Altair

HyperWorks

Altair HyperStudy 2019 Tutorials

HS-4210: Multi-Disciplinary Optimization Study

HS-4210: Multi-Disciplinary Optimization Study

This tutorial demonstrates how to perform a multi-disciplinary size optimization for two finite element models defined for OptiStruct that have common input variables. The sample base input templates plate1.tpl and plate2.tpl, can be found in <hst.zip>/HS-4210/ and copied to your working directory.

The objective is to minimize the volume of the plate under a stress and a frequency constraint. The input variables are the thickness of each of the three components, defined in the input deck via the PSHELL card. The thickness should be between 0.05 and 0.15; the initial thickness is 0.1 (shown below). The optimization type is size. To demonstrate the use of the optimization tool in a multi-disciplinary optimization, two models are created. One model is used for the stress analysis and one for the frequency analysis. Both models must have the same input variables.

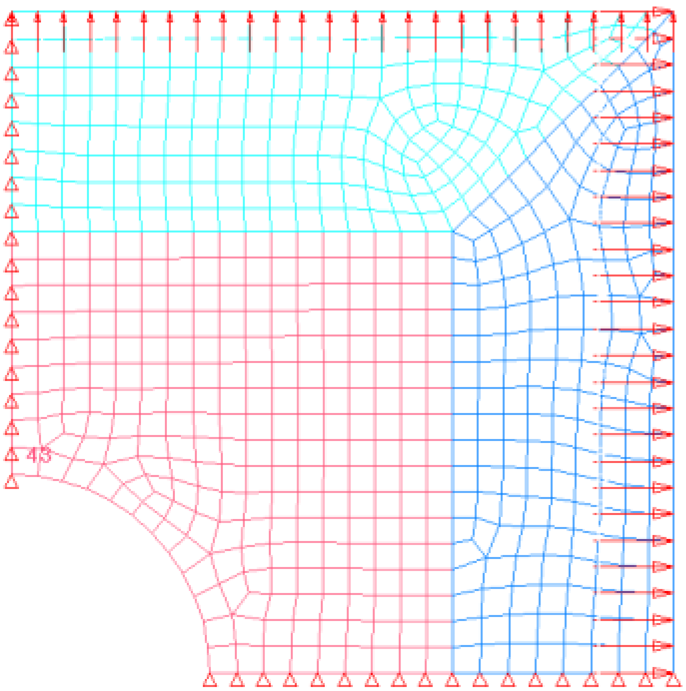
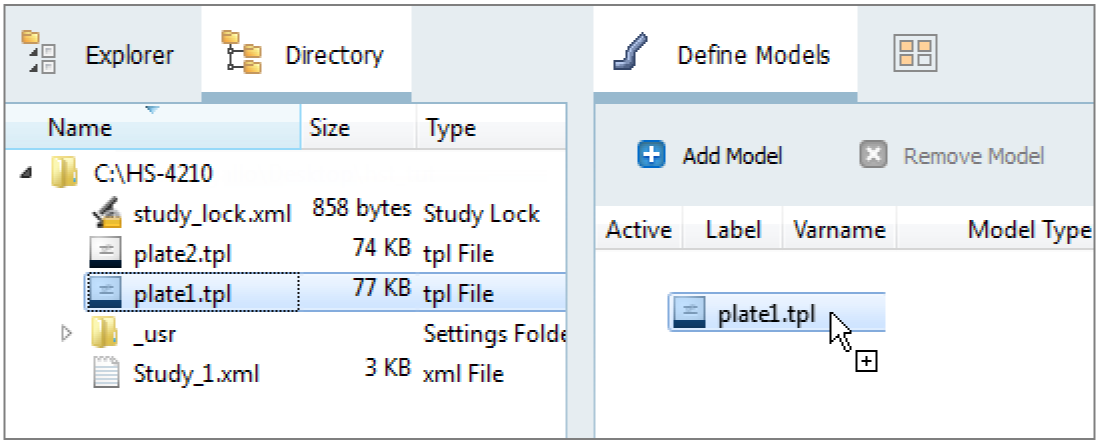


Figure 1. Double symmetric plate model.

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 plate1.tpl file into the work area.



b. In the **Solver input file** column, enter `plate1.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. Add a Parameterized File model.

a. From the **Directory**, drag-and-drop the `plate2.tpl` file into the work area.

b. In the **Solver input file** column, enter `plate2.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)***.

	Active	Label	Varname	Model Type	Resource	Solver input file	Solver execution script	Solver input arguments
1	☑	Model 1	m_1	{} Parameterized File	C:/.../HS-4210/plate1....	plate1.fem	OptiStruct (os)	${file}
2	☑	Model 2	m_2	{} Parameterized File	C:/.../HS-4210/plate2....	plate2.fem	OptiStruct (os)	${file}

7. Click ***Import Variables***. Six input variables are imported from the `plate1.tpl` and `plate2.tpl` resource files.

8. Go to the **Define Input Variables** step.

9. Review the input variable's lower and upper bound ranges.

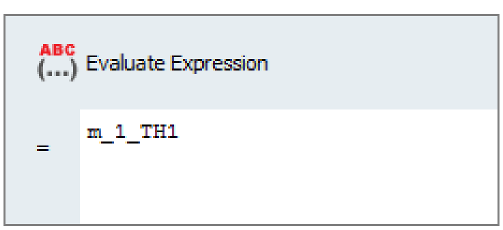
10. Click the ***Links*** tab.

11. In the **Expression** column of the input variable **Property 21**, click **...**.

12. In the **Expression Builder**, click the ***Input Variables*** tab.

13. In the work area, select ***Property 11***.

14. Click ***Insert Varname***. The expression `m_1_TH1` appears in the **Evaluate expression** field.



15. Click ***OK***. Property 21 of Model 2 is linked to Property 11 of Model 1.
16. Create two more links.
    - Link **Property 22** to **Property 12**.
    - Link **Property 23** to **Property 13**.

	Active	Label	Varname	Expression
1	☑	Property 11	m_1_TH1	...
2	☑	Property 12	m_1_TH2	...
3	☑	Property 13	m_1_TH3	...
4	☑	Property 21	m_2_TH1	m_1_TH1 ...
5	☑	Property 22	m_2_TH2	m_1_TH2 ...
6	☑	Property 23	m_2_TH3	m_1_TH3 ...

17. Go to the **Specifications** step.

Step 2: Perform the Nominal Run

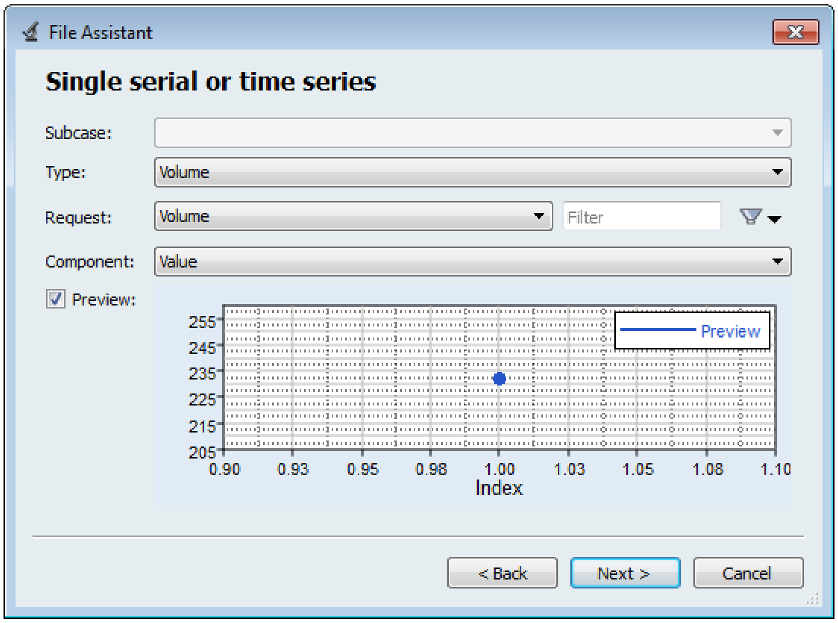
1. In the work area, set the **Mode** to ***Nominal Run***.
2. Click ***Apply***.
3. Go to the **Evaluate** step.
4. Click ***Evaluate Tasks***. An `approaches/nom_1/` directory is created inside the study directory. The `approaches/nom_1/run__00001/m_1` and `approaches/nom_1/run__00001/m_2` sub-directories contain the `plate2.out` (for the structural volume and frequency) and `plate1.h3d` (for the stresses) files, which are the results of the nominal run, and will be using during the Optimization.
5. Go to the **Define Output Response** step.

Step 3: Create and Define Output Responses

In this step you will create three output responses: Volume, Stress43, and Frequency1.

1. Create the Volume output response, which represents the volume of the plate.
    a. From the **Directory**, drag-and-drop the `plate2.out` file, located in the `approaches/nom_1/run__00001/m_2` directory, 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, then click ***Next***.

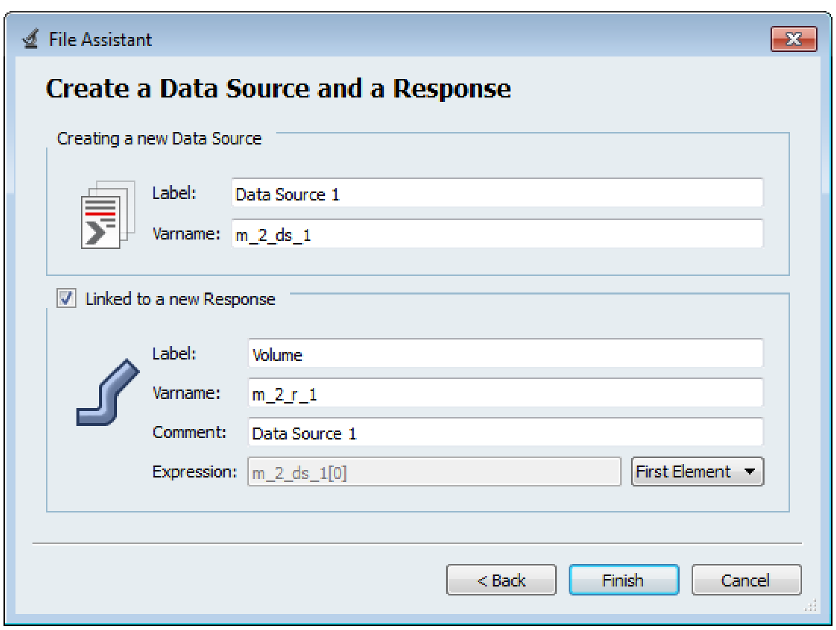
- Set **Type** to ***Volume***.
- Set **Request** to ***Volume***.
- Set **Component** to ***Value***.



e. Label the output response Volume.

f. Set **Expression** to ***First Element***.

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



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

2. Create the Stress43 output response, which represents the von Mises Stress of Element 43.
   a. From the **Directory**, drag-and-drop the `plate1.h3d` file, located in the `approaches/nom_1/run__00001/m_1` directory, into the work area. This file contains the analysis results, including stresses.
   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, then click ***Next***.
      - Set **Subcase** to ***Subcase 1(Load)***.
      - Set **Type** to ***Element Stresses (2D & 3D) (2D)***.
      - Set **Request** to ***E43***.
      - Set **Component** to ***vonMises (Mid)***.
   e. Label the output response Stress43.
   f. Set **Expression** to ***First Element***.
   g. Click ***Finish***. The Stress43 output response is added to the work area.
3. Create the Frequency1 output response, which represents frequency results.
   a. From the **Directory**, drag-and-drop the `plate2.out` file, located in the `approaches/nom_1/run__00001/m_2` directory, into the work area. This file contains the analysis results, including stresses.
   b. In the **File Assistant** dialog, click ***Next***.
   c. Select ***Single item in a time series***, then click ***Next***.
   d. Define the following options, then click ***Next***.
      - Set **Type** to ***Frequency***.
      - Set **Request** to ***Mode 1***.
      - Set **Component** to ***Value***.
   e. Label the output response Frequency1.
   f. Set **Expression** to ***First Element***.
   g. Click ***Finish***. The Frequency1 output response is added to the work area.
4. Click ***Evaluate Expressions*** to extract the output response values.

	Active	Label	Varname	Expression	Value	Comment
1	☑	Volume	m_2_r_1	m_2_ds_1[0] ...	Not Extracted	Data Source 1 ...
2	☑	Stress43	m_1_r_1	m_1_ds_1[0] ...	Not Extracted	Data Source 2 ...
3	☑	Frequency1	m_2_r_2	m_2_ds_2[0] ...	Not Extracted	Data Source 3 ...

Step 4: 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 dialog, set **Type** to ***Minimize*** and click ***OK***.

	Active	Label	Varname	Objectives	Constraints	Evaluate From	Expression	Comment
1	☑	Volume	m_2_r_1	Minimize ...	⊞	Solver	m_2_ds_1[0]	Data Source 1 ...
2	☑	Stress43	m_1_r_1	⊞	⊞	Solver	m_1_ds_1[0]	Data Source 2 ...
3	☑	Frequency1	m_2_r_2	⊞	⊞	Solver	m_2_ds_2[0]	Data Source 3 ...

7. Apply a constraint on the Stress43 output response.
    a. In the **Constraints** column for Stress43, click ⊞.
    b. In the pop-up dialog, define the following and click ***OK***.
        - Set **Type** to ***Deterministic***.
        - Set **Bound Type** to **<=**.
        - For **Bound Value**, enter `22`.
8. Apply a constraint on the Stress43 output response.
    a. In the **Constraints** column for Frequency1, click ⊞.
    b. In the pop-up dialog, define the following and click ***OK***.
        - Set **Type** to ***Deterministic***.
        - Set **Bound Type** to **>=**.
        - For **Bound Value**, enter `32`.

	Active	Label	Varname	Objectives	Constraints	Evaluate From	Expression	Comment
1	☑	Volume	m_2_r_1	Minimize ...	⊞	Solver	m_2_ds_1[0]	Data Source 1 ...
2	☑	Stress43	m_1_r_1	⊞	<= 22.000000 ...	Solver	m_1_ds_1[0]	Data Source 2 ...
3	☑	Frequency1	m_2_r_2	⊞	>= 32.000000 ...	Solver	m_2_ds_2[0]	Data Source 3 ...

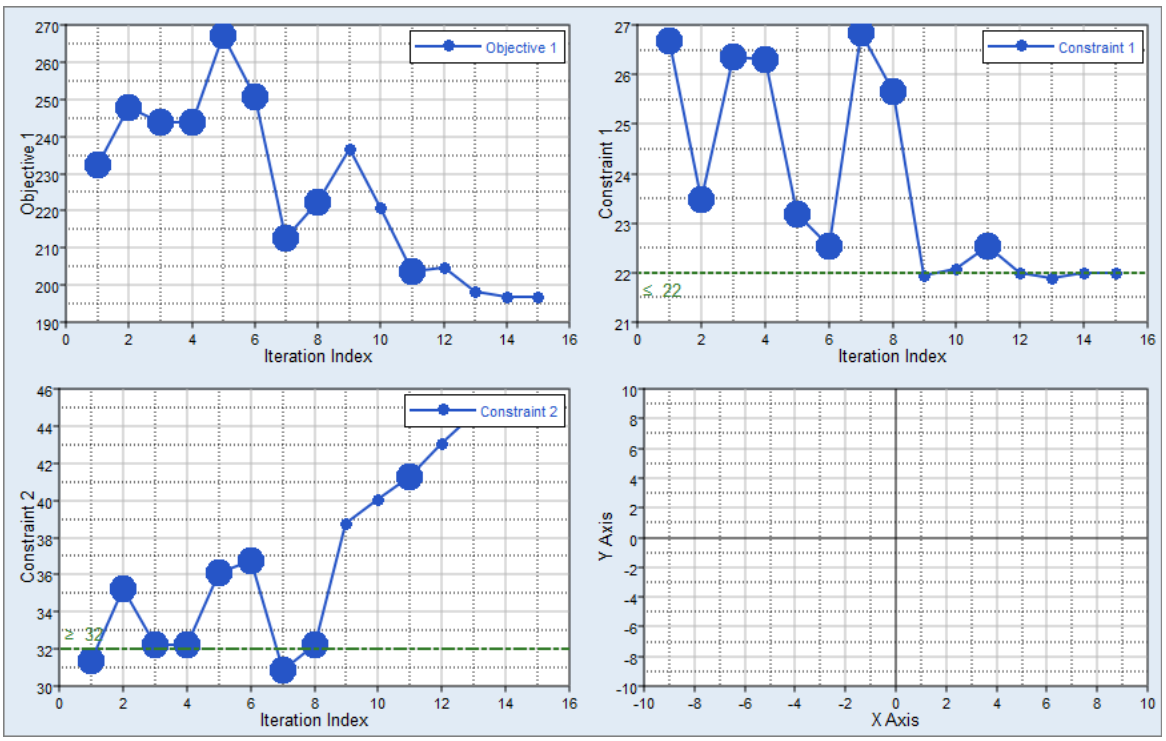
9. Click ***Apply***.
10. Go to the **Specifications** step.
11. In the work area, set the **Mode** to ***Adaptive Response Surface Method (ARSM)***.

    Note: Only the methods that are valid for the problem formulation are enabled.
12. Click ***Apply***.

13. Go to the **Evaluate** step.
14. Click ***Evaluate Tasks***.
15. Click the ***Iteration Plot*** tab to plot the progress of the Optimization iteration.

    Using the **Channel** selector, select ***Objective 1***, ***Constraint 1***, and ***Constraint 2***. Above the Channel selector, activate multiplot and enable the **Bounds** setting.

    Over the course of the optimization, the objective is minimized and at the conclusion, the constraints are satisfied. In the plots, the large markers indicate a design which has at least one violated constraint and a small marker indicates a feasible design. At the optimal design, the only active constraint is Constraint 1. In contrast, constraint 2 is not active at the optimum; this indicates Constraint 2 does not have an influence on the result.



Last modified: v2017.2 (12.1156684)