Altair

HyperWorks

Altair HyperStudy 2019 Tutorials

HS-4220: Size Optimization Study on an Impact Simulation Using RADIOSS

HS-4220: Size Optimization Study on an Impact Simulation Using RADIOSS

This tutorial demonstrates how to perform a size optimization on a finite element model defined for RADIOSS. The RADIOSS model shown in figure 1 is run using the RADIOSS Starter and Engine. The sample base input files `boxbeam1_0000.rad` and `boxbeam1_0001.rad` can be found in `<hst.zip>/HS-4220/`and copied to your working directly.

The objective is to minimize the mass of the beam under the following two constraints: the internal energy must be more than 450, and the resulting reaction force must be less than 75. The input variables are the thicknesses of the four components defined in the input deck `boxbeam1._0000.rad` via the /PROP/SHELL entries. They are combined into two input variables. The thickness should be between 0.5 and 2.0; the initial thickness is 1.0. The optimization type is size.

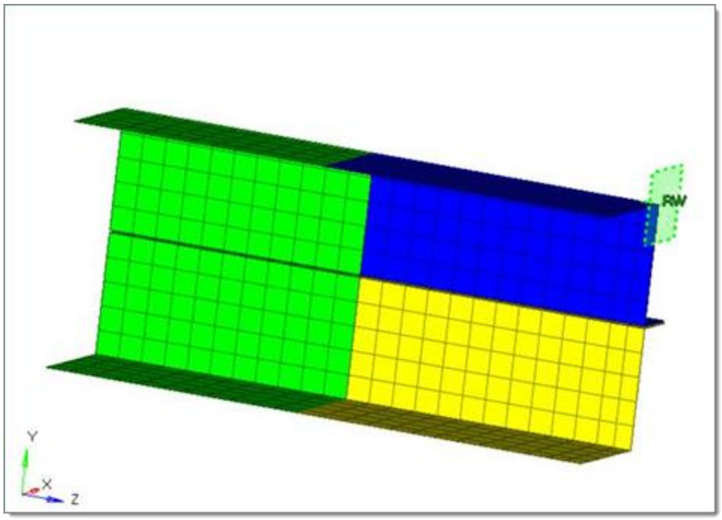


Figure 1. Boxbeam model, undeformed.

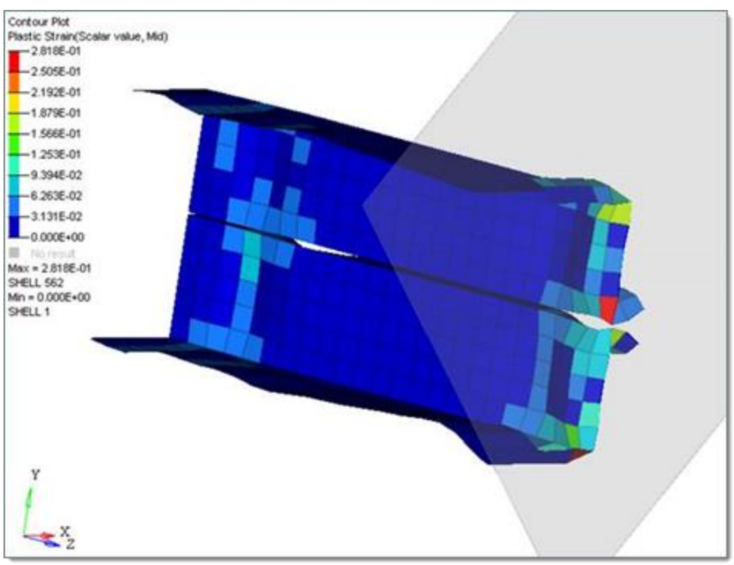
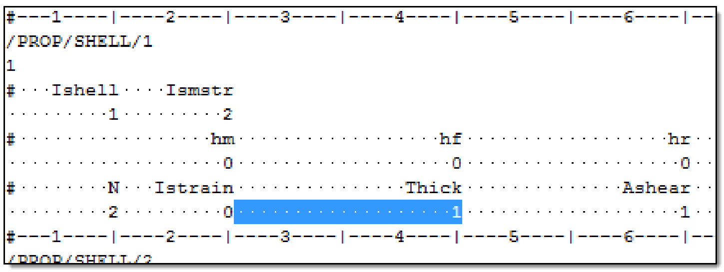


Figure 2. Boxbeam model, deformed, t = 2.001.

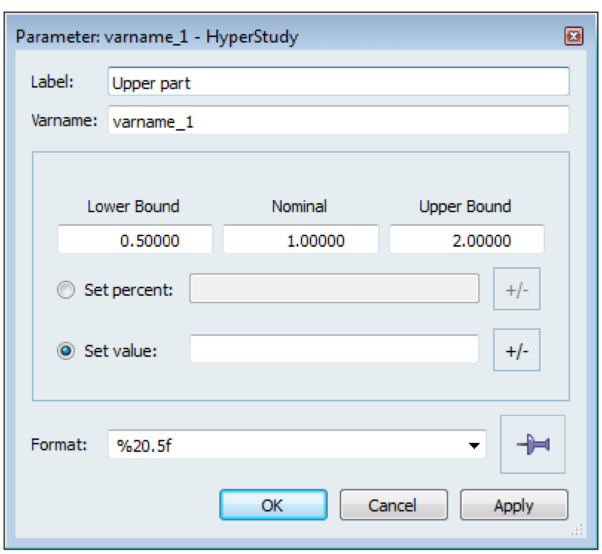
Step 1: Create the Base Input Template in HyperStudy

1. Start HyperStudy
2. From the menu bar, click ***Tools*** > ***Editor***. The **Editor** opens.
3. In the **File** field, navigate to your working directory and open the `boxbeam1_0000.rad` file.
4. In the **Find** area, enter `/PROP/SHELL/1`.
5. Click ▶ until you find /PROP/SHELL/1.
6. Highlight the field for thickness.

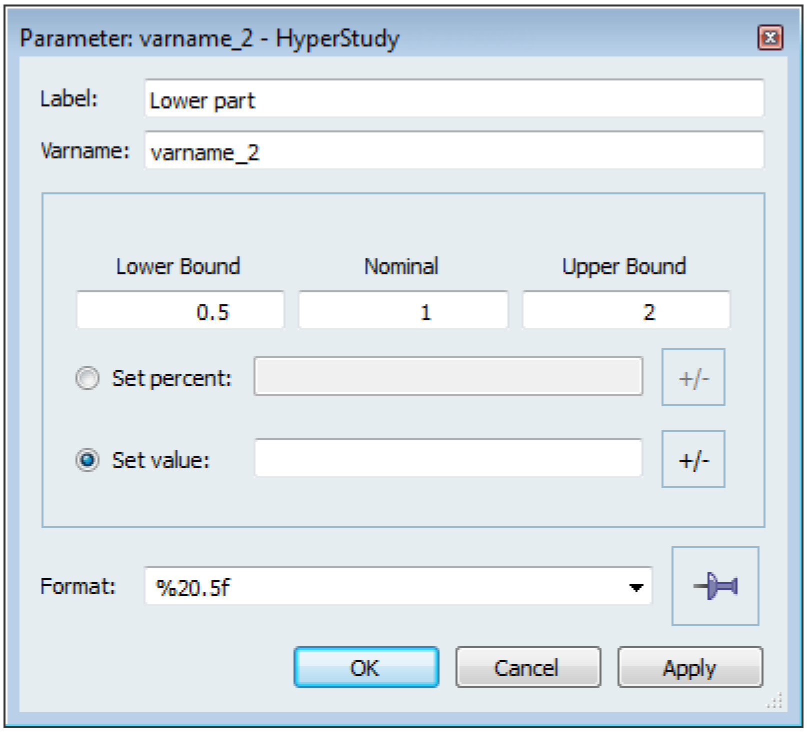
   Tip: To assist you in selecting 20-character fields, press ***CTRL*** to activate the **Selector** (set to 20 characters) and then click the value. HyperStudy highlights 20 fields.



7. Right-click on the highlighted fields and select ***Create Parameter*** from the context menu.
8. In the **Parameter - varname_1** dialog, **Label** field, enter `Upper part`.
9. Set the **Lower Bound** to `0.5`, the **Nominal** to `1.0`, and the **Upper Bound** to `2.0`.
10. Set the **Format** to %20.5f.
11. Click ***OK***.



12. Find **/PROP/SHELL/2** and highlight the field for thickness.
13. Assign it the same thickness as **/PROP/SHELL/1** by right-clicking on the highlighted fields and selecting **Attach to** > ***varname_1*** from the context menu.
14. Find **/PROP/SHELL/3** and highlight the field for thickness.
15. Right-click on the highlighted fields and select ***Create Parameter*** from the context menu.
16. In the **Parameter - varname_2** dialog, **Label** field, enter `Lower part`.
17. Set the **Lower Bound** to `0.5`, the **Nominal** to `1.0`, and the **Upper Bound** to `2.0`.
18. Set the **Format** to %20.5f.
19. Click ***OK***.



20. Find **/PROP/SHELL/4** and highlight the field for thickness.
21. Assign it the same thickness as **/PROP/SHELL/3** by right-clicking on the highlighted fields and selecting **Attach to** > ***varname_2*** from the context menu.
22. Click ***Save***.
23. In the **Save Template** dialog, navigate to your working directory and save the file as `boxbeam1.tpl`.
24. Close the **Editor**.

Step 2: Optional. View the Base Input Template in TextView

1. Start HyperGraph.
2. On the **Client Selector** toolbar, select ***TextView***.

3. From the menu bar, click ***File*** > ***Open*** > ***Document***.
4. In the **Open Document** dialog, open the `boxbeam1.tpl` file. The text editor displays the following input variables that are defined by Templex parameter statements:

```
{parameter(t1,"Upper part",1.0,0.5,2.0)}
{parameter(t2,"Lower part",1.0,0.5,2.0)}
```

5. On the **Text** toolbar, click .
6. In the **Find** dialog, **Find** field, enter `/PROP/SHELL`.
7. Click . The parameterized **/PROP/SHELL** cards, which reference the input variables, highlight.

```
/PROP/SHELL/1
1
#    Ishell    Ismstr
         1         2
#                  hm                  hf                  hr                  dm                  dn
                    0                   0                   0                   0                   0
#        N   Istrain               Thick              Ashear              Ithick     Iplas
         2         0{varname_1, %20.5f}                   1                   0         0
#---1----|----2----|----3----|----4----|----5----|----6----|----7----|----8----|----9----|---10----|
/PROP/SHELL/2
2
#    Ishell    Ismstr
         1         2
#                  hm                  hf                  hr                  dm                  dn
                    0                   0                   0                   0                   0
#        N   Istrain               Thick              Ashear              Ithick     Iplas
         2         0{varname_1, %20.5f}                   1                   0         0
#---1----|----2----|----3----|----4----|----5----|----6----|----7----|----8----|----9----|---10----|
/PROP/SHELL/3
3
#    Ishell    Ismstr
         1         2
#                  hm                  hf                  hr                  dm                  dn
                    0                   0                   0                   0                   0
#        N   Istrain               Thick              Ashear              Ithick     Iplas
         2         0{varname_2, %20.5f}                   1                   0         0
#---1----|----2----|----3----|----4----|----5----|----6----|----7----|----8----|----9----|---10----|
/PROP/SHELL/4
4
#    Ishell    Ismstr
         1         2
#                  hm                  hf                  hr                  dm                  dn
                    0                   0                   0                   0                   0
#        N   Istrain               Thick              Ashear              Ithick     Iplas
         2         0{varname_2, %20.5f}                   1                   0         0
#---1----|----2----|----3----|----4----|----5----|----6----|----7----|----8----|----9----|---10----|
```

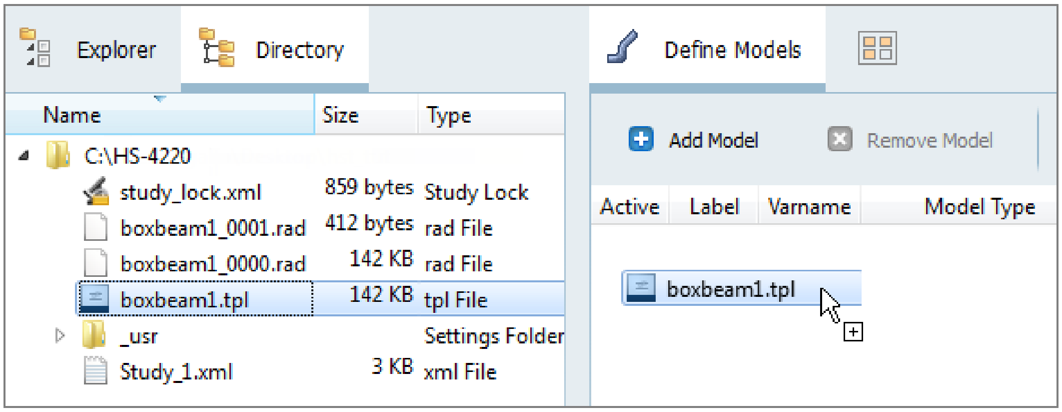
8. On the **Text** toolbar, click . The text editor evaluates the Templex statements, and replaces the parameters with their initial values.
9. Repeat steps 2.5 through 2.7, and search for **/PROP/SHELL** again. You will find:

```
/PROP/SHELL/1
1
#   Ishell    Ismstr
         1         2
#               hm                  hf                  hr                  dm                  dn
                 0                   0                   0                   0                   0
#        N   Istrain             Thick              Ashear              Ithick     Iplas
         2         0           1.00000                   1                   0         0
#---1----|----2----|----3----|----4----|----5----|----6----|----7----|----8----|----9----|---10----|
/PROP/SHELL/2
2
#   Ishell    Ismstr
         1         2
#               hm                  hf                  hr                  dm                  dn
                 0                   0                   0                   0                   0
#        N   Istrain             Thick              Ashear              Ithick     Iplas
         2         0           1.00000                   1                   0         0
#---1----|----2----|----3----|----4----|----5----|----6----|----7----|----8----|----9----|---10----|
/PROP/SHELL/3
3
#   Ishell    Ismstr
         1         2
#               hm                  hf                  hr                  dm                  dn
                 0                   0                   0                   0                   0
#        N   Istrain             Thick              Ashear              Ithick     Iplas
         2         0           1.00000                   1                   0         0
#---1----|----2----|----3----|----4----|----5----|----6----|----7----|----8----|----9----|---10----|
/PROP/SHELL/4
4
#   Ishell    Ismstr
         1         2
#               hm                  hf                  hr                  dm                  dn
                 0                   0                   0                   0                   0
#        N   Istrain             Thick              Ashear              Ithick     Iplas
         2         0           1.00000                   1                   0         0
#---1----|----2----|----3----|----4----|----5----|----6----|----7----|----8----|----9----|---10----|
```

10. Close HyperGraph; you do not need to save the session.

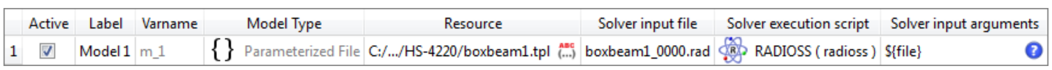
Step 3: Perform the Study Setup

1. Return to 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 `boxbeam1.tpl` file into the work area.



b. In the **Solver input file** column, enter `boxbeam1_0000.rad`. This is the name of the solver input file HyperStudy writes during any evaluation.

c. In the **Solver execution script** column, select ***RADIOSS (radioss)***.



6. Define a model dependency.

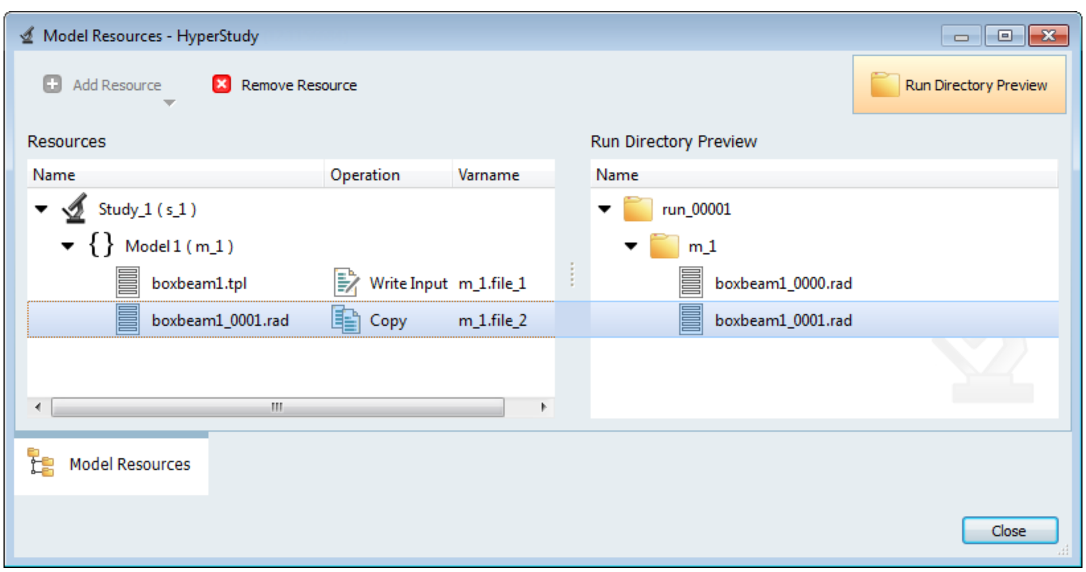
a. Click ***Model Resources***.

b. In the **Model Resource** dialog, click ***Add Resource*** > ***Add Input Resource***.

c. In the **Select File** dialog, navigate to your working directory and open the `boxbeam1_0001.rad` file.

d. Set **Operation** to ***Copy***.

e. Click ***Close***.



7. Click ***Import Variables***. Two input variables are imported from the `boxbeam1.tpl` resource file.

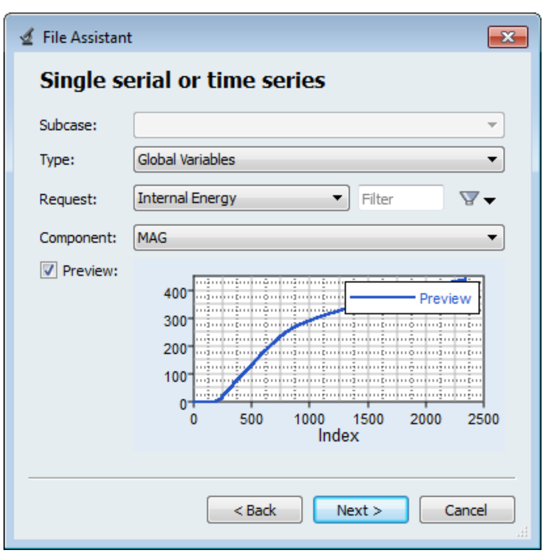
8. Go to the **Define Input Variables** step.
9. Review the input variable's lower and upper bound ranges.
10. Go to the **Specifications** step.

Step 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***. An `approaches/nom_1/` directory is created inside the study directory. The `approaches/nom_1/run__00001/m_1` directory contains the result files.
5. Go to the **Define Output Responses** step.

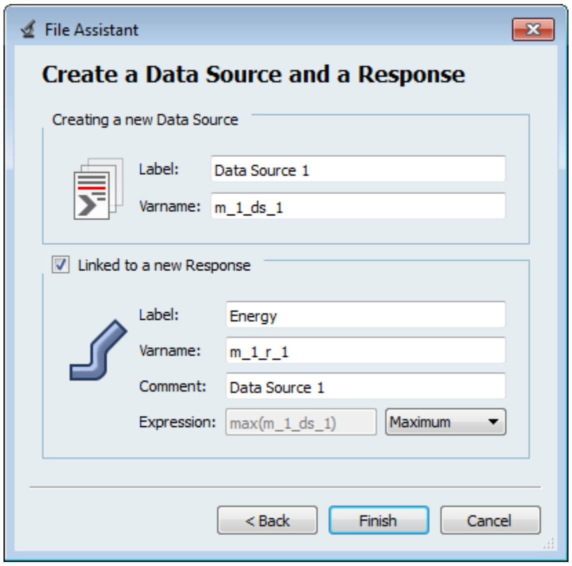
Step 5: Create and Define Output Responses

1. Create the Energy output response, which is the internal energy of the model.
   a. From the **Directory**, drag-and-drop the `boxbeam1T01` 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® (hgradioss++.exe)*** 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 ***Global Variables***.
      - Set **Request** to ***Internal Energy***.
      - Set **Component** to ***MAG***.



e. Label the output response `Energy`.

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



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

2. Create the Force output response, which is the resultant reaction force in the Z-direction.

   a. From the **Directory**, drag-and-drop the `boxbeam1T01` 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® (hgradioss++.exe)*** 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 ***Rigid wall/Wall Force***.
      - Set **Request** to ***1 RWALL 1***.
      - Set **Component** to ***FNZ-Z NORMAL FORCE***.

   e. Label the output response `Force`.

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

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

3. Create the Mass output response.

   a. From the **Directory**, drag-and-drop the `boxbeam1T01` 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® (hgradioss++.exe)*** 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 ***Global Variables***.
- Set **Request** to ***Mass***.
- Set **Component** to ***MAG***.

e. Label the output response `Mass`.

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

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

4. Click ***Evaluate*** to extract the output response values.

	Active	Label	Varname	Expression	Value	Comment
1	☑	Energy	m_1_r_1	max(m_1_ds_1) ...	438.58371	Data Source 1 ...
2	☑	Force	m_1_r_2	max(m_1_ds_2) ...	89.973709	Data Source 2 ...
3	☑	Mass	m_1_r_3	m_1_ds_3[0] ...	25.446430	Data Source 3 ...

Step 6: 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 lower and upper bound ranges of the input variables.
5. Go to the **Select Output Responses** step.
6. Apply an objective to the Mass output response.

   a. In the **Objectives** column for Mass, click ⊕.

   b. In the pop-up window, set **Type** to ***Minimize*** and click ***OK***.

	Active	Label	Varname	Objectives	Constraints	Evaluate From
1	☑	Energy	m_1_r_1	⊕	⊕	Solver
2	☑	Force	m_1_r_2	⊕	⊕	Solver
3	☑	Mass	m_1_r_3	Minimize ...	⊕	Solver

7. Apply a constraint to the Energy output responses.

   a. In the **Constraints** column for Energy, click ⊕.

   b. In the pop-up window, define the following and click ***OK***.

   - Set **Type** to ***Deterministic***.
   - Set **Bound Type** to **>=**.
   - For **Bound Value**, enter `450`.

8. Apply a constraint to the Force output responses.

a. In the **Constraints** column for Energy, click .

b. In the pop-up window, define the following and click ***OK***.

- Set **Type** to ***Deterministic***.
- Set **Bound Type** to **<=**.
- For **Bound Value**, enter `75`.

	Active	Label	Varname	Objectives	Constraints	Evaluate From
1	☑	Energy	m_1_r_1		>= 450.00000 ...	Solver
2	☑	Force	m_1_r_2		<= 75.000000 ...	Solver
3	☑	Mass	m_1_r_3	Minimize ...		Solver

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*** to launch the Optimization.

Step 7: View the Iteration History of an Optimization Study

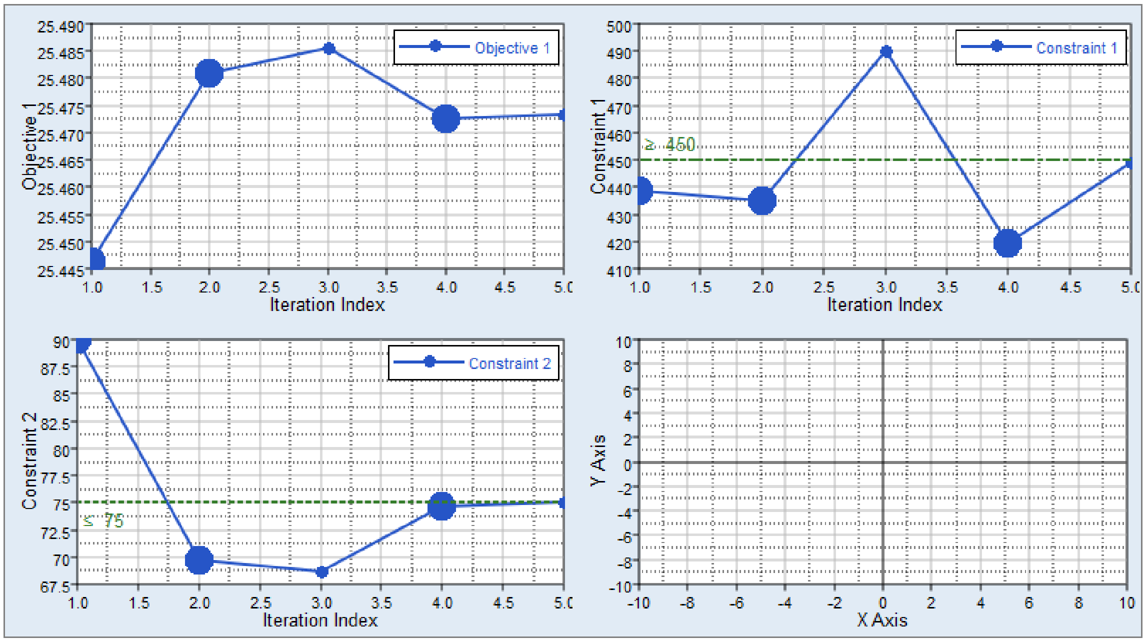
1. Click the ***Iteration History*** tab to display data in a tabluar view. The optimal design is highlighted green, the infeasible designs are shown with red text, and the violated constraints are indicated in bold text.

	Upper part	Lower part	Energy	Force	Mass	Objective 1	Constraint 1	Constraint 2	Iteration Index	Evaluation
1	1.0000000	1.0000000	438.58371	89.973709	25.446430	25.446430	**438.58371**	**89.973709**	1	1
2	1.1650000	1.0000000	434.71823	69.646622	25.480909	25.480909	**434.71823**	69.646622	2	2
3	1.0000000	1.1650000	490.14233	68.711662	25.485611	25.485611	490.14233	68.711662	3	3
4	1.0772704	1.0423280	419.19440	74.710403	25.472628	25.472628	**419.19440**	74.710403	4	4
5	1.0256669	1.0904766	449.24582	75.045273	25.473280	25.473280	449.24582	75.045273	5	5

2. Click the ***Iteration Plot*** tab to plot the iteration history of the study's objectives, constraints, and input variables.

   Using the **Channel** selector, select ***Objective 1***, ***Constraint 1***, and ***Constraint 2***.

   In the initial design, the design was infeasible as indicated by the large circular marker for the first iteration. A view of the constraint plots shows that the second constraint was violated in the initial design. Initially, the optimizer added some weight in order to satisfy the design constraints. Notice that both constraints are near their bounds in the optimal design.



Last modified: v2017.2 (12.1156684)