Altair

HyperWorks

Altair HyperStudy 2019 Tutorials

HS-4420: Optimization Study of a Spherical Impactor

HS-4420: Optimization Study of a Spherical Impactor

This tutorial demonstrates how to perform an advanced study that has both size and shape input variables on a RADIOSS finite element model.

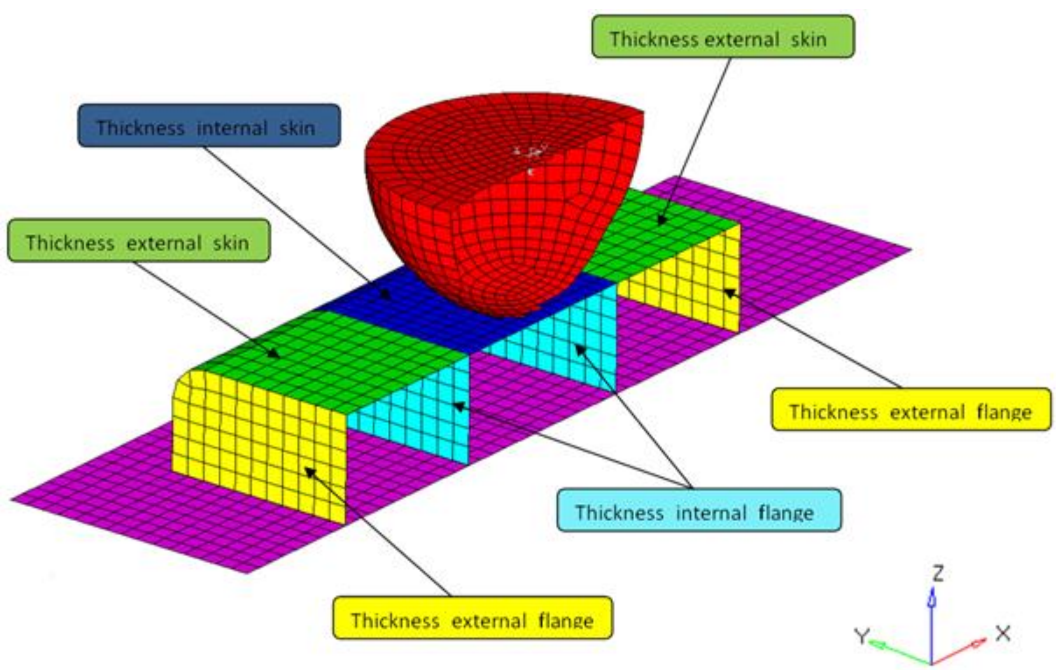
The sample base input template can be found in `<hst.zip>/HS-4420/`. Copy the files `impactor.hm`, `impactor_0000.rad`, and `impactor_0001.rad` from this directory to your working directory.

The steps taken in this tutorial demonstrate how to analyze the input variables in order to identify the most important variables and how to do an Optimization. The objective of the Optimization is to minimize the maximum acceleration of the impactor, while keeping maximum displacement lower than 16 mm.

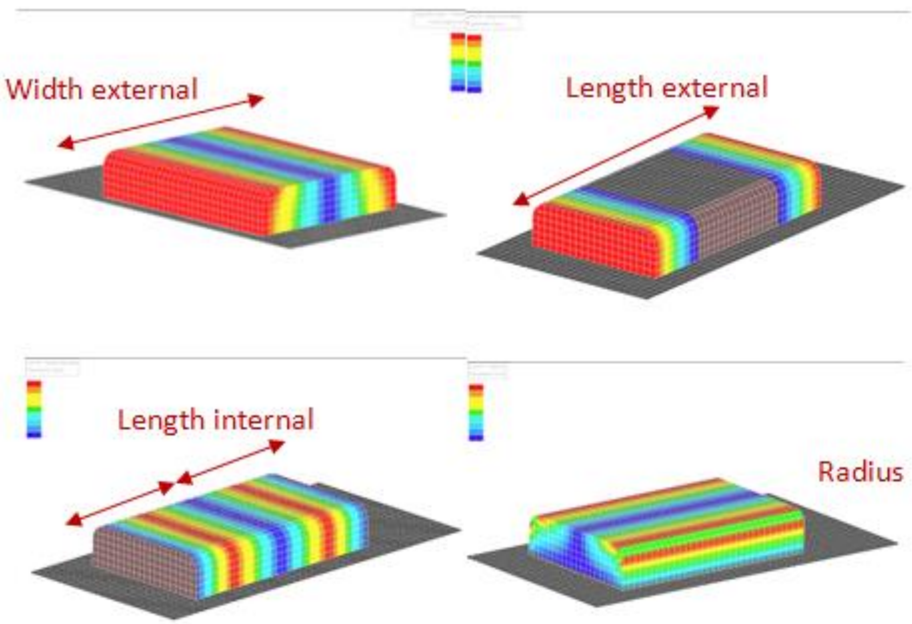
In this tutorial, you will:

- Create a base input template from a RADIOSS input file using the HyperStudy Editor
- Set up a study
- Run a DOE study (screening DOE)
- Post process DOE results in order to define the most important variables and reduce the number of variables (screening)
- Create a new DOE in order to create an approximation
- Create an approximation
- Run an Optimization study based on the approximation created

This model simulates the dynamic impact of a sphere with an initial velocity on a box. There are eight variables: four size variables, which are four box thickness, and four shapes variables.



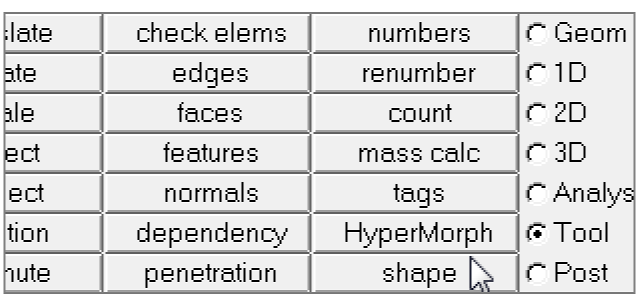
Size variables



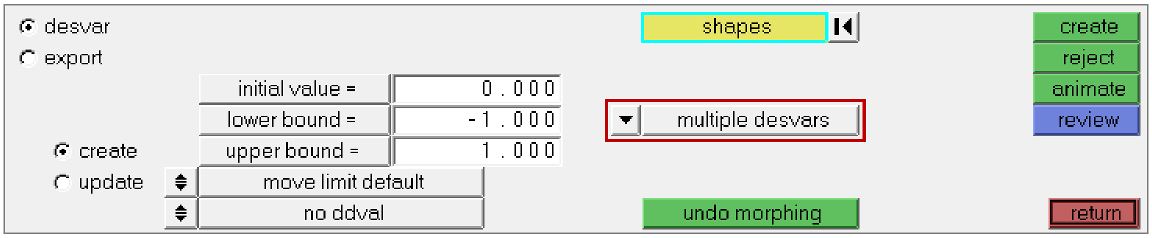
Shapes variables

Step 1: Export the Shape Parameterization from HyperMesh

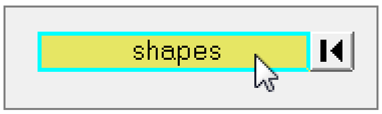
1. Start HyperMesh Desktop.
2. In the **User Profiles** dialog, change the user profile to ***RADIOSS***.
3. From the menu bar, click ***File*** > ***Open*** > ***Model***.
4. In the **Open Model** dialog, open the `impactor.hm` file. The `impactor.hm` database has the RADIOSS analysis setup, and the shapes have already been created. You must export the shapes variables so that they are included in the template file.
5. From the **Tool** page, click ***Shape***.



6. Go to the ***desvar*** subpanel.
7. Switch **single desvars** to ***multiple desvars***.

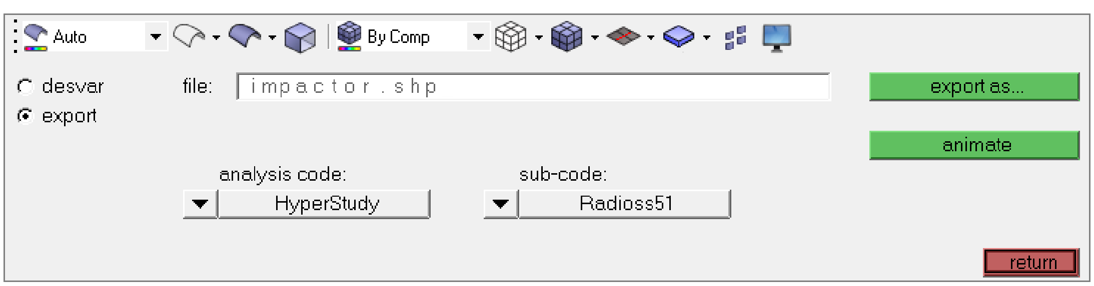


8. In the **initial value=** field, enter 0.
9. In the **lower bound=** field, enter -1.
10. In the **upper bound=** field, enter 1.
11. Click the ***shapes*** selector.



12. Select all of the shapes.
13. Click ***select***.
14. Click ***create***. A shape design variable is created for each shape.
15. Optional.
    - If you would like to animate or visualize the shapes, click ***animate***.
    - In the **Deformed** panel, click ***linear*** or ***modal*** to animate the shape variables in the graphics area.
    - While the shape is animating, you can adjust the animation speed by moving the slider as indicated in the image below.

16. Go to the ***export*** subpanel.
17. For **analysis code**, select ***HyperStudy***.
18. For **sub-code**, select ***Radioss51***.
19. In the **File** field, enter `impactor.shp`.



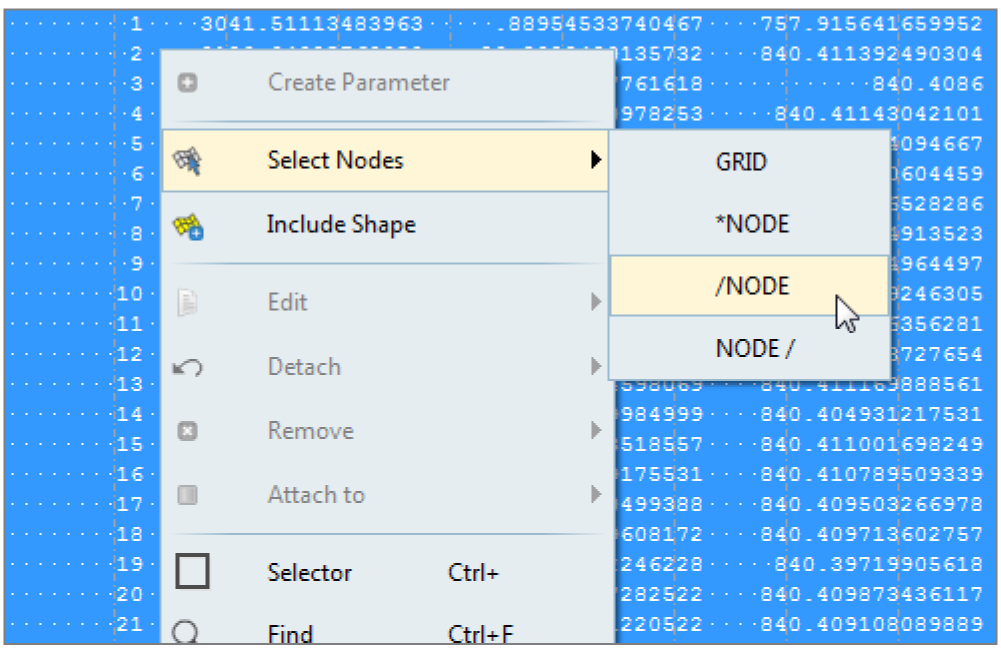
20. Click ***export as***.
21. In the **Save As** dialog, navigate to your working directory and save the file as `impactor.shp`. HyperMesh writes the following files:

`impactor.radioss51.node.tpl`	Grid coordinates template.
`impactor.shp`	Grid perturbation vector data read by impactor.radioss51.node.tpl.

22. Exit HyperMesh Desktop.

Step 2: Create the Base Input Template in HyperStudy

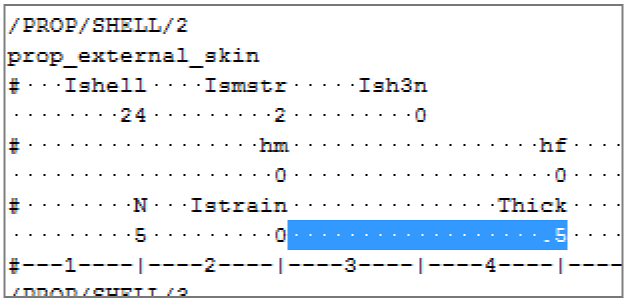
1. Start HyperStudy.
2. From the menu bar, click ***Tools*** > ***Editor***.
3. In the **Editor** dialog, **File** field, open the `impactor_0000.rad` file.
4. Right-click anywhere in the editor and select ***Select Nodes*** > ***/NODE*** from the context menu. All of the /NODE cards in the `impactor_0000.rad` file highlight.



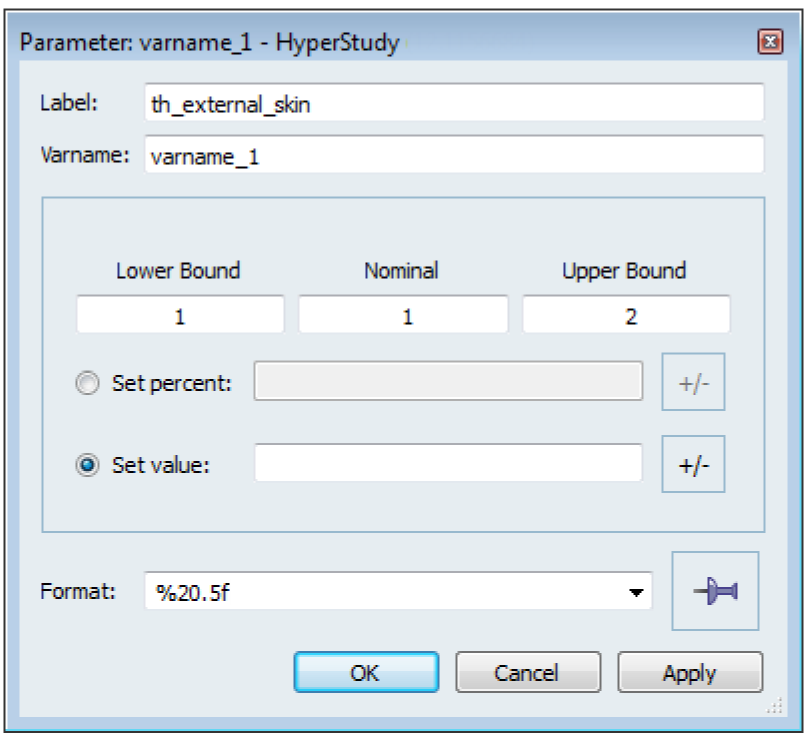
5. Right-click on the highlighted cards and select ***Include Shape*** from the context menu.
6. In the **Shape Template** dialog, open the `impactor.radioss51.node.tpl` file. The shape variables are now created and the grid has been replaced by the parameter file (which contains the grid parameterized by the shapes) exported during step 1.
7. Locate the shape variable prop_external_skin.
8. Select the thickness value for prop_external_skin, as indicated in the image below. In a

RADIOSS deck, each field within a card is 20 characters long.

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.



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



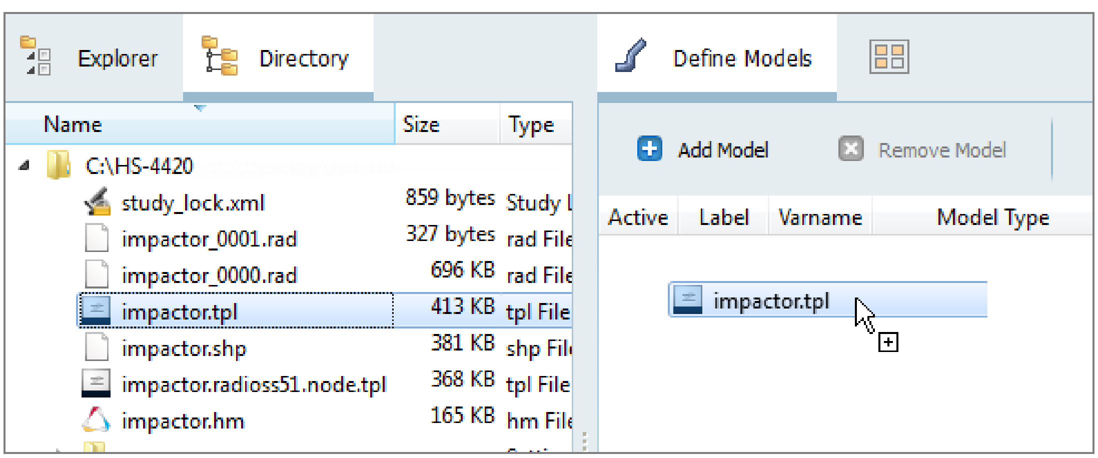
14. Define three more input variables for thickness using the information provided in the table below.

Input Variable	Label	Lower Bound	Nominal Value	Upper Bound	Format
prop_internal_skin	th_internal_skin	1.0	1.0	2.0	%20.5f
prop_external_flange	th_external_flange	1.0	1.0	2.0	%20.5f
prop_internal_flange	th_internal_flange	1.0	1.0	2.0	%20.5f

15. Click ***Save***.
16. In the **Save Template** dialog, navigate to your working directory and save the file as `impactor.tpl`.
17. Close the **Editor**.

Step 3: Perform the Study Setup

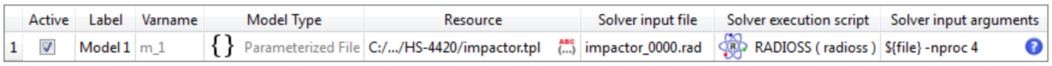
1. To start a new study, click ***File*** > ***New*** from the menu bar, or click on the toolbar.
2. In the **HyperStudy – Add** dialog, enter a study name, select a location for the study, and click ***OK***.
3. Go to the **Define models** step.
4. Add a Parameterized File model.
   a. From the **Directory**, drag-and-drop the `impactor.tpl` file into the work area.



   b. In the **Solver input file** column, enter `impactor_0000.rad`. This is the name of the starter input file HyperStudy creates from the parameterization, and the name of the Engine file.

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

d. In the **Solver input arguments** column, enter `-nproc 4` after `${file}`.



5. Define a model dependency.

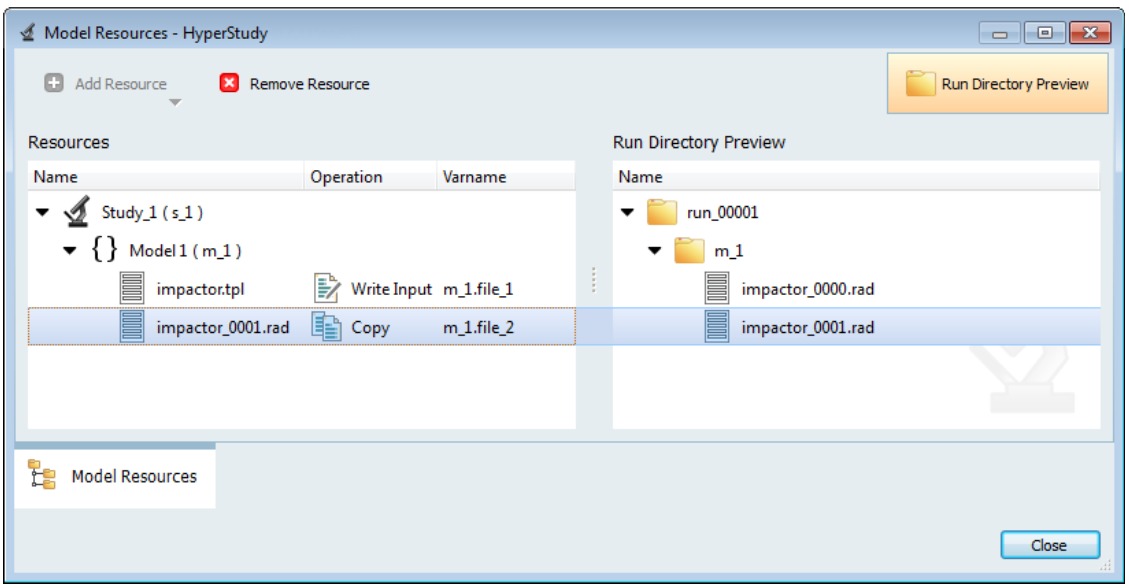
   a. Click ***Model Resources***.

   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 `impactor_0001.rad` file.

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

   e. Click ***Close***.



6. Click ***Import Variables***. Eight input variables are imported from the `impactor.tpl` resource file.
7. Go to the **Define Input Variables** step.
8. Review the lower and upper bound ranges of the input variables.
9. Go to the **Specifications** step.

Step 4: Perform the Nominal Run

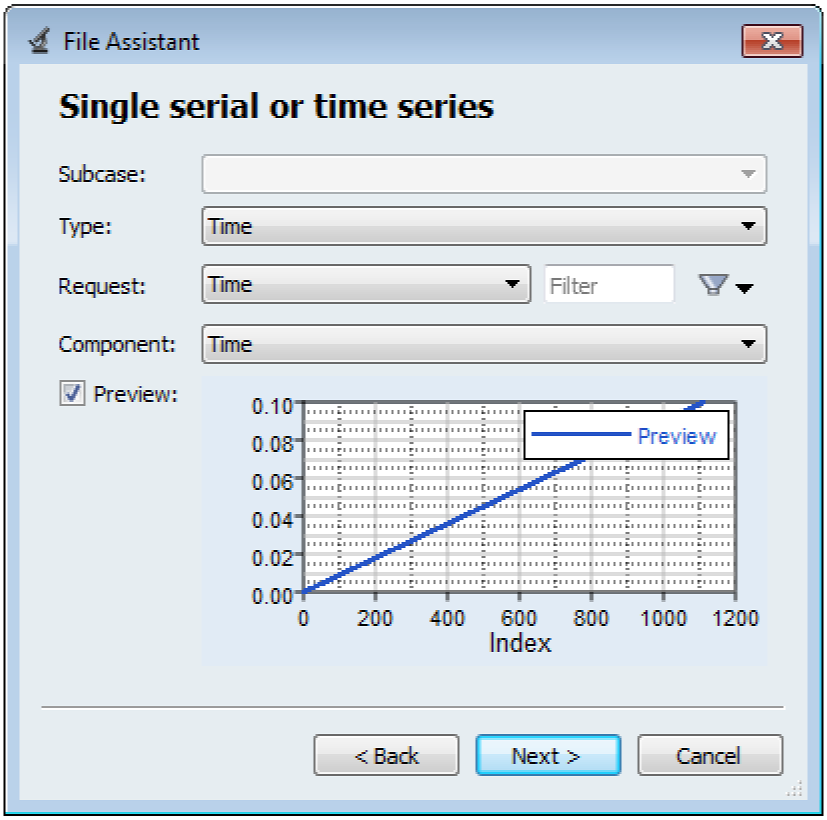
1. In the work area, set the **Mode** to ***Nominal Run***.
2. Click ***Apply***.

3. Go to the **Evaluate** step.
4. Click ***Evaluate Tasks***. An `approach/nom_1/` directory is created inside the study directory. The the `approaches/nom_1/run__00001/m_1` sub-directory contains the `impactorT01` file, which is the result of the nominal run.
5. Go to the **Define Output Responses** step.

Step 5: Create and Define Output Responses

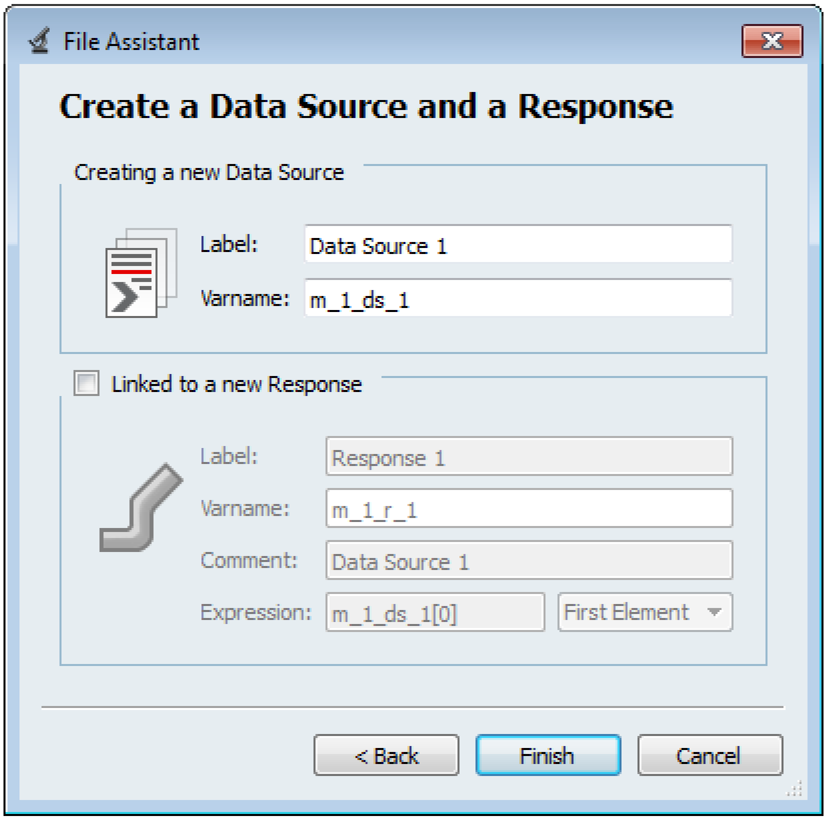
In this study, we want to analyze the maximum acceleration and the maximum displacement observed by the box. This study is a function of the time; we need to extract the maximum of each output response vector over time.

1. Create a file source for time.
    a. From the **Directory**, drag-and-drop the `impactorT01` 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, then click ***Next***.
    - Set **Type** to ***Time***.
    - Set **Request** to ***Time***.
    - Set **Component** to ***Time***.



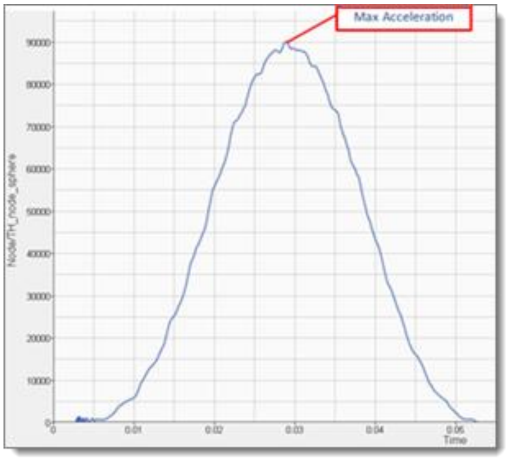
e. Clear the **Linked to a new Response** checkbox.

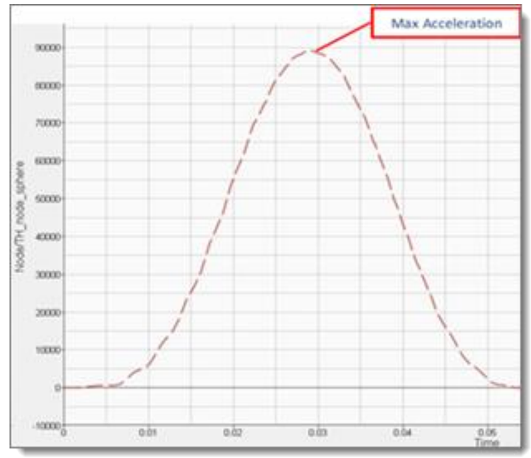
f. Click ***Finish***



2. Create a second file source for impactor acceleration along the Z axis. Repeat step 1, except define the following options:
    - Set **Type** to ***Node/TH_node_sphere***.
    - Set **Request** to ***4206 rigid_sphere_4206***.
    - Set **Component** to ***AZ-Z Acceleration***.
3. Create a third file source for impactor displacement along the Z axis. Repeat step 1, except define the following options:
    - Set **Type** to ***Node/TH_node_sphere***.
    - Set **Request** to ***4206 rigid_sphere_4206***.
    - Set **Component** to ***DZ-Z Displacement***.

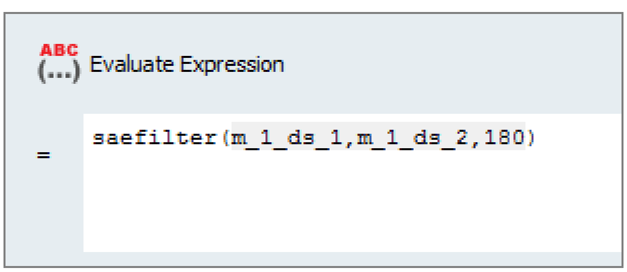
You have finished creating all of the result vectors for the Max_Acceleration output response. As you can see from the graph on the left-hand side below, it has some noise. To eliminate the noise, you will use a filter and work on the filtered output response as seen from the graph on the right-hand side below.





4. Add two output responses.
    a. Click ***Add Output Response*** twice.
    b. In the work area, change the labels for the output responses to Max_Acceleration and Max_Displacement.
5. Define the Max_Acceleration output response.
    a. In the **Expression** column of the output response Max_Acceleration, click **...**.
    b. In the **Expression Builder**, click the ***Functions*** tab.
    c. From the list of functions, select ***saefilter***. This function will apply a filter to the acceleration vector.
    d. Click ***Insert Varname***. The function `saefilter(,,)` appears in the ***Evaluate Expression*** field.

        You can now add the time vector and the acceleration vector as arguments to the function, with a class parameter of 180.
    e. In the **Evaluate Expression** field, enter `(m_1_ds_1,m_1_ds_2,180)` in the **saefilter** function.



    f. To calculate the max of the expression, add the max function to the beginning of the expression.

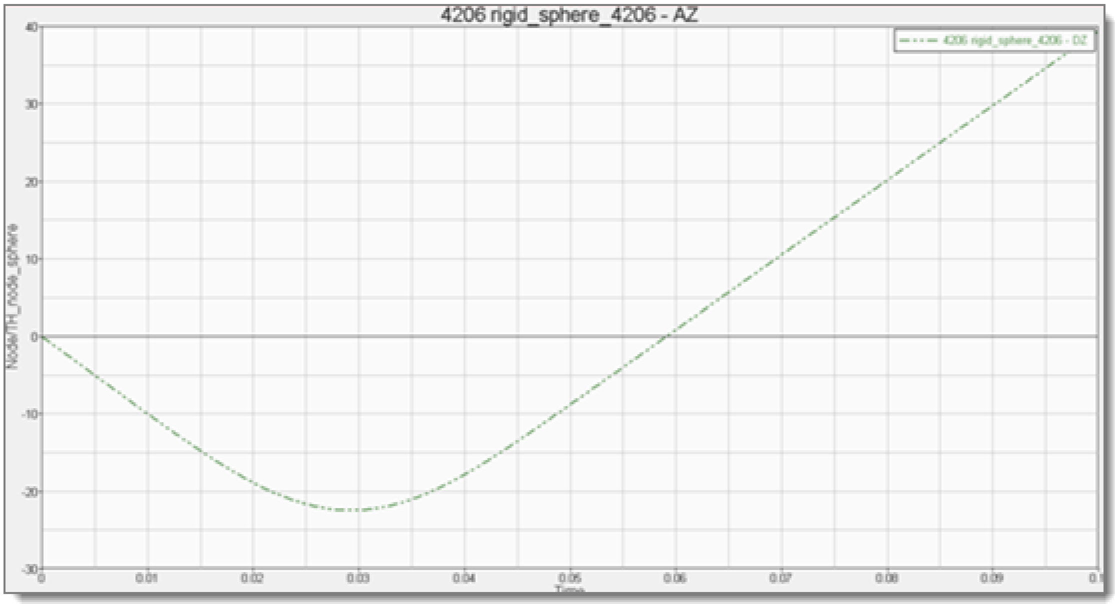
        The expression should read: `max(saefilter(m_1_ds_1,m_1_ds_2,180))`.
    g. To express the result in G, divide the max of the expression by 9810.

        The expression should read: `max(saefilter(m_1_ds_1,m_1_ds_2,180)/9810)`.

   h. Click ***OK***.

6. Define the Max_Displacement output response.

   a. Optional. Plot the displacement with respect to the time, to obtain the curve illustrated below:



   b. In the **Expression** column of the output response Max_Displacement, click **…**.

   c. In the **Expression Builder**, click the ***Functions*** tab.

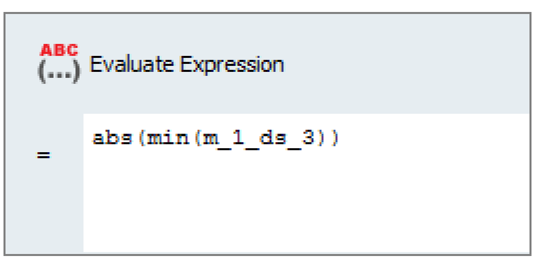
   d. From the list of functions, select ***abs***.

   e. Click ***Insert Varname***. The function `abs()` appears in the ***Evaluate Expression*** field.

   f. From the list of functions, select ***min***.

   g. Click ***Insert Varname***. The expression should now read, `abs(min())`.

   h. In the **Evaluate Expression** field, enter `m_1_ds_3` in the min function.



   i. Click ***OK***.

7. Click ***Evaluate*** to extract the output response values of each expression.

	Active	Label	Varname	Expression	Value
1	☑	Max_Acceleration	r_1	max(saefilter(m_1_ds_1,m_1_ds_2,180)/9810) ...	9.0879590
2	☑	Max_Displacement	r_2	abs(min(m_1_ds_3)) ...	22.425158

Step 6: Run a Screening DOE Study

The model has 8 variables which may lead to high computation times for direct optimization or even for creating a response surface.

You will reduce the number of actual input variables by running a screening experiment. A full factorial experiment with 8 factors at 2 levels will require 28 (256) runs and with 3 levels, it will increase to 6561 runs. You will try to screen out some input variables by first doing a Fractional Factorial screening 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 ***Fractional Factorial***.
5. In the **Settings** tab, set **Resolution** to ***IV***.

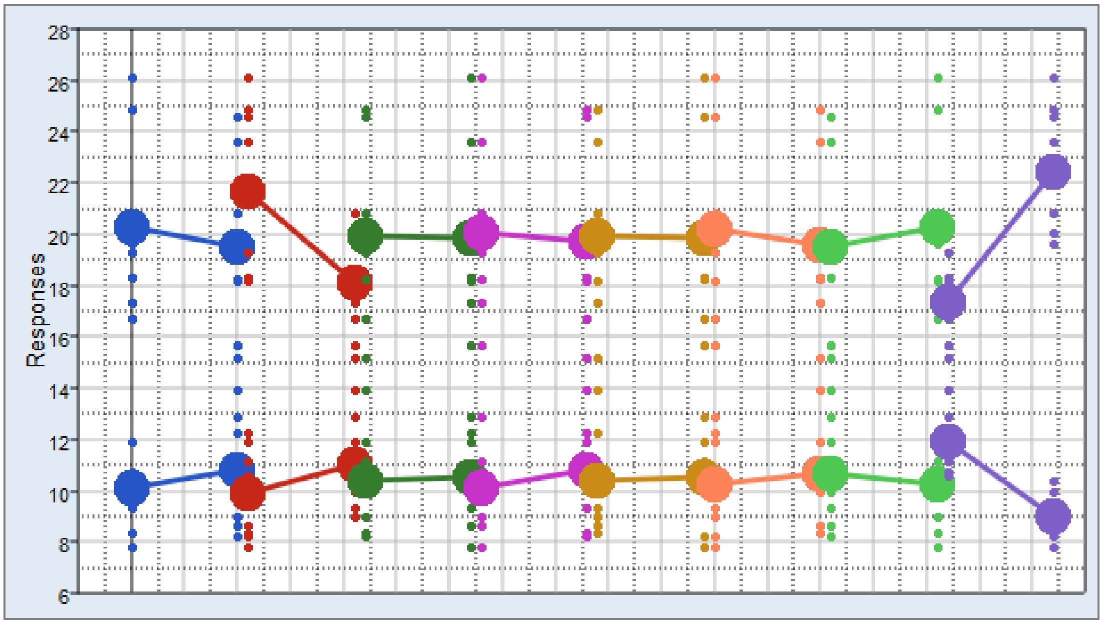
   Note: Resolution IV enables an estimate of main effects unconfounded by two-factor interactions. It also enables an estimate of two-factor interaction effects, which may be confounded with other two-factor interactions.

	Value
Resolution	IV
Number of runs	16
Use Inclusion Matrix	☐

6. Verify that the **Number of runs** is set to 16.
7. Click ***Apply***.
8. Go to the **Evaluate** step.
9. Click ***Evaluate Tasks*** to execute the run matrix and extract the output responses for all of the runs. HyperStudy runs 16 simulations in Fractional Factorial mode, therefore the evaluation will take some time.
10. Go to the **Post-Processing** step.

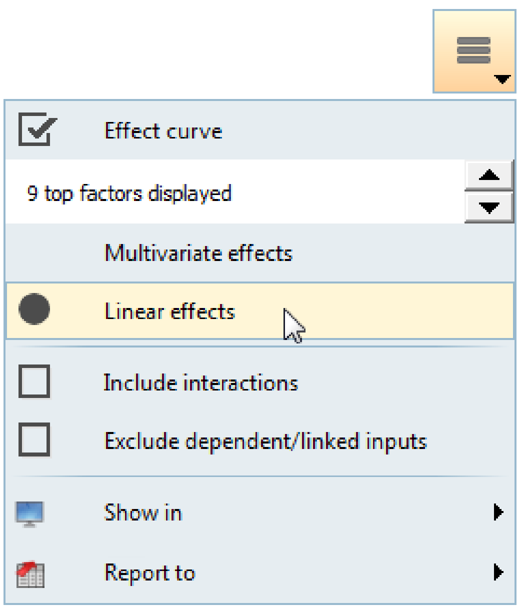
Step 7: Post Process the Screening DOE Study

1. Click the ***Linear Effects*** tab to review the linear effects. Observe the main effect of the input variables on both output responses.



2. Click the ***Pareto Plot*** tab, then use the **Channel** selector to select both of the output responses. Observe that results.

   Note: A linear effects plot and a pareto plot with the **Linear effects** option enabled (shown below) provide the same information. However, with a pareto plot, you can use a statistical measure (that is, the 80-20 rule) to decide which input variables are more significant and which input variables can be neglected.



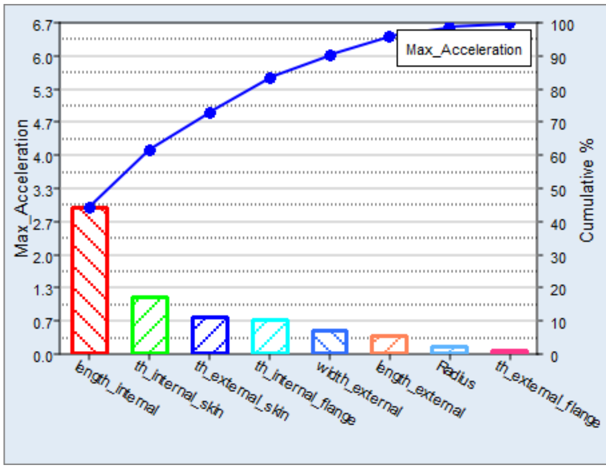
For this tutorial, you will use the 80/20 rule to eliminate input variables that are not significant to the study. The 80/20 rule is a Pareto principle that proposes 80% of the total effects comes from only 20% of the variables.

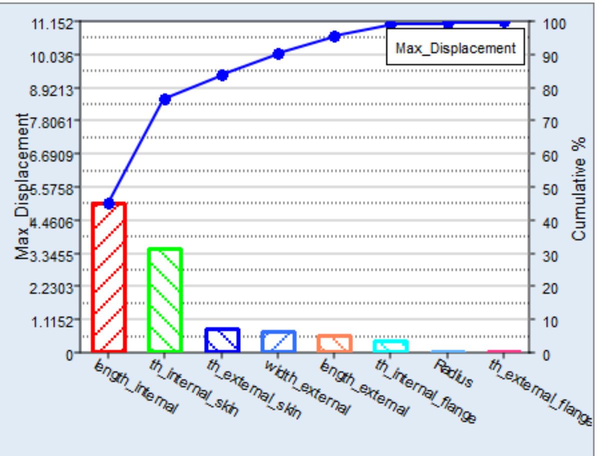
Note: You should also use other practices to eliminate input variables that you feel should be taken in consideration.

For screening purpose, you can see which input variables contribute to 80% or more of the given output response. In the images below you can see that:

- For Max_Acceleration, the input variables length_internal, th_internal_skin, and th_external_skin contribute to 80% of the linear effect.
- For Max_Displacement, the input variables length_internal and th_internal_skin fall under the 80/20 rule.

For n responses, you can list out the input variables that follow the 80/20 rule, and take union of the sets. In this case, the input variables that follow the 80/20 rule include: length_internal, th_internal_skin, and th_external_skin. This narrows your list to three significant input variables.





Step 8: Run a DOE Study for Approximation

Since this optimization is based on response surfaces, a central composite experiment will be used, which will create a 2nd order response surface.

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 **Select Input Variables** step.
4. In the **Active** column, keep only the three significant input variables (established in step 7) selected. Clear the corresponding checkboxes for all other input variables.

	Active	Label	Varname	Lowe
1	☑	th_external_skin	m_1_varname_1	1.00000
2	☑	th_internal_skin	m_1_varname_2	1.00000
3	☐	th_external_flange	m_1_varname_3	1.00000
4	☐	th_internal_flange	m_1_varname_4	1.00000
5	☐	Radius	m_1_Radius	-1.0000
6	☐	length_external	m_1_length_external	-1.0000
7	☐	width_external	m_1_width_external	-1.0000
8	☑	length_internal	m_1_length_internal	-1.0000

5. Go to the **Specifications** step.
6. In the work area, set the **Mode** to ***Central Composite***.
7. Click ***Apply***.
8. Go to the **Evaluate** step.
9. Click ***Evaluate Tasks*** to execute the run matrix and to extract the output responses.

Step 9: Run a DOE Study for the Validation Matrix

Other points will be used to check the quality of the approximation. The points will be defined by a new DOE. In this DOE study, a Latin Hypercube of 10 runs will be used.

1. Add a third Doe to the study by repeating **Step 8: Run a DOE Study for Approximation**.
   a. In the **Specifications** step, set the **Mode** to ***Latin HyperCube***.
   b. In the **Settings** tab, change the **Number of runs** to `10`.

Step 10: Create an Approximation

1. In the **Explorer**, right-click and select ***Add*** from the context menu.
2. In the **Add - HyperStudy** dialog, select ***Fit*** and click ***OK***.
3. Go to the **Select Matrices** step.
4. Click ***Add Matrix*** twice.
5. Define **FitMatrix1** and **FitMatrix2** by selecting the options indicated in the image below.

	Active	Label	Varname	Type	Matrix Source	Matrix Origin	Status
1	☑	FitMatrix 1	fitmatrix_1	Input	DOE 2 (doe_2)	DoeDOE 2	Import Pending
2	☑	FitMatrix 2	fitmatrix_2	Validation	DOE 3 (doe_3)	DoeDOE 3	Import Pending

6. Click ***Import Matrix***.
7. Go to the **Select Input Variables** step.
8. Review the input variables and output responses. Only the length_internal, th_internal_skin, and th_external_skin input variables should be active.
9. Go to the **Specifications** step.
10. In the work area, set the **Mode** to ***Moving Least Squares (MLSM)***.
11. In the **Settings** tab, verify that **Regression Model** is set to ***Linear*** (default).

    Note: It is advisable to start with lowest order and increment it in case model Residuals and Diagnostics do not look feasible.
12. Keep all other parameters to default as well.
13. Click ***Apply***.
14. Go to the **Evaluate** step.
15. Click ***Evaluate Tasks***.
16. Go to the **Post-Processing** step.
17. To assess the accuracy of the regression equations, click the ***Residuals*** and ***Diagnostics*** tab.
18. To review the output response curves and surfaces, click the ***Trade-Off*** tabs.

    In the **Trade-Off 3D** tab, use the **Channel** selector to plot input variables and output responses. The values for the input variables which are not plotted are modified in the top frame (**Inputs**). Move the sliders in the **Value** column to modify the other input variables, while studying the output response throughout the design space.

Step 11: 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. In the **Active** column, clear the checkboxes for all input variables except length_internal, th_internal_skin and th_external_skin.
5. Go to the **Select Output Responses** step.
6. Apply an objective on the Max Acceleration output response.
    a. In the **Objectives** column for Max Acceleration, click .
    b. In the pop-up window, set **Type** to ***Minimize*** and click ***OK***.
    c. Set **Evaluate From** to ***Max_Acceleration__MLSM (r_1_fit_1)***.

	Active	Label	Varname	Type	Apply On	Evaluate From	Target Value
1	☑	Objective 1	obj_1	Minimize	Max_Acceleration (r_1)	Max_Acceleration_MLSM (r_1_fit_1)	1.0000000

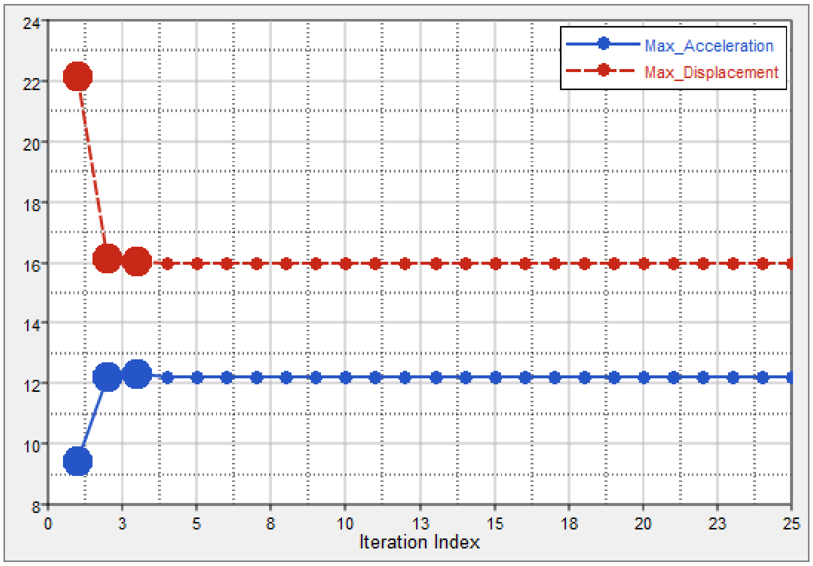
7. Apply a constraint on the Max Displacement output response.
   a. In the **Constraints** column for Max Displacement, click .
   b. In the pop-up window, *define the following* and click ***OK***.
      - Set **Type** to ***Deterministic***.
      - Set **Bound Type** to **<=** (less than or equal to).
      - For **Bound Value**, enter `16`.
   c. Set **Evaluate From** to Max_Displacement__MLSM (r_2_fit_1).

	Active	Label	Varname	Type	Apply On	Bound Type	Bound Value	Evaluate From
1	☑	Constraint 1	c_1	Deterministic	Max_Displacement (r_2)	<=	16.000000	Max_Displacement__MLSM (r_2_fit_1)

8. Click ***Apply***.
9. Go to the **Specifications** step.
10. In the work area, set the **Mode** to ***Genetic Algorithm (GA)***.

    Note: Only the methods that are valid for the problem formulation are enabled.
11. In the **Settings** tab, change the **Constraint violation tol**. to `0.0`.

    Note: Optimization algorithms are usually solved so that a small violation in the constraint is acceptable in order to save time in resolving the variables' precision beyond a physical significance. Setting the Constraint violation tol. to zero will force HyperStudy to satisfy the constraint identically, which can increase the run time of the optimization in some cases.
12. Click ***Apply***.
13. Go to the **Evaluate** step.
14. Click ***Evaluate Tasks*** to optimize the design that minimizes the maximum acceleration while keeping the displacement of node 35527 smaller than 16.
15. Click the ***Iteration Plot*** tab to monitor the Optimization iteration.



16. Click the ***Iteration History*** tab to review a table of each iteration. The iterations that do not respect the constraint are displayed red, the optimal design is displayed green.

Step 12: Evaluate the Results of the Approximate Optimal Design

You have now determined the best design evaluated from the approximation. You can launch a solver run with the following input variables to check if the solution found by the approximation is close to the solver results.

1. In the **Explorer**, study **Setup**, go to the **Define Input Variables** step.
2. In the work area, change the **Nominal** values to the values defined in the table below:

Variable Name	Optimum Value
th_external_skin	1.9216688
th_internal_skin	1.9999998
th_external_flange	1.00
th_internal_flange	1.00
Radius	0.00
length_external	0.00
width_external	0.00
length_internal	-0.9999824

3. Go to the **Evaluate** step.

4. Click ***Evaluate Tasks***.
5. Go to the **Define Output Responses** step.
6. Compare the optimum solution evaluated from the approximation to the same design evaluated by the solver.

   Note: Due to the use of different solver versions, results may vary. In following comparison, the solver and approximation are in very good agreement with < 1% error.

Output Responses	Approximation	Solver
Max_Acceleration	12.246401	12.293556
Max_Displacement	15.999990	16.046738

Last modified: v2017.2 (12.1156684)