment of inertia increases, the volume increases as well. This curve represents the trade-off of the best available designs given the competing objective requirements.



18. Click the **Scatter** tab to plot the objectives along the same axes shown in the Optima plot.

    This scatter plot shows all of the runs from the optimization. When comparing the scatter and optima plots, note that the optima plot contains only a subset of runs which are non-dominated. A dominated design is a design for which both objectives could be improved. A non-dominated design is one in which one objective may only be improved at the expense of another.



Last modified: v2017.2 (9.1144596)