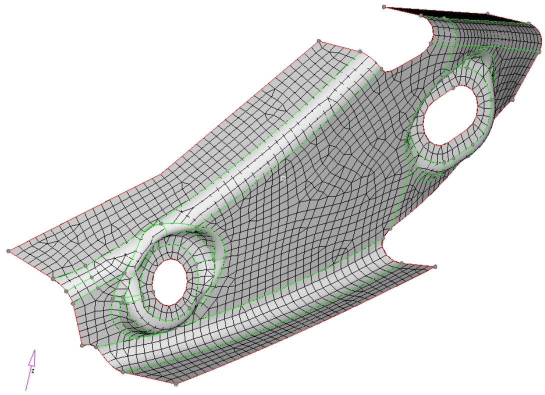
Tutorial:

Modal Analysis with Altair OptiStruct / HyperMesh

Some hints



- All components in the model must have material and properties assigned/defined. Make sure units are consistent and density is defined.
  (Example - if model is in mm for Steel then: Youngs Modulus = 210.000 MPa, Density = $7.9e^{-9}$ t/mm^3)
- Modal analysis is typically a free or constrained model. A free analysis doesn't require constraints but will generate rigid body modes. You can avoid extracting these using variables on the EIGRL card.

Modeling process

- Mesh components and assign properties
- Define constraints (if desired) and modal extraction load collector
- Build load step / subcase
- Define any special output requests and/or control cards

Let's get started

- Start HyperMesh and set the user profile to OptiStruct
- Open the model *model_start.hm*

Meshing

Mesh the model using **AutoMesh**; element size of approx. 2mm (mixed element types)

1. Access the **Mesh** panel
   On the **Menu** Bar, click ***Mesh***>***Create***>***2D AutoMesh***
2. Select ***surfs >> displayed***

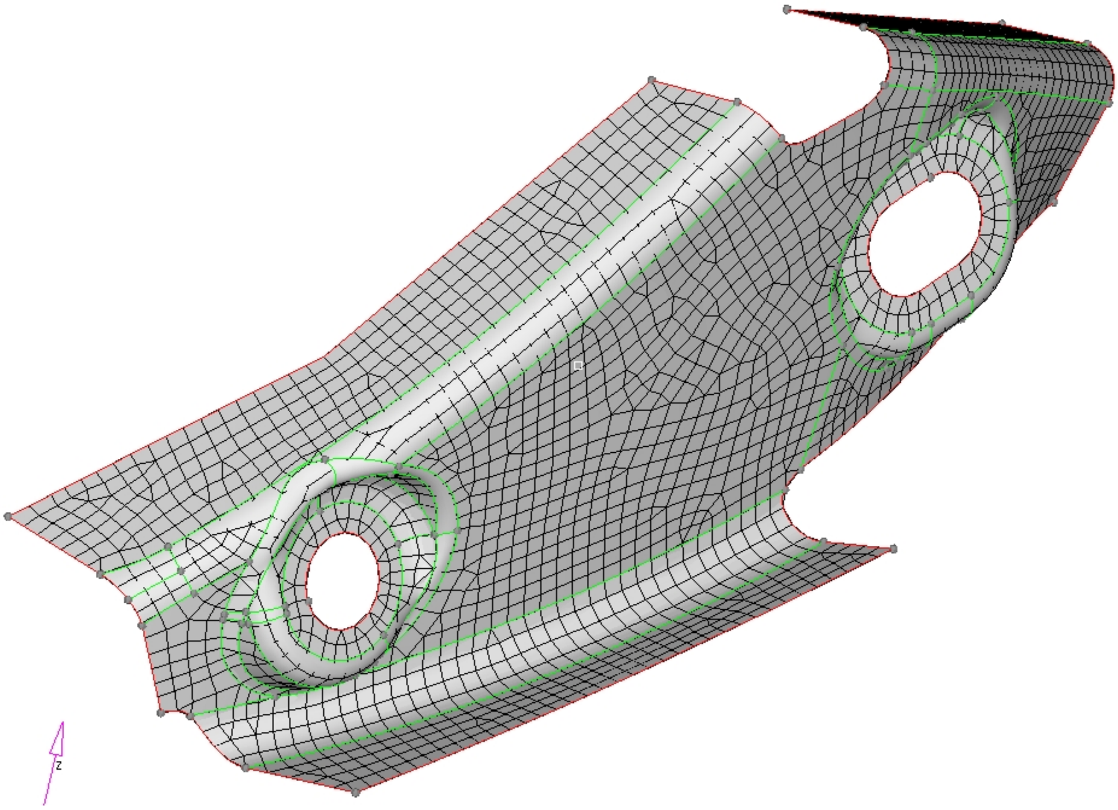
   For **element size=**, specify 2Set the **mesh type:** to ***mixed***.

   On the panel's bottom-left corner, set ***interactive*** as the active mesh mode (it may currently be on **automatic**).Ensure that the **elements to surf comp/elements to current comp** toggle is set to ***elems to current comp***.

   Click ***mesh*** to enter the meshing module

   (Notice that you are in the **density** subpanel of the meshing module. There is node seeding and a number on each surface edge. The number is the number of elements that were created along the edge)

Click ***return*** to accept the mesh as the final mesh

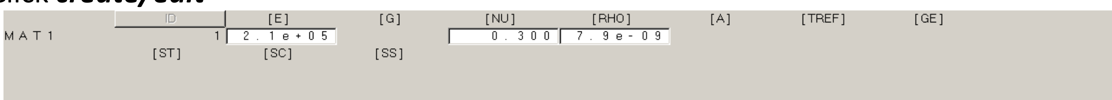


Material definition

Create a material of type MAT1… make it steel

1. Click the ***Materials*** icon  (alternatively, the material can be created in different other ways, e.g. Model Browser, pull-down menu etc.).

   Make sure the **create** subpanel is selected using the radio buttons on the left-hand side of the panel.
2. Click ***mat name =*** and enter steel
3. Click ***type =***and select ***ISOTROPIC***4. Click ***card image =*** and select ***MAT1***
5. Click ***create/edit***



The **MAT1** card image appears.

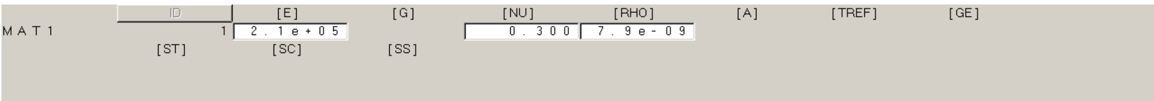
(Notice: If a material property in brackets does not have a value below it, it is off. To edit these material properties, click the property in brackets you wish to edit and an entry field will appear below it. Click the entry field and enter a value)

6. Enter the following values for:

   E as 2.1e5

   NU as 0.3

   RHO as 7.9e-09



7. Click ***return*** twice.

A new material, steel, has been created. The material uses OptiStruct's linear isotropic material model, MAT1. This material has a Young's Modulus of 2.1e+05 MPa, a Poisson's Ratio of 0.3, and a density of 7.9e^{-9} t/mm^3

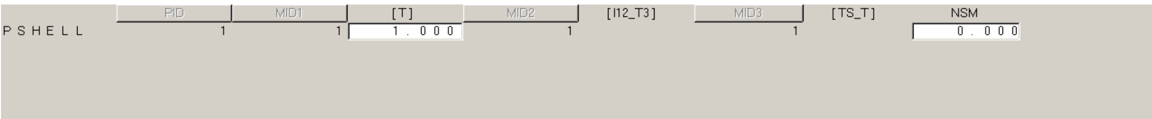
At any time, the card image for this collector can be modified using the **Card Editor** (alternatively, the card image can be modified trough the Model Browser too)

Property definition

Create the property collector named prop_model, make it a PSHELL 1mm thick made of steel

Click the ***Properties*** icon (alternatively, the property can be created in different other ways, e.g. Model Browser, pull-down menu etc.).
Make sure the **create** subpanel is selected using the radio buttons on the left-hand side of the panel.

1. Click ***prop name =*** and enter prop_model
2. Click ***type =*** and select ***2D***.Click ***card image =*** and select ***PSHELL***
4. Click ***material =*** and select ***steel***.Click ***create/edit***The **PSHELL** card image appears.



7. Click [***T***] and enter 1.0 as the thickness of the plate
8. Click ***return*** twice and go back to the main menu.
   The property of the shell structure has been created as 2D PSHELL. Material information is also linked to this property.

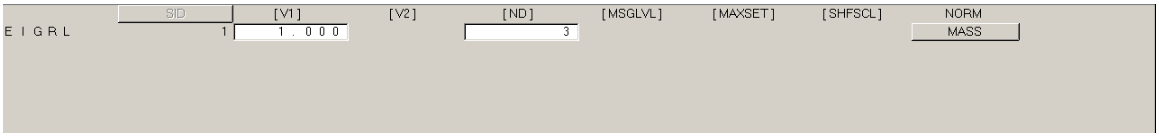
9. Go to the assign subpanel (in the panel), set type to all and for property select the property created in the previous step, then click on elements and select ALL and then click on Assign.

Loads

Create/Edit load collector called freq of type EIGRL
This can be done using the **Load Collectors** panel and the **create** subpanel (again, load collector can be created in many different ways).

1. From the toolbar, enter the **Load Collectors** panel by clicking the ***Load Collectors*** icon . Make sure the **create** subpanel is selected using the radio buttons on the left-hand side of the panel.

2. Click ***loadcol name =*** and enter modal
3. Click ***card image =*** and select **EIGRL**Click ***create/edit***.

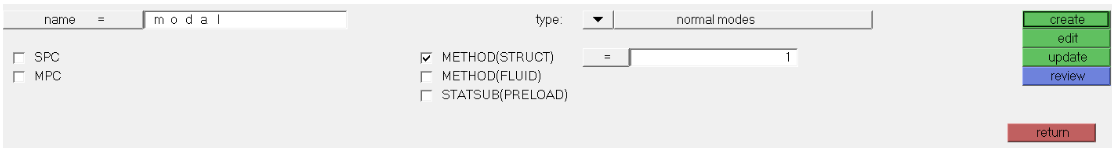


Note V1 = 1.0 and ND = 3
V1&V2 define the range you wish to extract the modes over. ND defines the number of modes you wish to find. Various combinations of these cards can be used to control what is extracted. This combination extracts the first 3 modes above 1 Hz. Setting V1 to 1.0 is a “trick” to avoid the extraction of rigid body modes.5. Click ***return***.

Loadstep / subcase

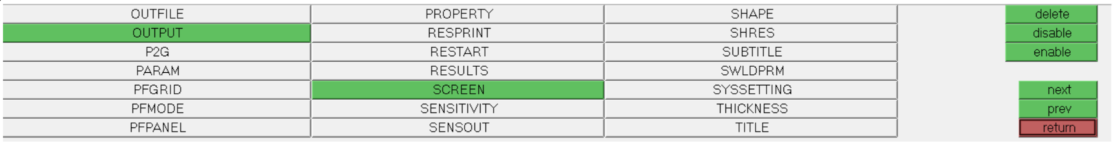
Create a subcase called *modal* that points to the EIGRL card defined above. This is done using the METHOD card as shown below.

1. Click ***Setup > Create > LoadSteps*** to open the **LoadSteps** panel.
2. For **name =**, enter modal and type select **normal modes**
3. Select METHOD and then click on the field next to method to select the load collector with the EIGRL card image created in the previous step.



Analysis

In the Control Cards (located on page Analysis→Control Cards→) set the SCREEN output to OUT

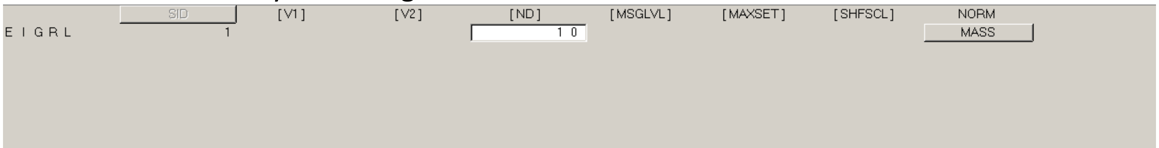


Save the model then run the analysis from the OptiStruct panel (***Analysis → Optistruct)***
You should see the following results. Note the first frequency is 429 Hz.

Subcase	Mode	Frequency	Eigenvalue	Stiffness	Mass
1	1	4.296100E+02	7.286324E+06	7.286324E+06	1.000000E+00
1	2	8.960945E+02	3.170059E+07	3.170059E+07	1.000000E+00
1	3	1.429868E+03	8.071453E+07	8.071453E+07	1.000000E+00

- On the post processing page (in HyperMesh) or start HyperView, go to the deformed panel. You will be able to animate the 3 mode shapes using the modal button.

- Edit the EIGRL card by removing V1 and find the first 10 modes.

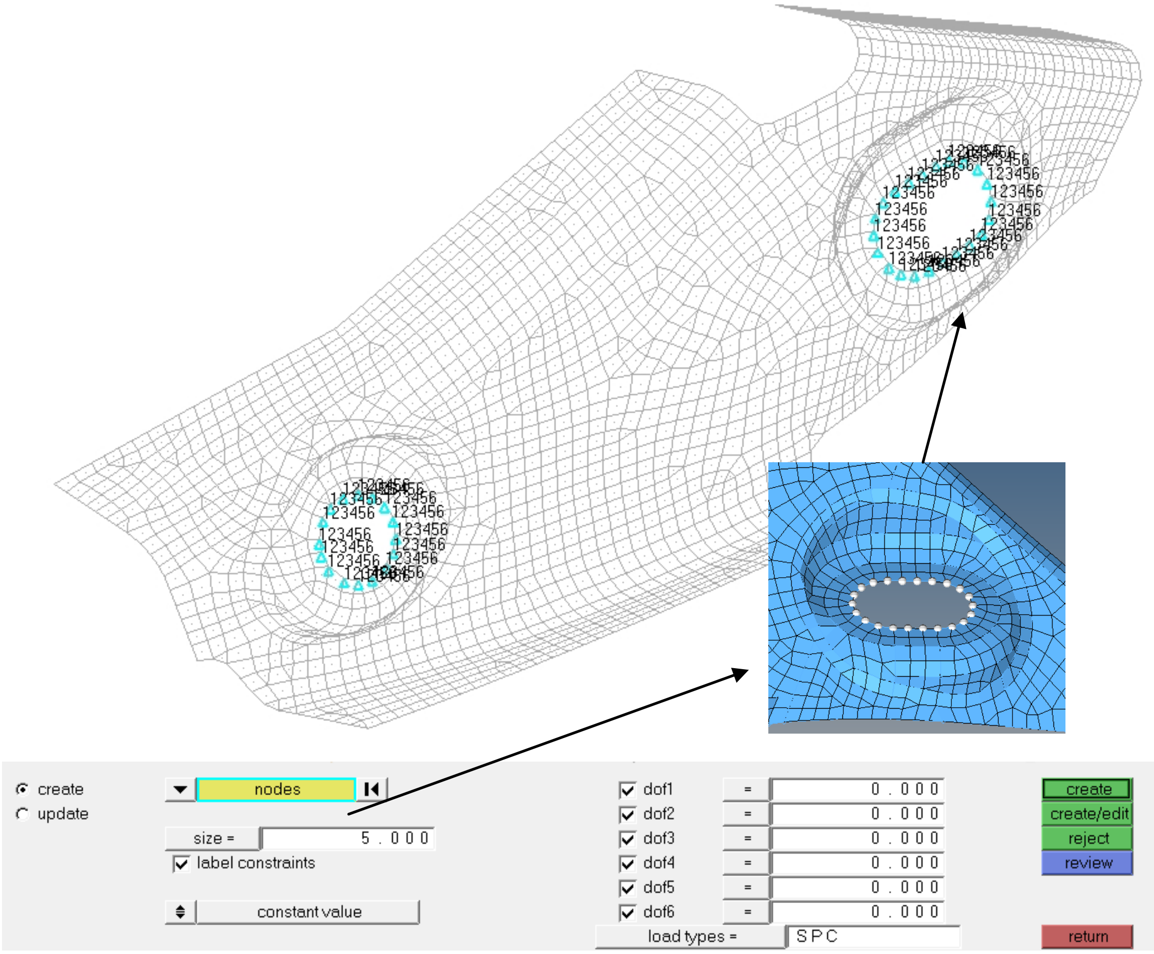


Re-run the model. Note the first 6 modes are now very small – these are the 3 transversal and rotational degrees of freedom.

Subcase	Mode	Frequency	Eigenvalue	Stiffness	Mass
1	1	6.797740E-03	1.824269E-03	1.824269E-03	1.000000E+00
1	2	7.368257E-03	2.143331E-03	2.143331E-03	1.000000E+00
1	3	7.779286E-03	2.389127E-03	2.389127E-03	1.000000E+00
1	4	7.857455E-03	2.437382E-03	2.437382E-03	1.000000E+00
1	5	8.119594E-03	2.602725E-03	2.602725E-03	1.000000E+00
1	6	8.581614E-03	2.907352E-03	2.907352E-03	1.000000E+00
1	7	4.296100E+02	7.286324E+06	7.286324E+06	1.000000E+00
1	8	8.960945E+02	3.170059E+07	3.170059E+07	1.000000E+00
1	9	1.429868E+03	8.071453E+07	8.071453E+07	1.000000E+00
1	10	1.776882E+03	1.246455E+08	1.246455E+08	1.000000E+00

Modal analysis with constrain model

1. Create another load collector (using <u>no</u> card image) called **constraints**
2. From **Model Browser** expand **LoadCollectors**, right-click on **constraints** and click ***Make Current*** to set constraints as the current load collector.
3. Fully constrain (all 6 dofs) the nodes around each hole as illustrated in the image below.



- Click ***BCs > Create > Constraints or Analysis page > constraints*** to open the **Constraints** panel. Make sure **nodes** are selected from the entity selection switch.
- Click ***nodes*** and select ***the nodes around the two holes***
- Constrain **dof1**, **dof2**, **dof3**, **dof4**, **dof5**, and **dof6** and set all of them to a value of 0.0.

Notice:
Dofs with a check will be constrained, while dofs without a check will be free.
Dofs 1, 2, and 3 are x, y, and z translation degrees of freedom.
Dofs 4, 5, and 6 are x, y, and z rotational degrees of freedom.

- Click ***create***

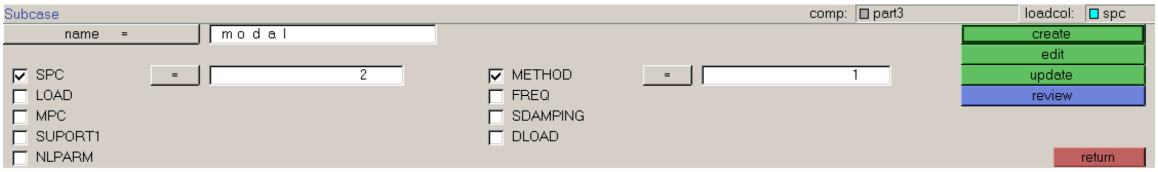
This applies the constraints to the selected nodes.

- Click ***return*** to go to the main menu

- Update the subcase to include this SPC (= constraint) set

    - Click ***Setup > Create > LoadSteps*** to open the **LoadSteps** panel.

- Select SPC and then click on the field next to SPC and select the constraints load collector created above,



- Re-run the analysis. Note that the rigid body modes are now gone.

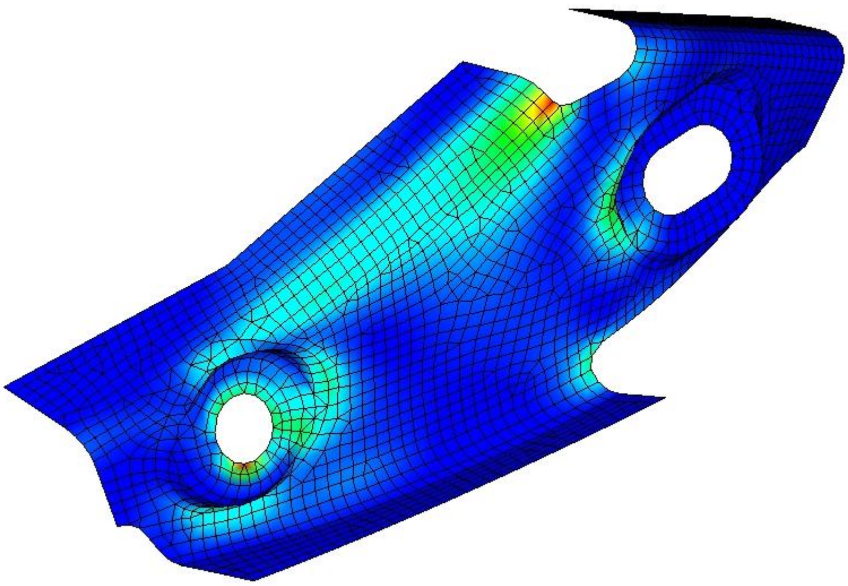
Subcase	Mode	Frequency	Eigenvalue	Stiffness	Mass
1	1	1.349106E+03	7.185419E+07	7.185419E+07	1.000000E+00
1	2	1.847394E+03	1.347345E+08	1.347345E+08	1.000000E+00
1	3	1.859691E+03	1.365342E+08	1.365342E+08	1.000000E+00
1	4	2.034115E+03	1.633468E+08	1.633468E+08	1.000000E+00
1	5	2.111028E+03	1.759332E+08	1.759332E+08	1.000000E+00
1	6	2.611230E+03	2.691846E+08	2.691846E+08	1.000000E+00
1	7	2.806940E+03	3.110470E+08	3.110470E+08	1.000000E+00
1	8	3.123768E+03	3.852274E+08	3.852274E+08	1.000000E+00
1	9	3.228458E+03	4.114812E+08	4.114812E+08	1.000000E+00
1	10	3.552249E+03	4.981574E+08	4.981574E+08	1.000000E+00

More post processing

- Sometimes you want more data from your analysis or your results in different formats. Note that by default OptiStruct creates an HTML file summarizing the analysis and writes a HyperMesh results file. To get results in other formats e.g. H3D use the output control card (Analysis→Control Cards→….).

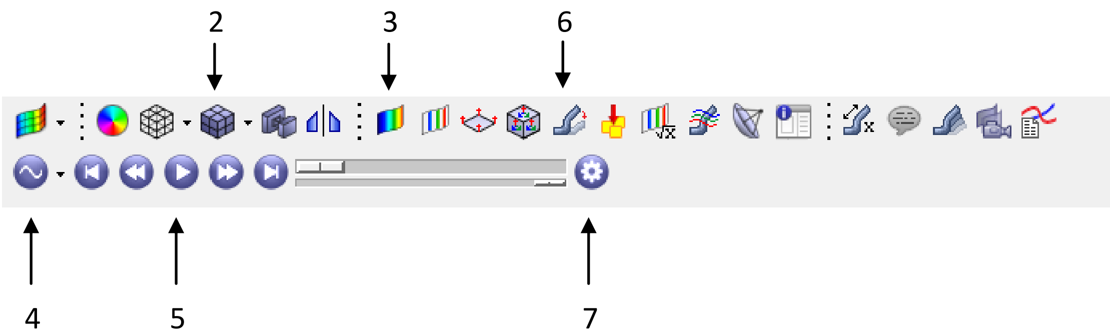
- Another output that is often useful in modal analysis is element strain energies. You can request these using the ESE function of the GLOBAL_OUTPUT_REQUEST control card (Analysis→Control Cards→….).

The strain energy results allow you to identify areas critical to given modes

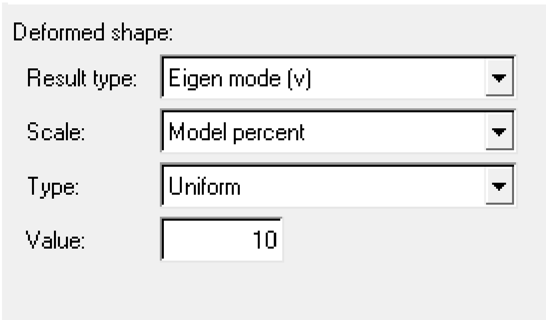


Post processing with HyperView

1. Load the .h3d file into HyperView using *Load model*
2. Use the *Shaded Elements and Mesh lines* button to turn the mesh on
3. Use the *Contour* button and plot Strain Energy
4. Change the *traffic light* from transient to linear/modal
5. Animate the first mode by clicking the *Play* button
6. If the deformation is too large change it to *Model percent* (10%) using the *Deformed* options
7. Add more frames and control the speed of the animation using the *Animation controls* button
8. Step through the modes using *load case selector*



Deformed Panel (6):



Animation controls panel (7):

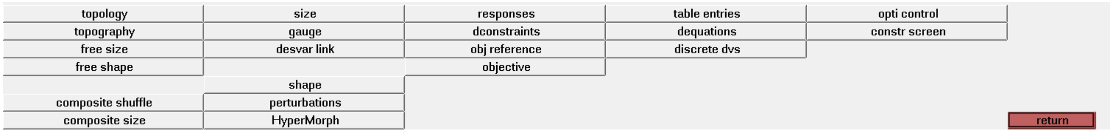
Optimization

Optimization is often very simple and effective when applied to modal problems. The setup in HyperMesh makes **gauge optimization** very easy.

Define a simple gauge optimization problem on the modeled part.

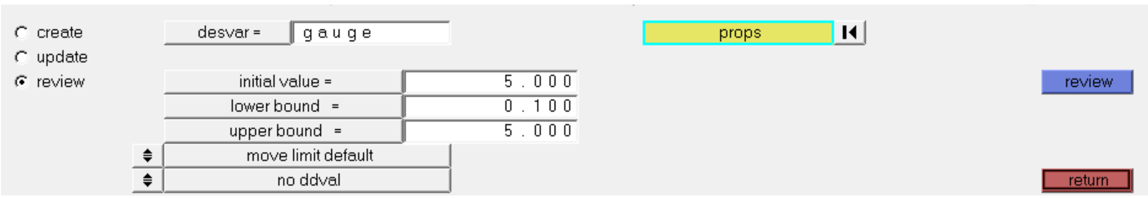
- Minimize mass
- Design variable is the gauge
- Minimum frequency is 1000 Hz

The following steps can be performed from the panels of the optimization module, to go to the optimization module go to Analysis page → optimization

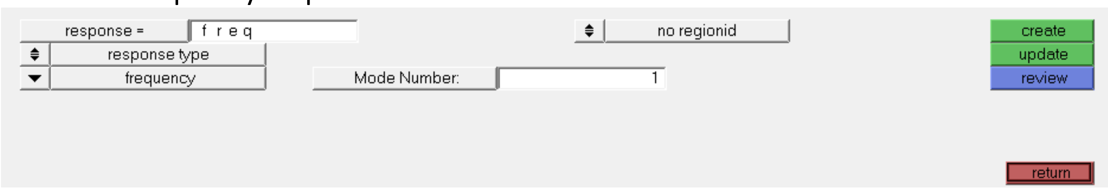


Set up the design variable

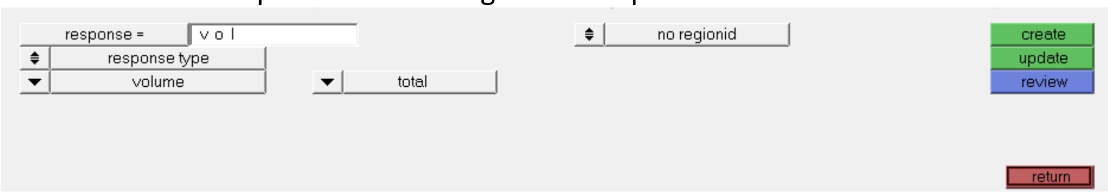
- go to the optimization / gauge panel
- select the property "prop_model" and define the desvar range to be between 0.1 and 5 mm



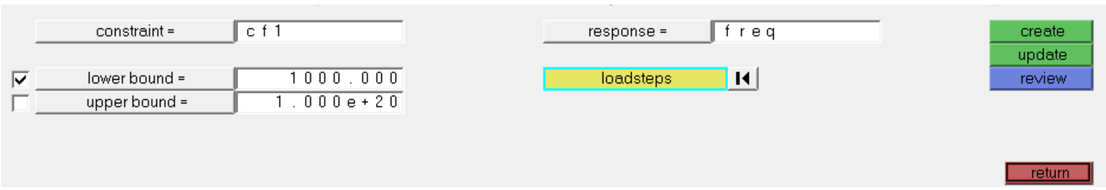
- Define responses of interest
  - define a frequency response for the first mode



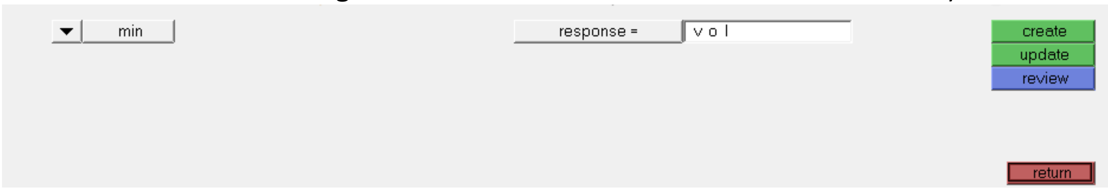
- Define a volume response of the designable component



- Define the constraint that the minimum first mode is > 1000 Hz



- Define the objective to be: minimize the volume (This could be mass, cost, whatever. Volume is quite good to use as the units are usually “big” whereas mass units can be small e.g. 5.3×10^{-5} t. which can lead to tolerance issues).



- Save and run the model…

- From the screen output and the .out file you should note that OptiStruct rapidly determines the optimum gauge (here 0.629 mm) to use to meet the frequency target

RETAINED RESPONSES TABLE

Response User-ID	Type	Response Label	Subcase /RANDPS +Frqncy	Grid/ Element/ MID/PID/ Mode No.	DOF/ Comp /Reg	Response Value	Objective Reference/ Constraint Bound	Viol. %
2	VOLUM	vol	--	--	TOTL	5.183E+03	MIN	
1	FREQ	freq	1	1	--	9.993E+02	> 1.000E+03	0.0 A

Design Variable ID	Design Variable Label	Lower Bound	Design Variable	Upper Bound
1	gauge	1.000E-01	6.629E-01	5.000E+00

DESIGNED PROPERTY ITEMS TABLE

DVPREL1/2	USER-ID	PROP-TYPE	PROP-ID	ITEM-CODE	PROP-VALUE
DVPREL1	1	PSHELL	2	T	6.629E-01

****

OPTIMIZATION HAS CONVERGED.

FEASIBLE DESIGN (ALL CONSTRAINTS SATISFIED).