Altair

HyperWorks

Altair OptiStruct 2019

Basic Linear and Dynamic Solutions

altairhyperworks.com

Technical Support

Altair provides comprehensive software support via web FAQs, tutorials, training classes, telephone and e-mail.

Altair Support on the World Wide Web

The Altair web site is a valuable online companion to Altair software. Visit www.altairhyperworks.com for tips and tricks, training course schedules, training/tutorial videos, and other useful information.

Altair Training Classes

Altair training courses provide a hands-on introduction to our products, focusing on overall functionality. Courses are conducted at our main and regional offices or at your facility. If you are interested in training at your facility, please contact your account manager for more details. If you do not know who your account manager is, please send an e-mail to training@altair.com and your account manager will contact you.

Telephone and E-mail

When contacting Altair support, please specify the product and version number you are using along with a detailed description of the problem. Many times, it is very beneficial for the support engineer to know what type of workstation, operating system, RAM, and graphics board you have, so please have that information ready. If you send an e-mail, please specify the workstation type, operating system, RAM, and graphics board information in the e-mail.

To contact an Altair support representative, reference the following table or the information available on the HyperWorks website: www.altairhyperworks.com/ClientCenterHWSupportProduct.aspx.

Location	Telephone	E-mail
Australia	64.9.413.7981	anzsupport@altair.com
Brazil	55.11.3884.0414	br_support@altair.com
Canada	416.447.6463	support@altairengineering.ca
China	86.400.619.6186	support@altair.com.cn
France	33.1.4133.0992	francesupport@altair.com
Germany	49.7031.6208.22	hwsupport@altair.de
India	91.80.6629.4500 1.800.425.0234 (toll free)	support@india.altair.com
Israel		israelsupport@altair.com
Italy	39.800.905.595	support@altairengineering.it

Location	Telephone	E-mail
Japan	81.3.6225.5830	support@altairjp.co.jp
Malaysia		aseansupport@altair.com
Mexico	55.56.58.68.08	mx-support@altair.com
South Africa	27 21 8311500	support@altair.co.za
South Korea	82.70.4050.9200	support@altair.co.kr
Spain	34 910 810 080	support-spain@altair.com
Sweden	46.46.460.2828	support@altair.se
United Kingdom	01926.468.600	support@uk.altair.com
United States	248.614.2425	hwsupport@altair.com

For questions or comments about this help system, send an email to connect@altair.com.

In addition, the following countries have resellers for Altair Engineering: Colombia, Czech Republic, Ecuador, Israel, Russia, Netherlands, Turkey, Poland, Singapore, Vietnam, Indonesia

Official offices with resellers: Canada, China, France, Germany, India, Malaysia, Italy, Japan, Korea, Spain, Taiwan, United Kingdom, USA

See www.altair.com for complete contact information.

Table of Contents
OptiStruct for Linear Analysis

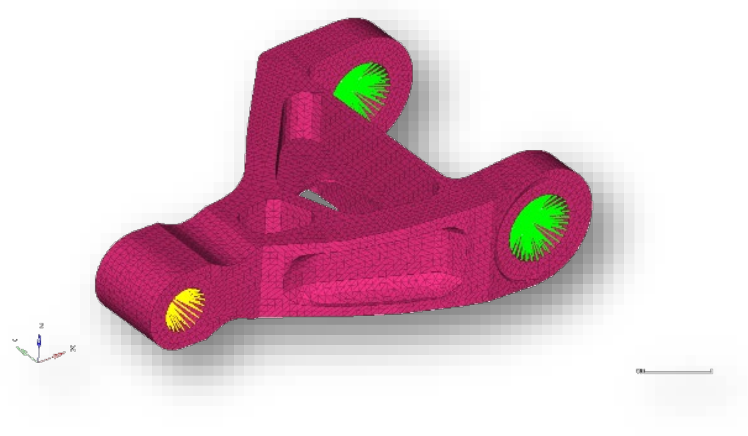
Basic Linear and Dynamic Solutions

Chapter 2

Linear Static Analysis

Exercise 2a: Static Analysis of a Solid Bracket

In this exercise, a structural analysis is performed on a bracket modeled with solid elements. The objective is to set up a linear static analysis from scratch starting just with the meshed model.



Model Information

- Force = (12000,12000, -20000) N
- Material Aluminium:
  - E =70000 MPa
  - Nu = 0.33
  - S_0 = 240 Mpa
  - $S_{ADM} = 0.7*S_0$
- UNITS: N, mm, ton, s

File Name and Location

…\STUDENT-EXERCISE\2a_Torsion_Link\torsion_link.hm

Step 1: Open the model in HyperMesh Desktop with OptiStruct user profile selected

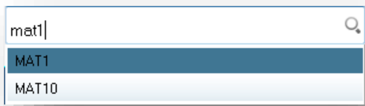
Step 2: Review the model and check the dimensions of the model

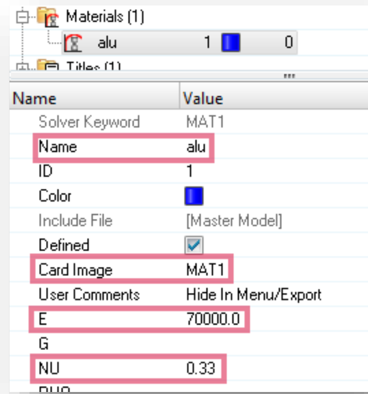
Tip: The length system is reasonable to be millimeter, the consistent units are mm, MPa, N. The mesh size is about 10 mm. There are three components:

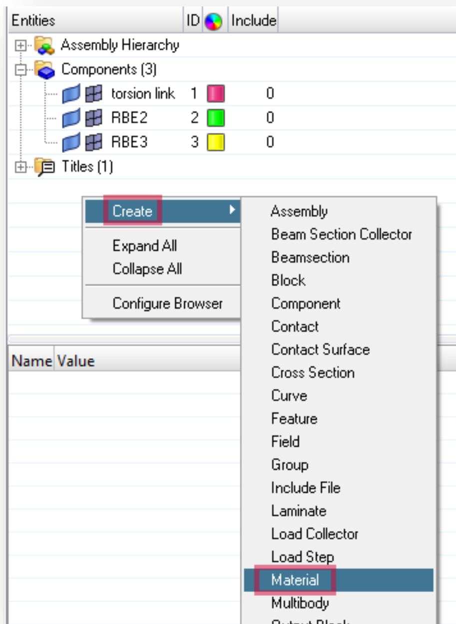
- `torsion link` for tetra elements
- `RBE2` for RBE2 elements for two supports
- `RBE3` for a RBE3 element to apply the force. `RBE2` would include a rigid condition that doesn't exist.

Step 3: Create a `MAT1` material `alu` for aluminum with the properties: Young's modulus 70000 MPa, Poisson's ratio 0.33

Tip: Click right mouse button in the **Model Browser**, select ***Create* > *Material*** or use HyperMesh's Quick Access Tool (`Ctrl+f`) to create accordingly `MAT1` material card.



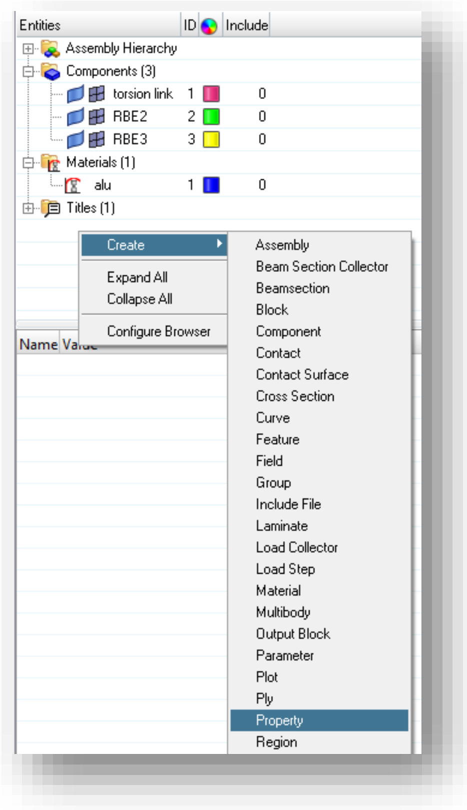


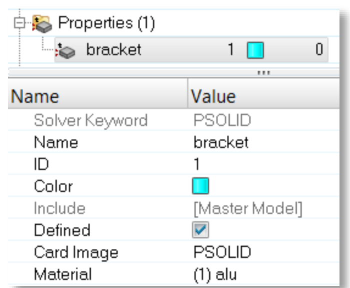


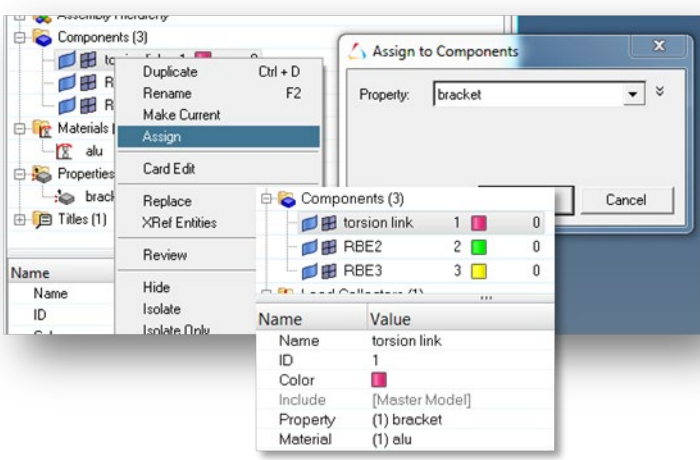
Here we will use MAT1 which is a linear isotropic material that can represent the aluminum behavior well. For more details about this material or other material formulations, please refer to the HyperWorks Online Help

Step 4: Create a `PSOLID` property `bracket` referencing material `alu` and assign it to the `torsion link` component

Tip: Click right mouse button in the model browser, select ***Create > Property*** or use HyperMesh's Quick Access Tool (`Crtl+f`) to create according `PSOLID` property card and assign property to `torsion link` component by right mouse click on the component, select **Assign**. This will make all elements in this component use this property. Note that this assignment serves as a default for this collector and will not change any element from this component has another property directly associated with it in its element definition

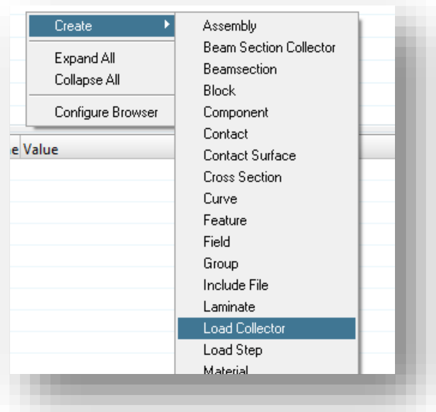


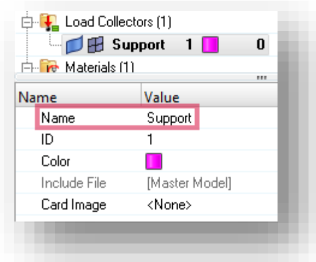




Step 5: Create a load collector `Support` (no card image) with the following `SPC` load type constraint, node 4830: DOFs 1-6 and node 4831: DOFs 1, 3-6

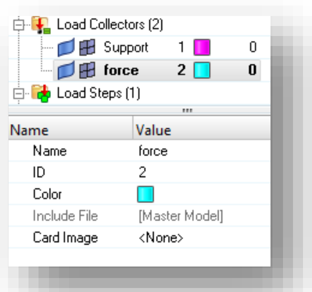
Tip: Click right mouse button in the **Model Browser**, select ***Create* > *Load Collector*** and create the two SPCs. The boundary conditions can be created from the menu ***BCs* > *Create* > *Constraints***

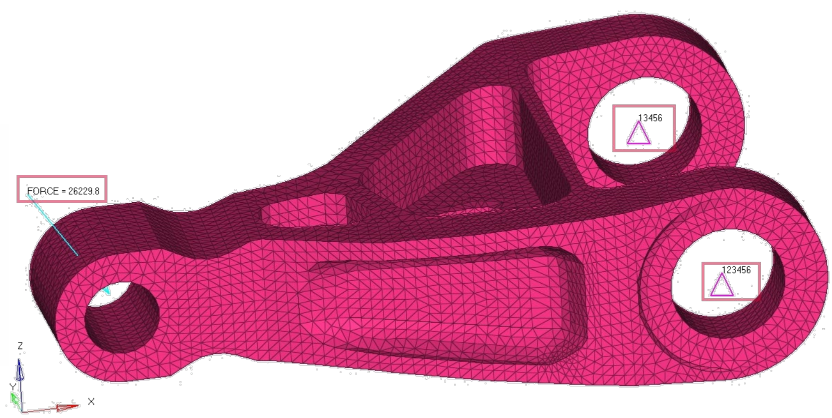




Step 6: Create a load collector `force` (no card image) containing a force on node 1 with constant components {12000, 12000, -20000}

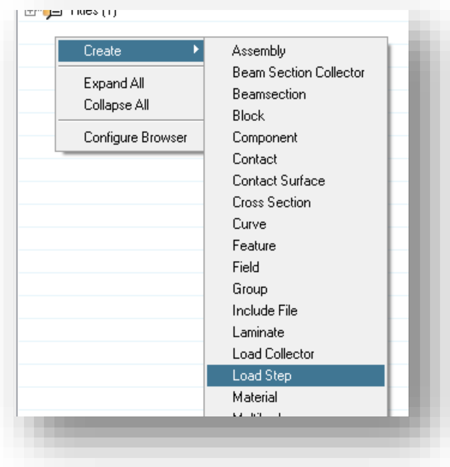
Tip: Click right mouse button in the **Model Browser**, select ***Create* > *Load Collector*** and create the force using the top menu ***BCs* > *Create* > *Forces***

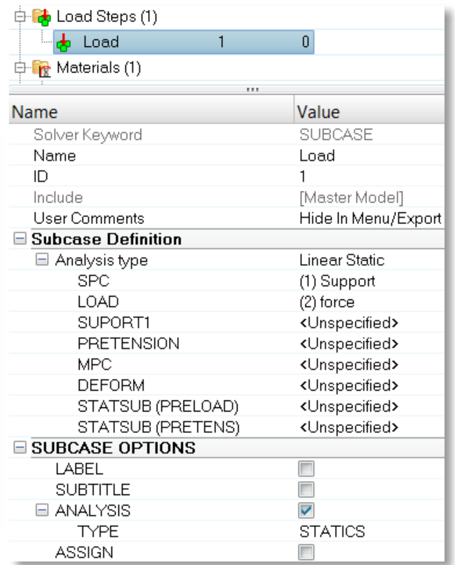




Step 7: Create a load step `Load` of type `Linear Static` using `Support` as `SPC` and `force` as `LOAD` entry

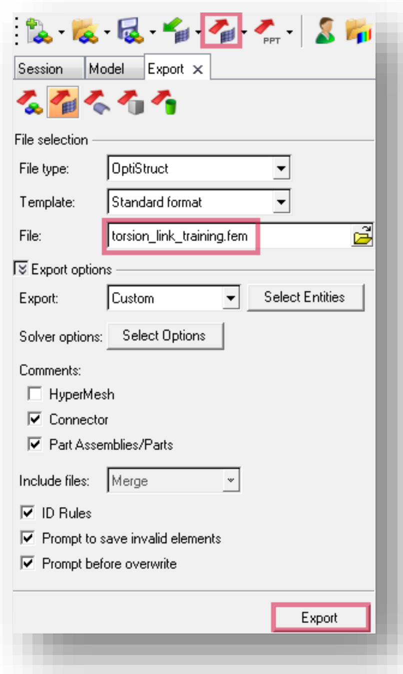
Tip: Click right mouse button in the **Model Browser**, select ***Create* > *Load Step***





Step 8: Export the model as solver deck

Tip: Click on button **Solver Export Deck**, choose file name and hit **Export**



Step 9: Review the `.fem` file in a text editor and understand the references

Tip: Exported `.fem` file without HyperMesh comments

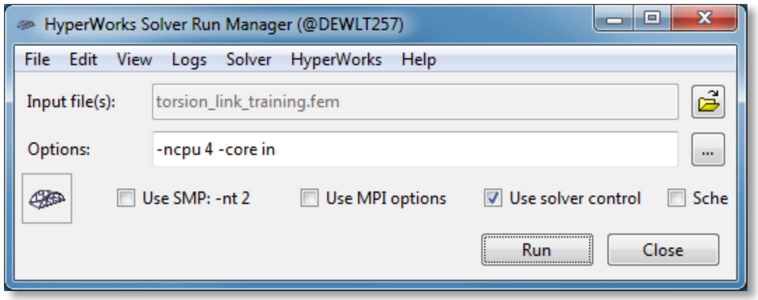
- Constraint reference

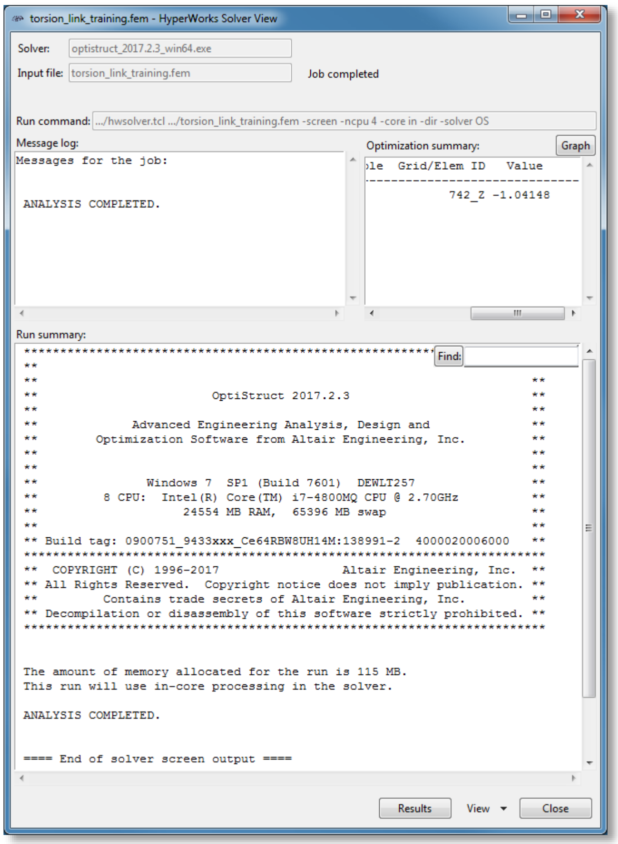
- Load reference
- Property reference
- Material reference

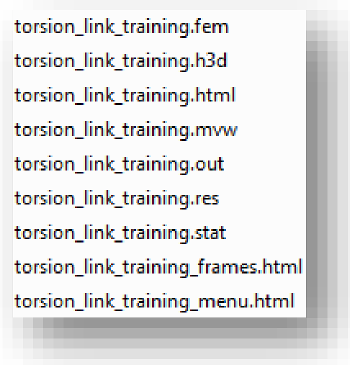
```
$$ optistruct
SUBCASE       1
  LABEL Load
ANALYSIS STATICS
  SPC =        1
  LOAD =        2
BEGIN BULK
GRID           1        -520.701-61.91252.0557-5
...
...
...
CTETRA     42822           1     5866     6434    10115    10208
PSOLID         1           1
MAT1           170000.0          0.33
SPC            1    4830  123456      0.0
SPC            1    4831   13456      0.0
FORCE          2       1       01.0      12000.0 12000.0 -20000.0
ENDDATA
```

Step 10: Run the analysis with OptiStruct

Tip: Use the HyperWorks Solver Run Manager to run the exported `.fem` file, make sure that OptiStruct states “ANALYSIS COMPLETED” and review the created files.





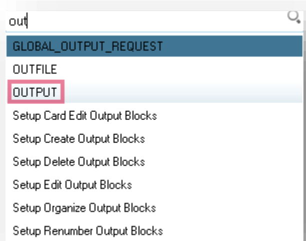


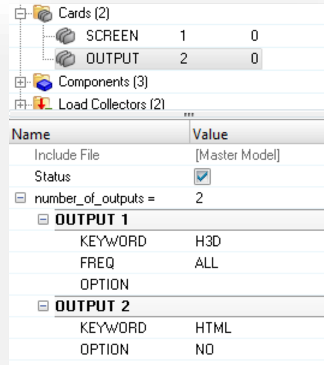
Step 11: Add output requests in order to get desired

1. echo .out file on the screen
2. get results only in H3D format and suppress the html file output

Tip: Use HyperMesh's Quick Access Tool (`Crtl+f`) to add control cards

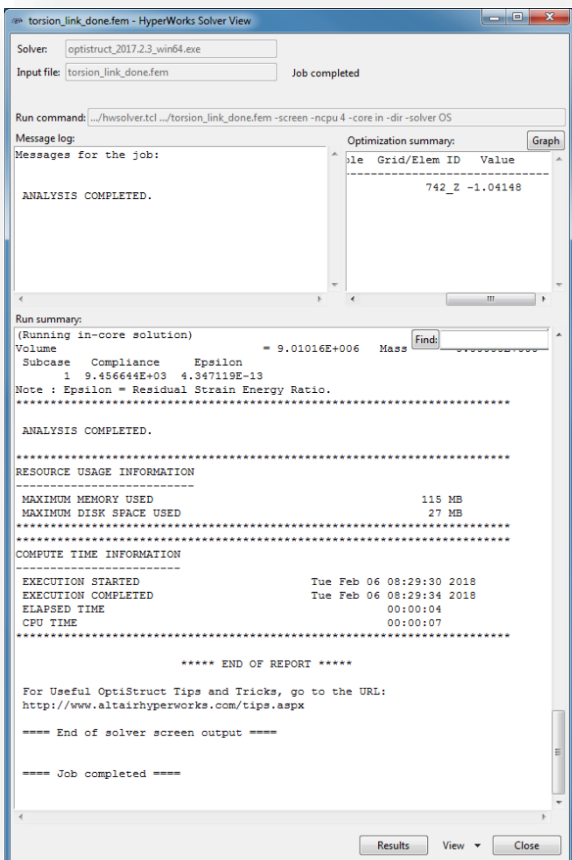
```
SCREEN,OUT
OUTPUT,H3D,ALL
OUTPUT,HTML,,NO
```

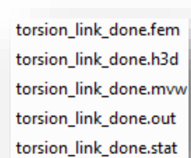




Step 12: Rerun the analysis with OptiStruct

Tip: Review the created files and note the additional output in the HyperWorks Solver View





Step 13: Review the `.out` file wrt warnings, errors and Auto-SPC

Tip: There are 19 elements that exceeded recommended range (warning) for the element quality check.

OptiStruct auto-SPCed 1344 degrees-of-freedom (DOF).

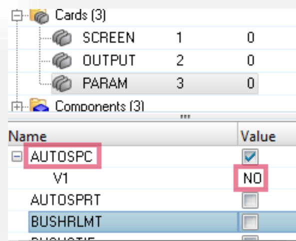
```
Element # 13814, element type TETRA.
WARNING - Outside of recommended range: Face Vertex Angle =    12.440
          lower limit =    15.000

NOTE : other similar error/warning messages were suppressed,
       use PARAM,CHECKEL,FULL to obtain full report


Element Quality Check Summary
-----------------------------
Total # of elements that exceeded recommended range (warning) =    19
Note: Only element with the highest violation of each check is listed below.

Recommended range violations:
------------------------------------------------------------------------------
Element      Property           # of     Recommended Range     Max.  Viol.  Elem.
                                Viol.     Lower     Upper     Value  type    No.
------- ------------------- -------- --------- --------- --------- --- -------
TETRA   Face Vertex Angle        19     15.00    165.00     11.95  L    15597


List of Auto-SPC d.o.f.s for loadcase   1
Total number of Auto-SPC d.o.f.s = 1344

               ------------------------------
                 Grid No.         Component
               ------------------------------
                      3870             4 5 6
                      3871             4 5 6
                      3872             4 5 6
                      3873             4 5 6
                      3874             4 5 6
                      3875             4 5 6
                      3876             4 5 6
                      3877             4 5 6
                      3878             4 5 6
                      3879             4 5 6
                      3880             4 5 6
                      3881             4 5 6
```

By default, OptiStruct uses AUTOSPC, ON as it helps to prevent undesired stops or failure runs. For example, if the model has an element unattached to the structure with no constraint applied to it, the run would stop complaining about a rigid body movement. With AUTOSPC ON, OptiStruct would automatically fix this element and run the analysis.

The user should be aware of any DOF fixed by the AUTOSPC as it can lead to a wrong behavior. Also, do not forget that in the end, if the run is made with "AUTOSPC ON", to verify which DOF was fixed and if this has not affected the solution.

Step 14: Add parameter to deactivate automatic constraining

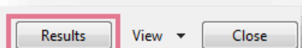
Tip: Use HyperMesh's Quick Access Tool (`Crtl+f`) to add control card
`PARAM,AUTOSPC,NO`
Note that this card will be added in the bulk section.



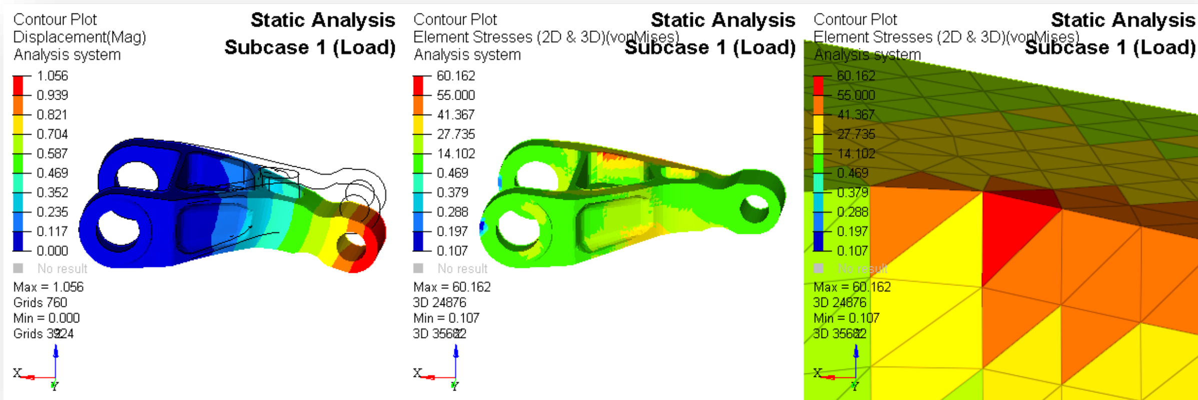
Step 15: Rerun the analysis with OptiStruct and review the `.out` file again

Step 16: Review the displacements and stresses in HyperView

Tip: Click on Results in the HyperWorks Solver View and HyperView will directly open the according `.mvw` session file created by OptiStruct.



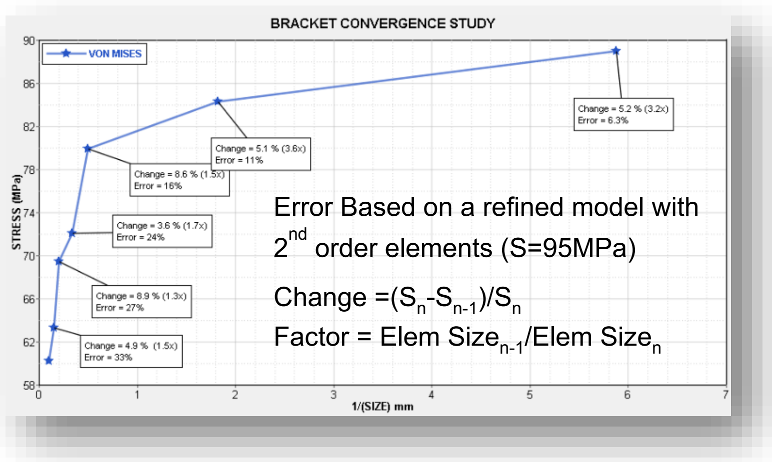
In the displacement contour plot the deformed shape is scaled by 100. In the stress contour plot it is easy to notice that the stress results are not ideal due to discontinuities in the mesh. The next step would be to rerun this model with a refined mesh.

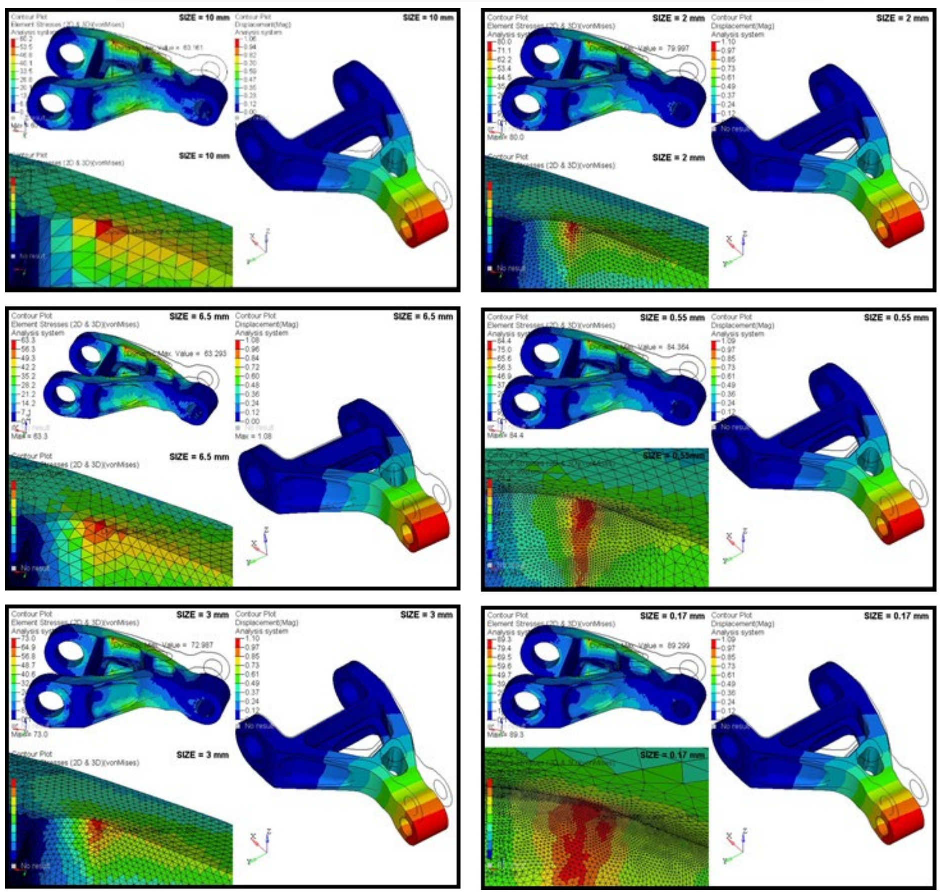


A finer mesh results on the one hand typically in a more accurate solution, on the other hand increases the computation time.

In order to get an idea of a finite element model that is good enough to predict an accurate solution for a problem with a reasonable model size, a convergence study can be performed:

- Create a mesh using low, but reasonable number of elements and do an analysis
- Refine the mesh, do a reanalysis and compare the results for the first mesh.
- Keep refining the mesh and reanalyzing until the results like max. stress and max displacement converge.


BRACKET CONVERGENCE STUDY
VON MISES
STRESS (MPa)
1/(SIZE) mm
Change = 4.9 % (1.5x)
Error = 33%
Change = 8.9 % (1.3x)
Error = 27%
Change = 3.6 % (1.7x)
Error = 24%
Change = 8.6 % (1.5x)
Error = 16%
Change = 5.1 % (3.6x)
Error = 11%
Change = 5.2 % (3.2x)
Error = 6.3%
Error Based on a refined model with
2nd order elements (S=95MPa)
Change =(Sn-Sn-1)/Sn
Factor = Elem Sizen-1/Elem Sizen

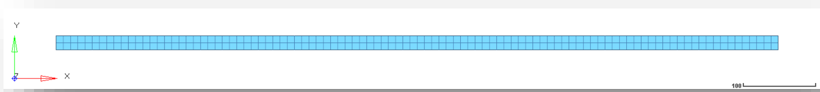

SIZE = 10 mm
SIZE = 2 mm
SIZE = 6.5 mm
SIZE = 0.55 mm
SIZE = 3 mm
SIZE = 0.17 mm

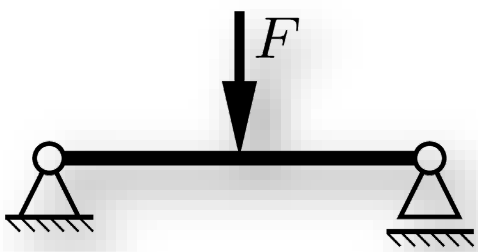
Element Size (mm)	VonMises (MPa)	Displacement (mm)
10	60.2	1.06
6.5	63.3	1.08
5	69.5	1.09
3	73.0	1.10
2	80.0	1.10
0.55	84.4	1.09
0.17	89.3	1.09

Exercise 2b: Static Analysis of a Simply Supported Beam

In this exercise, a structural analysis is performed on a simply supported beam. The objective is to create a finite element model that is good enough to predict the theoretical solution for this model.

- Beam modelled by shell elements, length = 1000 mm, height = 20 mm, width = 10 mm
- Material steel (Young's modulus 210000 MPa, Poisson's ratio 0.33, density 7.85e-9 t/mm3)
- Force of 1000 N in the center of the beam.





Theoretical Results

$$\sigma_{\max} = \frac{M_{\max} c}{I} = \frac{\frac{F*L}{4} * \frac{H}{2}}{\frac{B*H^3}{12}} = \frac{3FL}{2BH^2} = 375MPa$$

$$U_{\max} = -\frac{FL^3}{48EI} = -\frac{FL^3}{48E\,\frac{BH^3}{12}} = -\frac{FL^3}{4EBH^3} = 14.881mm$$

File Name and Location

…\STUDENT-EXERCISE\2b_Simple_Beam\beam.hm.

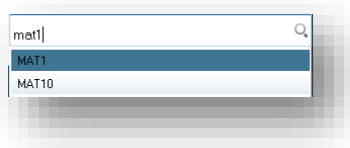
Step 1: Open the model in HyperMesh Desktop with OptiStruct user profile selected

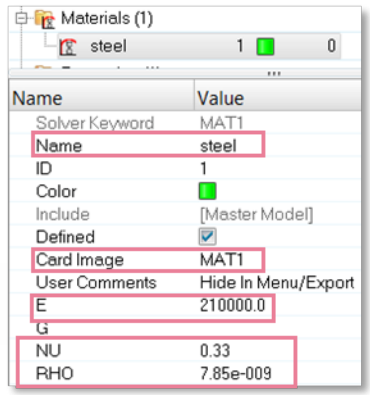
Step 2: Review the model and check the dimensions of the model

Tip: The length system is reasonable to be millimeter, the consistent units are mm, MPa, N. The mesh size is 10 mm. There is one components beam containing elements and surfaces

Step 3: Create a `MAT1` material `steel` for steel with the given properties

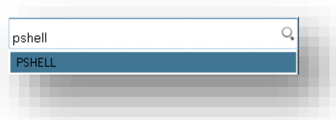
Tip: Use HyperMesh's Quick Access Tool (`Crtl+f`) to create according `MAT1` material card

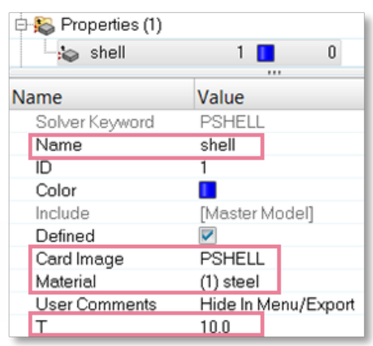


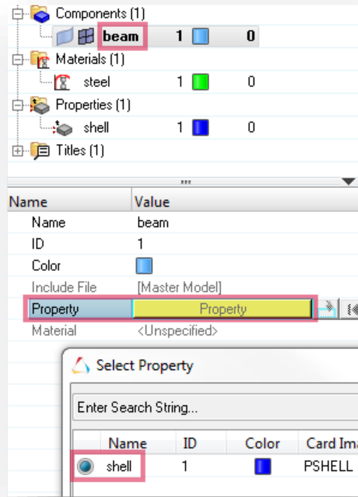


Step 4: Create a `PSHELL` property shell with a thickness of `10 mm` referencing material `steel` and assign it to the `beam` component

Tip: Use HyperMesh's Quick Access Tool (`Crtl+f`) to create according `PSHELL` property card and assign property to `beam` component in the entity editor





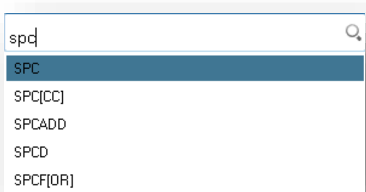


Step 5: Create a load collector `SPC` (no card image) with the following `SPC` load type constraint

- **Node 205: DOFs `1-5`**

- **Node 1: DOFs `2-3`**

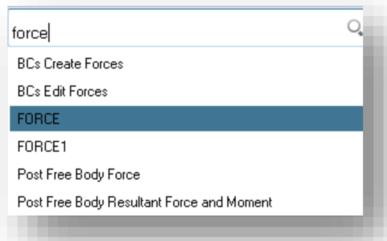
Tip: Use HyperMesh's Quick Access Tool (`Crtl+f`) with `SPC` to create the two `SPC`s



Step 6: Create a load collector `FORCE` (no card image) containing a force on node 53 with constant components {0, -1000, 0}

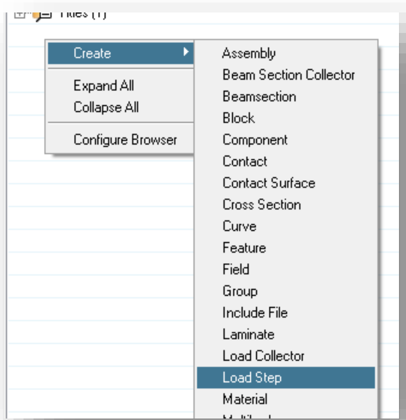
Tip: Use HyperMesh's Quick Access Tool (`Crtl+f`) with `FORCE` to create the load

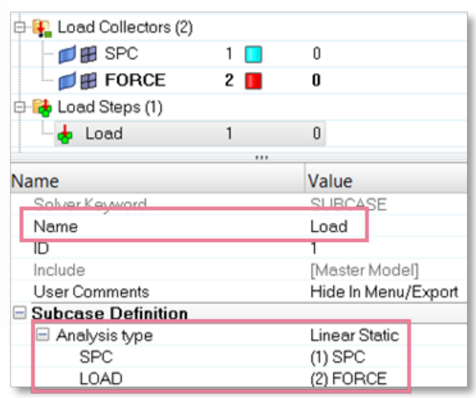
Be aware, that loads and constraints can generate singularities. These can lead to a very high stress that is not physical, and appears only in the mathematical model

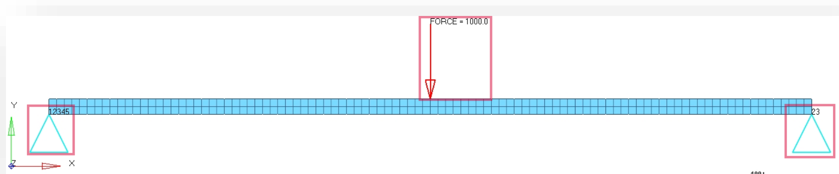


Step 7: Create a load step `Load` of type Linear Static using `SPC` as `SPC` and `FORCE` as `LOAD` entry

Tip: Click right mouse button in the **Model Browser**, select ***Create > Load Step*** and set ***Analysis type*** to `Linear Static`







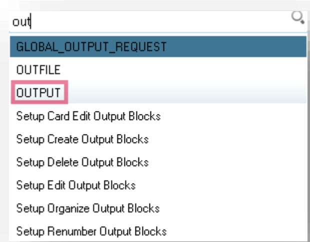
Step 8: Add output requests in order to

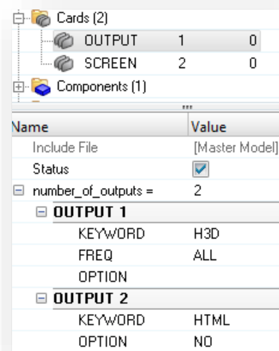
- **echo `.out` file on the screen**

- **get results only in `H3D` format and suppress the `html` file output**

Tip: Use HyperMesh's Quick Access Tool (`Crtl+f`) to add control cards

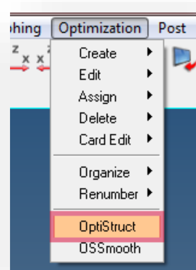
```
SCREEN OUT

OUTPUT, H3D, ALL

OUTPUT, HTML,, NO
```

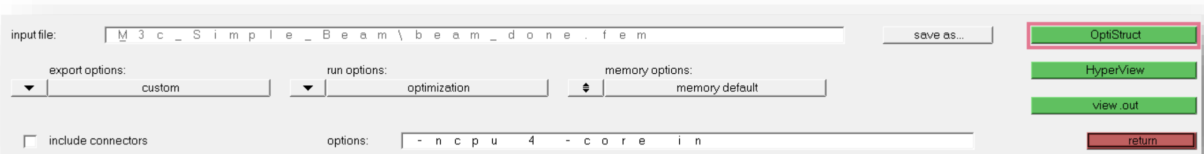




Step 9: Run the analysis with OptiStruct

Tip: Run the model in OptiStruct using e.g. the OptiStruct panel via pull-down menu ***Optimization > OptiStruct***





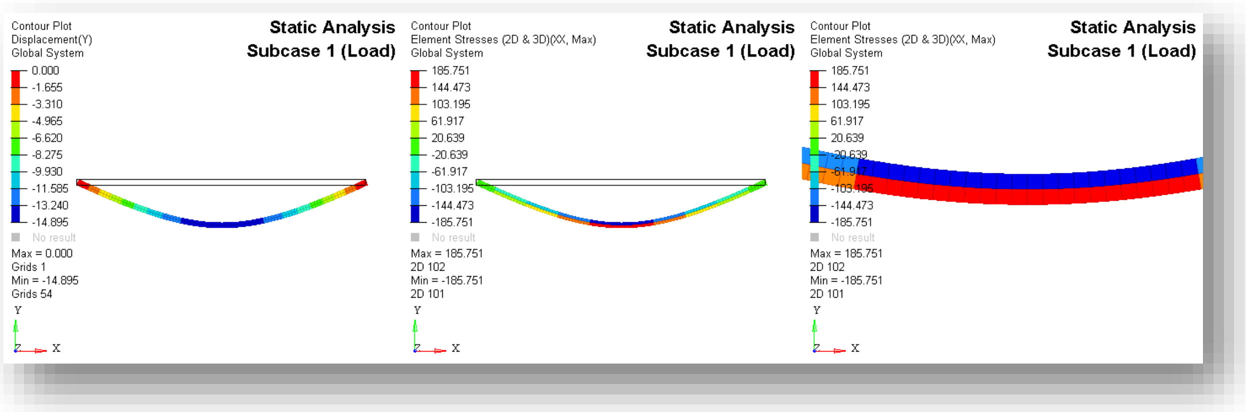
Step 10: Review the `.out` file wrt warnings, errors and Auto-SPC

Step 11: Review the displacements and stresses in HyperView and check

- **max. displacement in y-direction**

- **max. stress in (global) xx-direction**

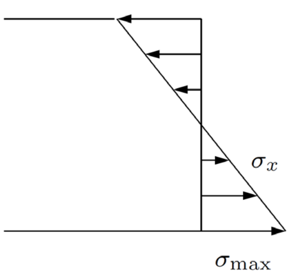
Tip: In the contour plots the deformed shapes are scaled by 10. The maximum displacement is 14.895 mm, the maximum stress is ±185.751 MPa (both in the center of the beam).



Step 12: Calculate the theoretical results for

- **max. displacement in y-direction**

- **max. stress in xx-direction**



$$u_{\max} = \frac{F\,l^3}{48\,EI} = \frac{F\,l^3}{48\,E\,\frac{t\,h^3}{12}} = \frac{F\,l^3}{4\,Eth^3} = \frac{-1000 \cdot 1000^3}{4 \cdot 210000 \cdot 10 \cdot 20^3} = 14.881\text{ mm} \quad \text{versus 14.895 mm}$$

$$\sigma_{\max} = \frac{M_{\max}\, z_{\max}}{I} = \frac{\frac{F\,l}{4}\left(\pm\frac{h}{2}\right)}{\frac{t\,h^3}{12}} = \pm\frac{3\,F\,l}{2\,th^2} = \pm\frac{3 \cdot 1000 \cdot 1000}{2 \cdot 10 \cdot 20^2} = \pm\,375\text{ MPa} \quad \text{versus } \pm 185.751\text{ MPa}$$

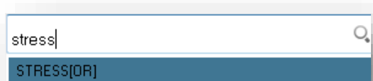
The displacement result of the analysis is very good with an error ~0.5%. However, the stress results look not good with an error superior to 50%.

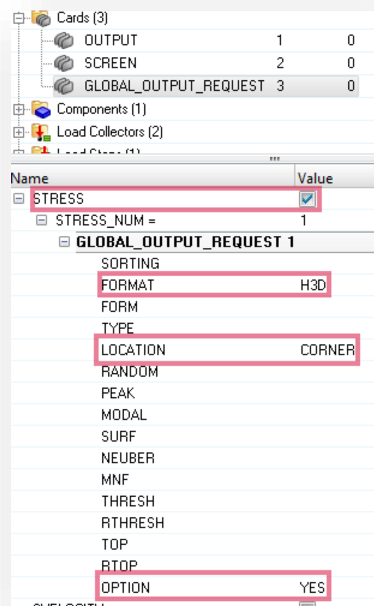
But as in OptiStruct element stresses for shell (and solid) elements are output at the element center only, you may not compare OptiStruct's stress result with s at ± h/2, but ± h/4:

$$\sigma = \frac{M_{\max}\left(\pm\frac{h}{4}\right)}{I} = \pm\frac{3\,F\,l}{4\,th^2} = \pm\frac{3\cdot1000\cdot1000}{4\cdot10\cdot20^2} \quad = \pm187.5\ \mathrm{MPa}$$ versus ±185.751 MPa

Step 13: Add the global output request `STRESS(H3D,CORNER)=YES`

Tip: Use HyperMesh's Quick Access Tool (`Crtl+f`) with `STRESS` to add this control card/global output request `STRESS(H3D,CORNER)=YES`



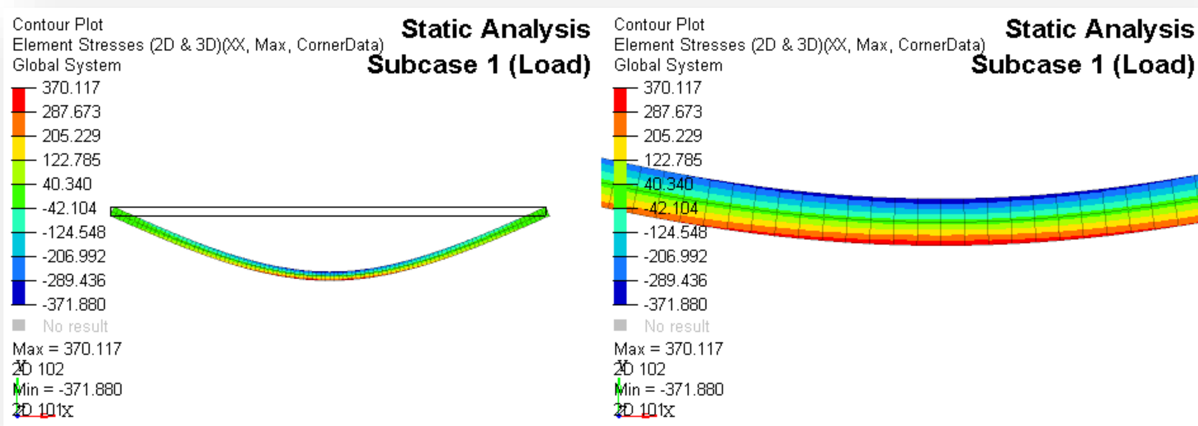


Step 14: Rerun the analysis with OptiStruct

Step 15: Review the stresses in HyperView and check

- **max. displacement in y-direction max. stress in (global) xx-direction**

The maximum stress is ±370 MPa versus ±375 MPa (theoretical result). Do not forget to set activate use corner data.

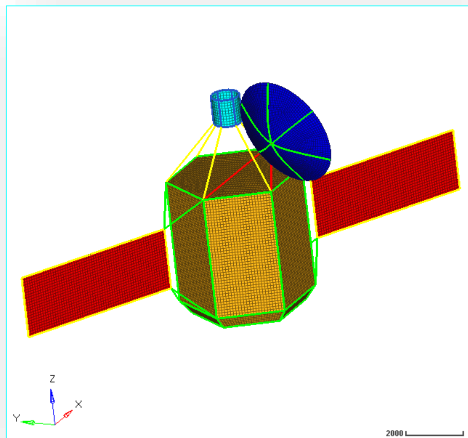


Chapter 3

Inertia Relief Analysis

Exercise 3a: Satellite Inertia Relief Analysis

This exercise runs an inertia relief load case on a simple satellite. This is a test made with aerospace structures that will need to support inertia loads. The objective of this kind of test is to verify if the structure is strong enough to support these loads without a static failure.



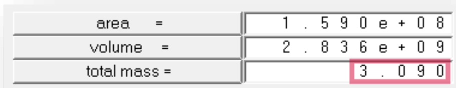
File Name and Location

…\STUDENT-EXERCISE\3a_Satellite\satellite.hm

Step 1: Open the model in HyperMesh Desktop

Step 2: Review the model and check total mass

Tip: Total mass: 3.090 t



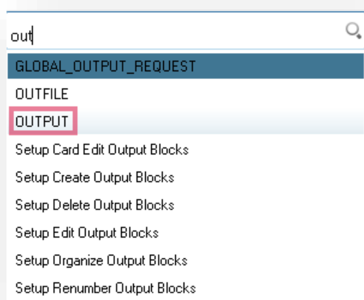
Step 3: Set common `OUTPUT` requests

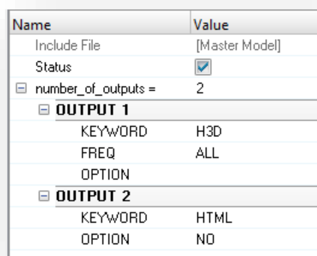
Use HyperMesh's Quick Access Tool (`Crtl+f`) to add control cards

`SCREEN OUT`

`OUTPUT,H3D,ALL`

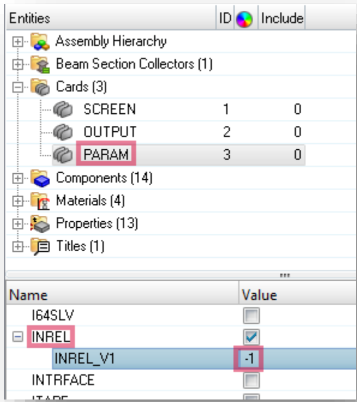
`OUTPUT,HTML,,NO`





Step 4: Set control card `PARAM,INREL` to `-1`

Tip: Use HyperMesh's Quick Access Tool (`Crtl+f`) to set control card `PARAM,INREL` to `-1`



Step 5: Create a load collector `Support` (no card image) with the following `SUPORT1` load type constraints representing the fictitious support

- **Node 2: DOF `2`**
- **Node 3: DOFs `1-3`**
- **Node 4: DOFs `1-2`**

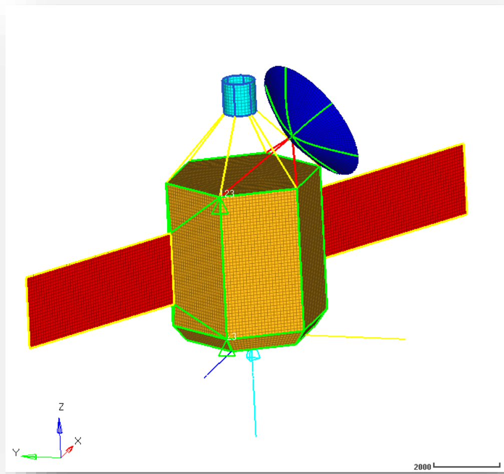
Tip: Switch the **Load Type** for constraint in HyperMesh from `SPC` to `SUPORT1` before creating the constraints

Step 6: Create the following load collectors (no card image) and forces on Node 1 of the model

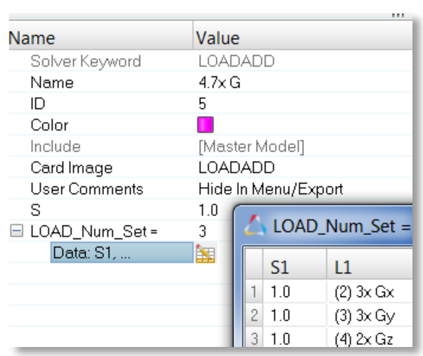
- **`3x Gx`: force components {92700,0,0} (equivalent to three times gravity in x direction)**

- **`3x Gy`: force components {0,92700,0} (equivalent to three times gravity in y direction)**

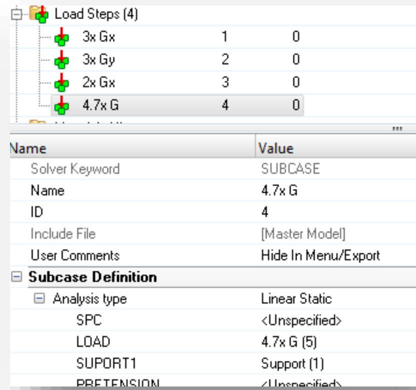
- **`2x Gz`: force components {0,0,61800} (equivalent to two times gravity in z direction)**



Step 7: Create a following load collector `4.7x G` with card image `LOADADD` and select all three above created load collectors with scale factor `1.0`



Step 8: Create four Linear Static load cases using `Support` as the `SUPORT1` entry for each of the four load collectors created before



Step 9: Run the analysis with OptiStruct

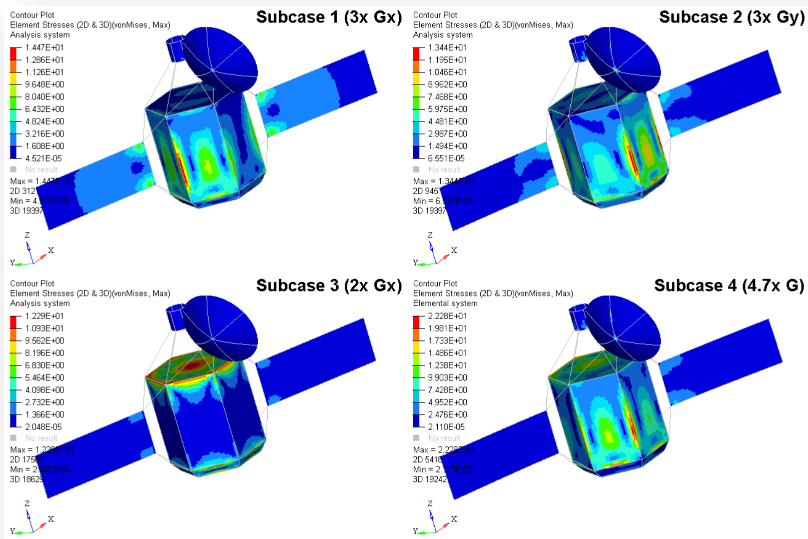
Step 10: Review the `.out` file wrt warnings and errors

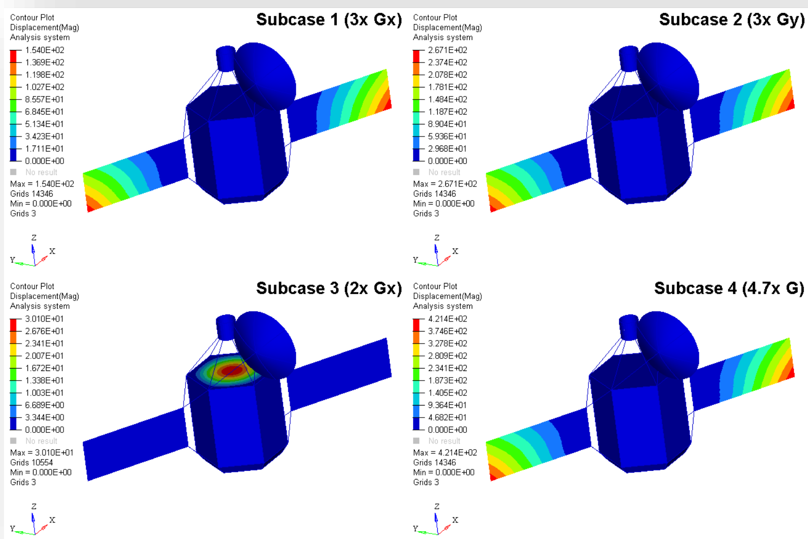
Step 11: Review the results in HyperView and check if

- **Max. relative displacement < 500 mm**

- **Max. von Mises stress < 70 MPa for 2D modeled components**

Tip: Max. relative displacement = 421.4 mm (load case 4.7x G) < 500 mm and Max. von Mises stress = 22.3 MPa (load case 4.7x G) < 70 MPa


Subcase 1 (3x Gx)
Subcase 2 (3x Gy)
Subcase 3 (2x Gx)
Subcase 4 (4.7x G)


Subcase 1 (3x Gx)
Subcase 2 (3x Gy)
Subcase 3 (2x Gx)
Subcase 4 (4.7x G)

Chapter 4

Modal Analysis

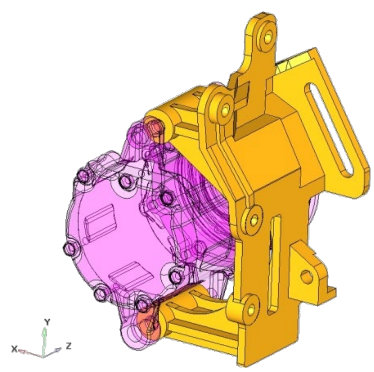
Exercise 4a: Compressor Bracket Modal Analysis

This exercise runs a modal analysis on a compressor system. This is very common problem for an engine designer, who needs to find the best way to link the compressor with the engine. To make this system viable the vibration produced by the engine can't have resonance with the compressor system, and then the key to the project is to develop a bracket that makes the frequencies higher than excitations. Suppose that our 4-cycle engine can work up to 8000 RPM, and then the excitations from the second order (2 explosions per cycle) are up to ~266 Hz.

Then the objective of this project is to have a Bracket with the first frequency higher than 350 Hz.

In this exercise, you will learn how to:

- Determine if a FEA model is well defined
- Understand how well the modal results represent the model



File Name and Location

```
…\STUDENT-
EXERCISE\4a_Bracket_Compressor\bracket_compressor_2nd.hm
```

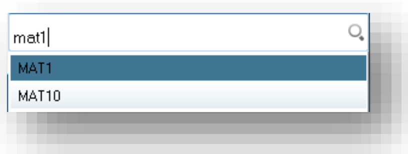
Step 1: Open the model in HyperMesh Desktop

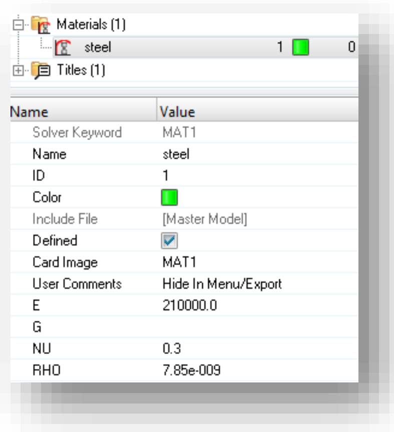
Step 2: Review the model

Tip: The length system is reasonable to be millimeter. There is no representation for the bolts and the compressor. To do this kind of simplification the analyst needs to have know-how about the system behavior, in general we can assume that the bolt is strong enough to not change the modal result. But the compressor geometry needs to be studied before any simplification. In this case we will add a mass element to represent the compressor.

Step 3: Create a `MAT1` material `steel` for steel with the properties: Young's modulus `210000` MPa, density `7.85E-9` t/mm3, Poisson's ratio `0.3`

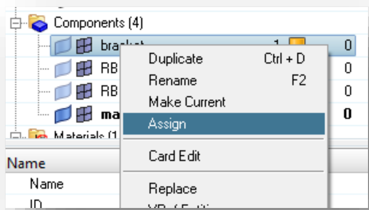
Tip: Use HyperMesh's Quick Access Tool (`Crtl+f`) to add according `MAT1` material card

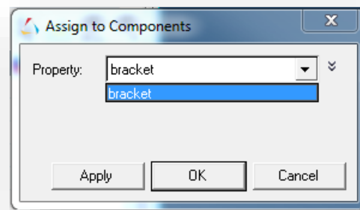


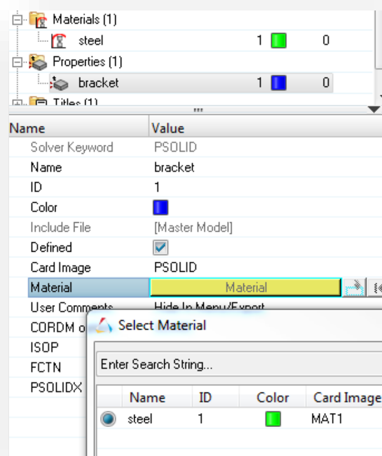


Step 4: Create a `PSOLID` property `bracket` referencing material `steel` and assign it to the `bracket` component

Tip: Use HyperMesh's Quick Access Tool (`Crtl+f`) to create property `PSOLID` and assign it to the `bracket` component with the right mouse menu

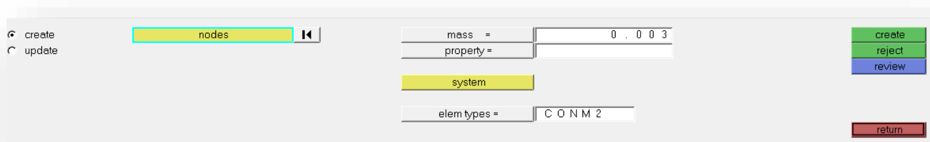


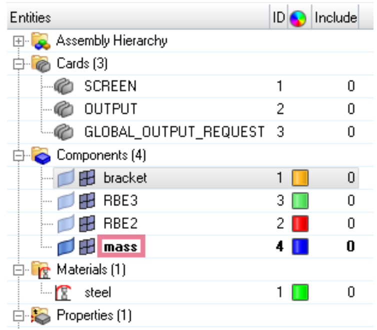




Step 5: Create a mass element at node 6 (dependent node of the RBE3 element) with value `0.003`

Tip: Check that `mass` is the active collector (marked in bold in the **Model Browser**), and create a `CONM2` element (Concentrated Mass Element Connection, Rigid Body Form). You can reach the panel with HM's Quick Access Tool or with the pull-down menu ***Mesh > Create > Masses***





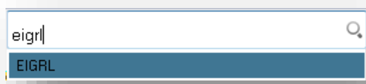
Note that a RBE3 element is used to link the mass element to the bracket. An RBE2 would include a rigid condition between the compressor links that doesn't exist. As optional exercise you can rerun the model with an RBE2 instead and compare the results.

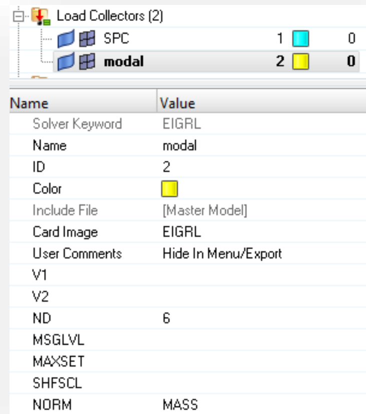
Step 6: Create a load collector SPC (no card image) and with constraints to all five bolt locations RBE2 independent nodes (1-5) for DOF 1-3 each

Tip: Use HyperMesh's Quick Access Tool (Crtl+f) with SPC Note that with these five constraints (DOF 1-3) the engine all is considered to be rigid. It might be that the engine all is thin on the region where the bracket is fixed, and it can be very important on the modal behavior. Here the analyst needs to study the region to make the right assumption

Step 7: Create a load collector modal (card image EIGRL) and set the number of desired roots (ND) to 6

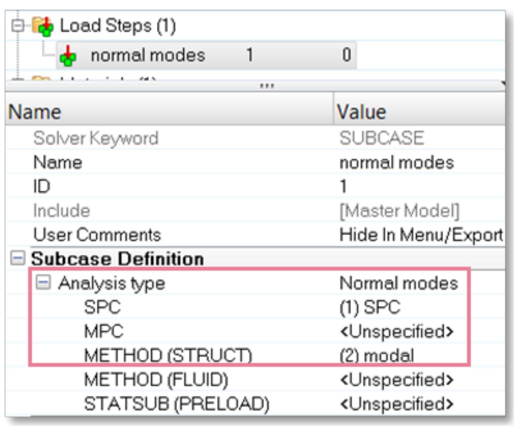
Tip: Use HyperMesh's Quick Access Tool (Crtl+f) with EIGRL and set ND to 6





Step 8: Create a load step `normal modes` and reference the two local collectors accordingly

Tip: Set **Analysis type** to `Normal modes` in order to reduce the number of Subcase Information Entries



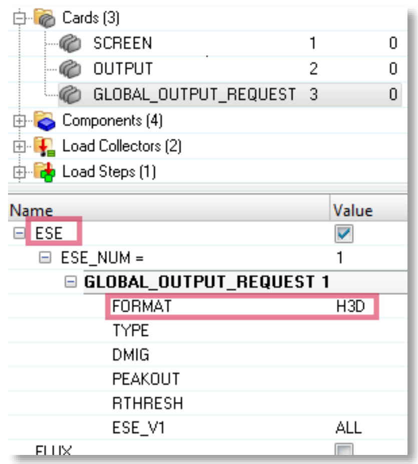
Step 9: Set common control cards requests

Tip: Use HyperMesh's Quick Access Tool (Crtl+f) to add control cards

```
SCREEN OUT
OUTPUT,H3D,ALL
OUTPUT,HTML,,NO
```

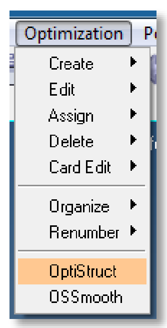
Step 10: Request the strain energy results using global output request `ESE`

Tip: Do the same for `ESE`



Step 11: Run the analysis with OptiStruct

Tip: Run the model in OptiStruct using e.g.the OptiStruct panel via pull-down menu ***Optimization → OptiStruct***



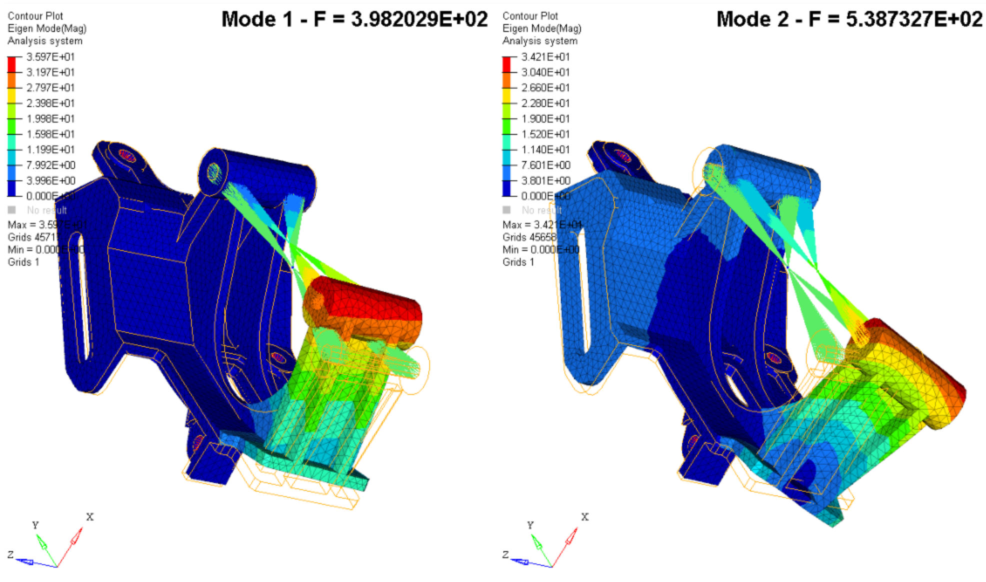
Step 12: Review the `.out` file wrt warnings and errors and check if f1 > 350 Hz

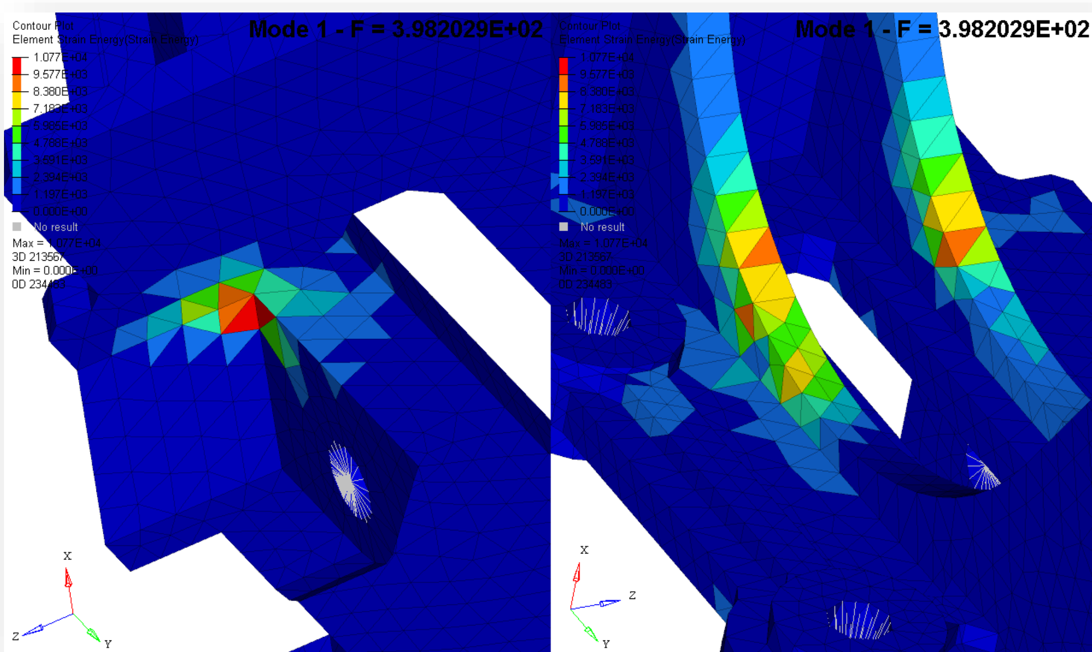
Tip: There are 22 elements that exceeded recommended range (warning) for the element quality check. f1 = 398 Hz > 350 Hz, so the constraint is fulfilled

Subcase	Mode	Frequency	Eigenvalue	Generalized Stiffness	Generalized Mass
1	1	3.982029E+02	6.259916E+06	6.259916E+06	1.000000E+00
1	2	5.387327E+02	1.145794E+07	1.145794E+07	1.000000E+00
1	3	1.142351E+03	5.151795E+07	5.151795E+07	1.000000E+00
1	4	1.540108E+03	9.364018E+07	9.364018E+07	1.000000E+00
1	5	2.053619E+03	1.664943E+08	1.664943E+08	1.000000E+00
1	6	2.363966E+03	2.206186E+08	2.206186E+08	1.000000E+00

Step 13: Review contours of the `mode shapes` and `strain energy` in HyperView

Tip: Screenshot shows mode shapes 1 and 2. Strain energy can give to the analyst a very good indication if the mode is well refined or there is need for a mesh refinement. It works like the stress for a static analysis.



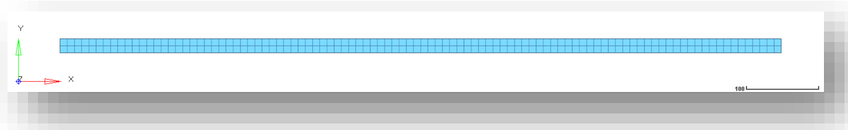


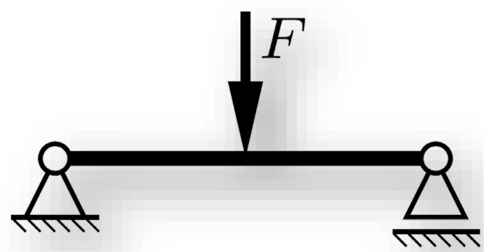
Exercise 4b (optional): Simply Supported Beam Model Analysis

In this exercise, a modal analysis is performed on a simple supported beam from exercise 3c:

- Beam modelled by shell elements, length = 1000 mm, height = 20 mm, width = 10 mm
- Material steel (Young's modulus 210000 MPa, Poisson's ratio 0.33, density 7.85e-9 t/mm3)
- Force of 1000 N in the center of the beam.

The objective is to compare the results of the first three eigen frequencies of the finite element model with the theoretical solution.





File Name and Location

…\STUDENT-EXERCISE\ 4b_Simple_Beam\beam_modal.hm

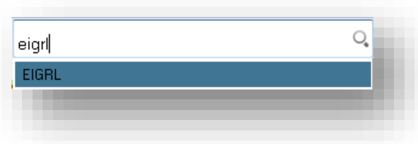
Step 1: Open the model in HyperMesh Desktop with OptiStruct user profile selected

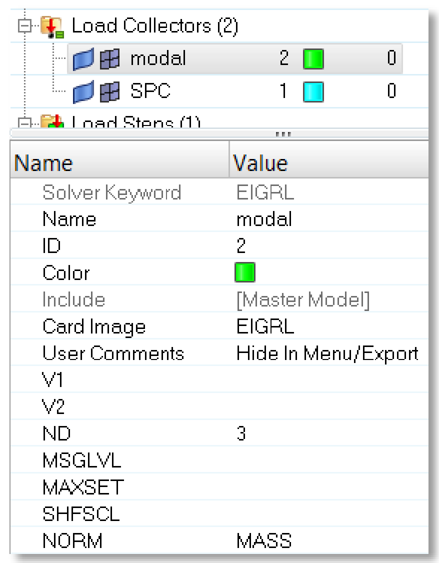
Step 2: Review the model

Step 3: Constrain all nodes additionally in the z-direction in order to get only the shapes in xy-plane

Step 4: Create a load collector `modal` (card image `EIGRL`) and set the number of desired roots (ND) to `3`}

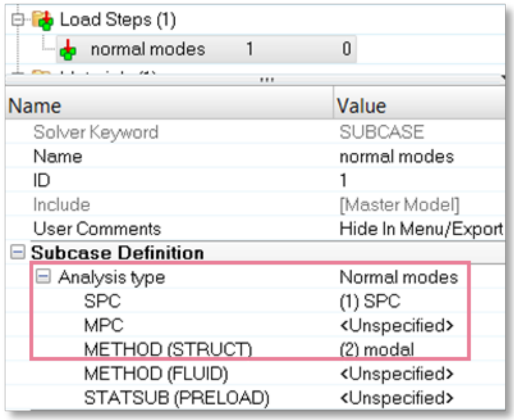
Tip: Use HyperMesh's Quick Access Tool (`Crtl+f`) with `EIGRL` and set ND to `3`





Step 5: Create a load step `MODAL` and reference the two load collectors accordingly

Tip: Set ***Analysis type*** to `Normal modes` in order to reduce the number of Subcase Information Entries



Step 6: Run the analysis with OptiStruct

Step 7: Review the `.out` file wrt warnings and errors and check f1 , f2 and f3

Tip: from the `.out` file:

Subcase	Mode	Frequency
1	1	4.682963E+01
1	2	1.863840E+02
1	3	4.155289E+02

Step 8: Calculate the theoretical results for f1 , f2 and f3

Tip: The equations for simply supported beam shown below

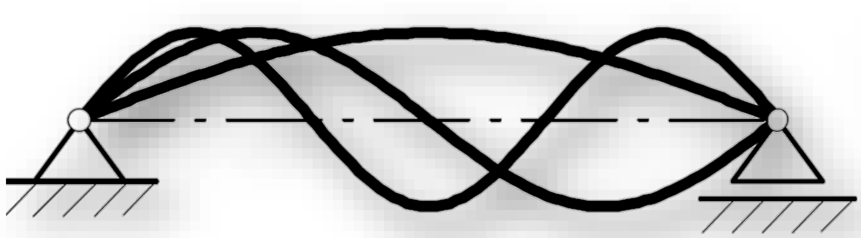
$f_j = \frac{j^2\,\pi}{2}\sqrt{\frac{EI}{\rho\, t\, h\, l^4}} = \frac{j^2\,\pi}{2}\sqrt{\frac{E\,h^2}{12\,\rho\, l^4}} = \frac{j^2\,h\,\pi}{4\,l^2}\sqrt{\frac{E}{3\,\rho}}$ for a simply supported beam

$f_j = \frac{j^2\cdot 20\,\pi}{4\cdot 10000^2}\sqrt{\frac{210000}{3\,x\,0.00000000785}}$ Hz $= 46.9\,j^2$ Hz for this example

f_1=46.9 Hz versus 46.8 Hz

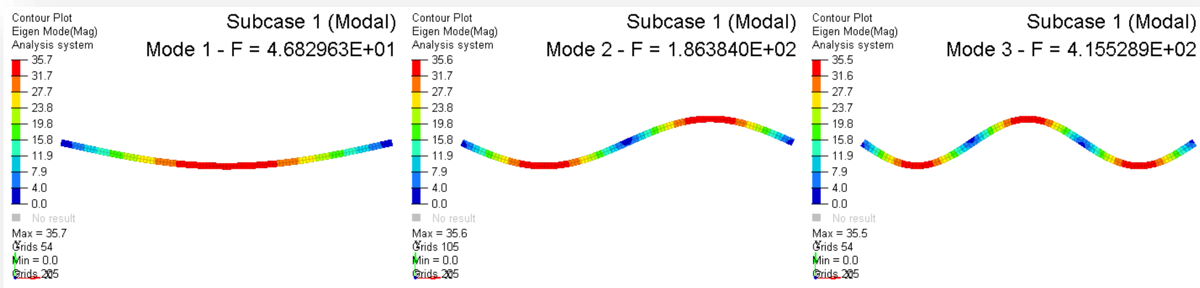
f_2=187.6 Hz versus 186.4 Hz

f_3=422.2 Hz versus 415.5 Hz



Step 9: Review contours of the three mode shapes HyperView

Tip: In the contour plots the deformed shapes are scaled by 2



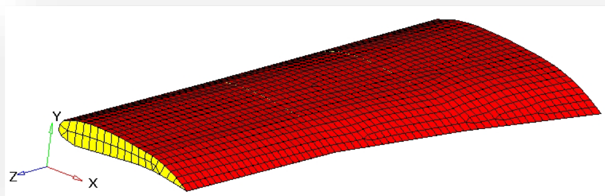
Chapter 5

Linear Buckling Analysis

Exercise 5a: Wing Linear Buckling Analysis

This exercise runs a linear buckling analysis on a simple aircraft wing. This is a typical problem in aerospace structures that need to be very light and consequently become slender. Because the structure has a high slenderness ratio, the buckling failure verification becomes necessary.

The objective of this project is to determine if the 3 static load cases applied to the wing will cause failure, the positive buckling factors should be higher than 1.5.



File Name and Location

```
…\STUDENT-EXERCISE\5a_Wing\wing.hm
```

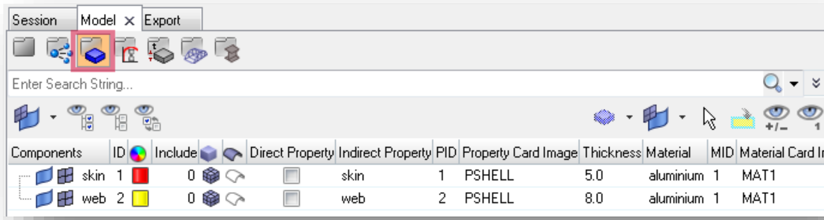
Step 1: Open the model in HyperMesh Desktop

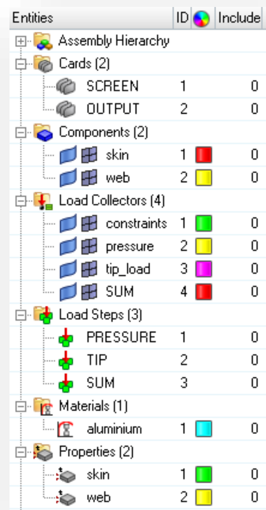
Step 2: Review the model (control cards, components, properties, materials, load collectors & steps)

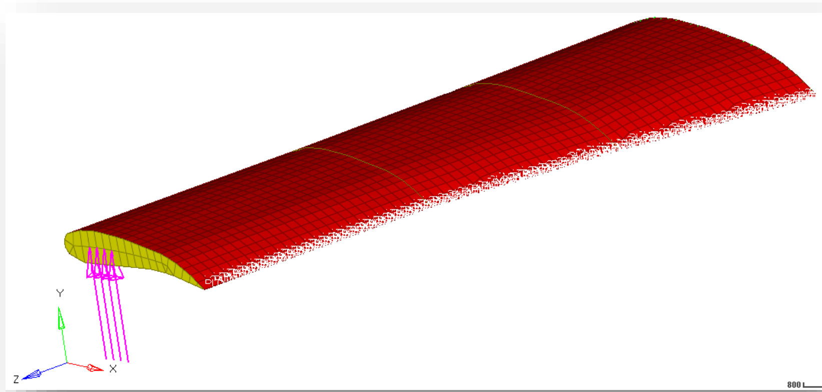
Tip: To review the model wrt components, properties and materials the component view in the model browser is well suited.

Common control cards requests are set:
`SCREEN` and `OUTPUT`

There are three static load steps defined:
(1) pressure on skin
(2) load on tip
(3) combination of both using `LOADADD` card





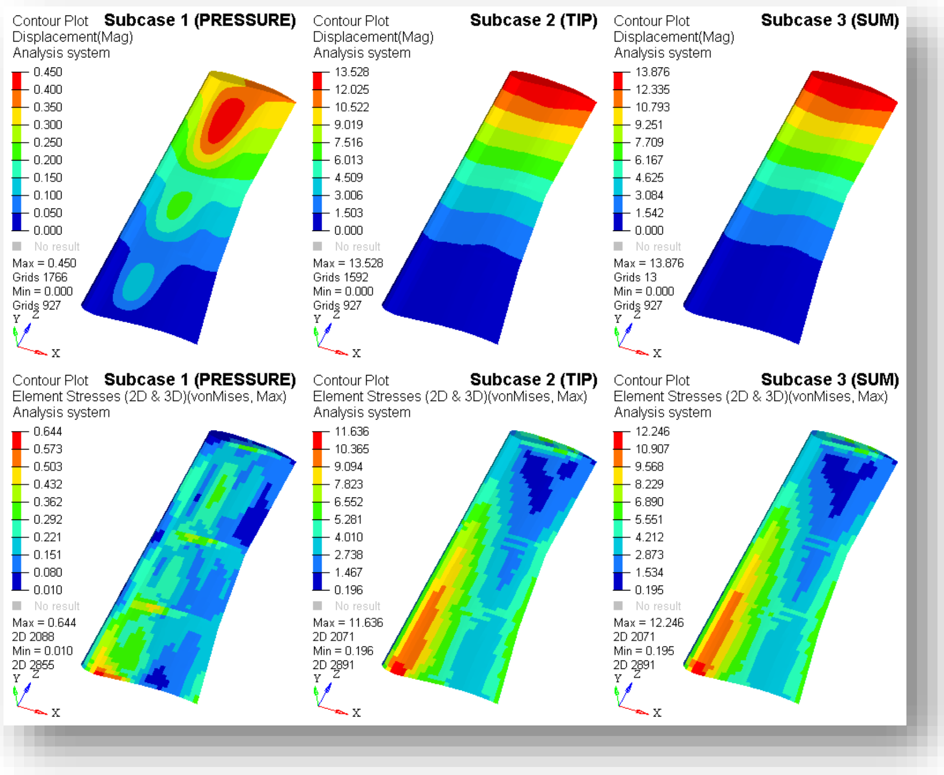


Step 3: Run the static analysis in OptiStruct, review the results in HyperView and check if

- **Max. static displacement < 20 mm for all load cases.**

- **Max. von Mises stress < 70 MPa**

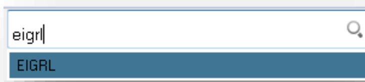
Tip: Max. displacement = 13.9 mm
(load case **SUM**) < 20 mm
Max. von Mises stress = 12.25 MPa
(load case **SUM**) < 70 Mpa

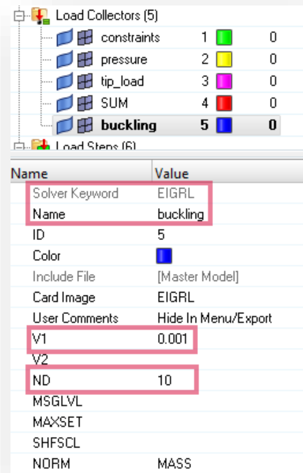


Step 4: Create a load collector `buckling` (card image `EIGRL`) and set the number of desired roots (ND) to `10` and V1 to `0.001`

Tip: Use HyperMesh's Quick Access Tool (`Crtl+f`) to create an `EIGRL` card.

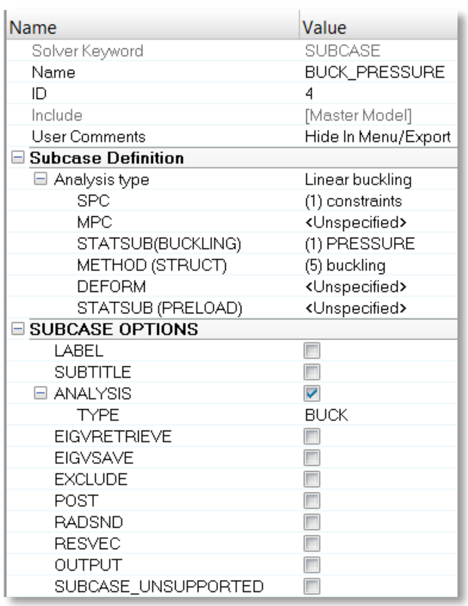
Note that with `0.001` as V1 negative buckling factors (i.e. load with opposite direction) are not considered.

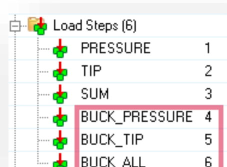


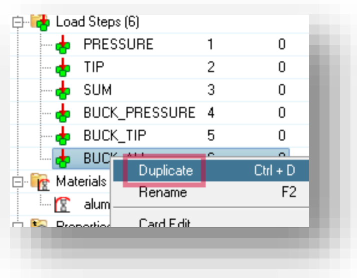


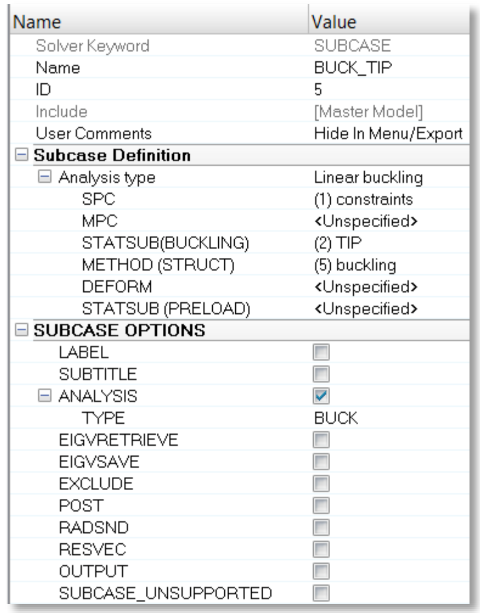
Step 5: Create the buckling load steps for each static load step

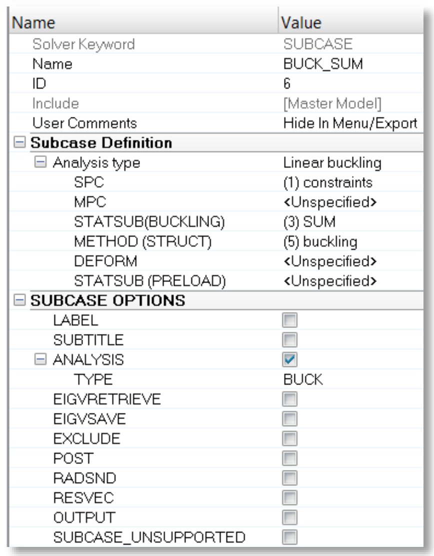
Tip: As the three new linear buckling load steps only differ by `STATSUB` entry, it is easier to create the first and use duplicate functionality for the others






Load Steps (6)
PRESSURE 1 0
TIP 2 0
SUM 3 0
BUCK_PRESSURE 4 0
BUCK_TIP 5 0
Duplicate Ctrl + D
Rename F2
Materials
alum


Name
Value
Solver Keyword
SUBCASE
Name
BUCK_TIP
ID
5
Include
[Master Model]
User Comments
Hide In Menu/Export
Subcase Definition
Analysis type
Linear buckling
SPC
(1) constraints
MPC
<Unspecified>
STATSUB(BUCKLING)
(2) TIP
METHOD (STRUCT)
(5) buckling
DEFORM
<Unspecified>
STATSUB (PRELOAD)
<Unspecified>
SUBCASE OPTIONS
LABEL
SUBTITLE
ANALYSIS
TYPE
BUCK
EIGVRETRIEVE
EIGVSAVE
EXCLUDE
POST
RADSND
RESVEC
OUTPUT
SUBCASE_UNSUPPORTED


Name
Value
Solver Keyword
SUBCASE
Name
BUCK_SUM
ID
6
Include
[Master Model]
User Comments
Hide In Menu/Export
Subcase Definition
Analysis type
Linear buckling
SPC
(1) constraints
MPC
<Unspecified>
STATSUB(BUCKLING)
(3) SUM
METHOD (STRUCT)
(5) buckling
DEFORM
<Unspecified>
STATSUB (PRELOAD)
<Unspecified>
SUBCASE OPTIONS
LABEL
SUBTITLE
ANALYSIS
TYPE
BUCK
EIGVRETRIEVE
EIGVSAVE
EXCLUDE
POST
RADSND
RESVEC
OUTPUT
SUBCASE_UNSUPPORTED

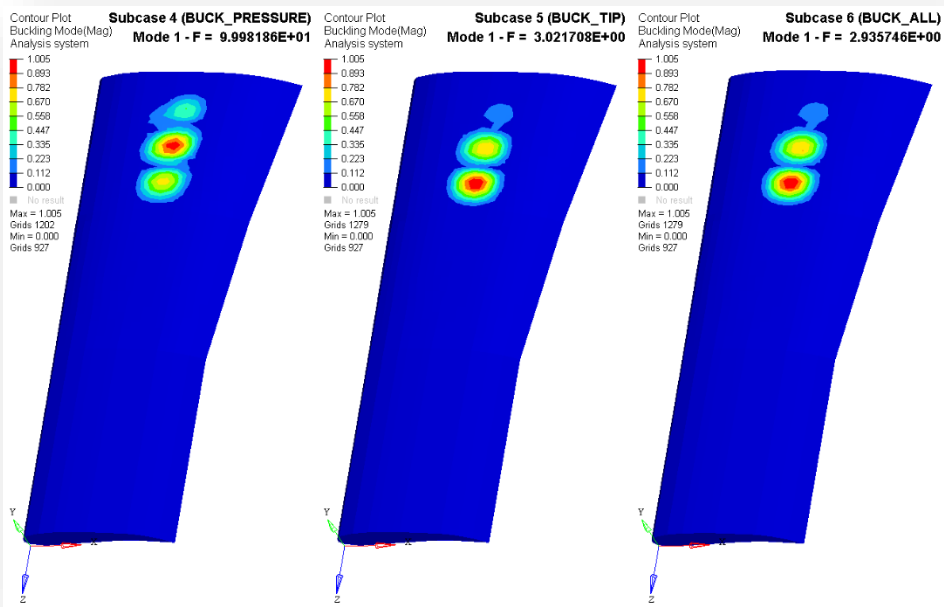
Step 6: Run the analysis with OptiStruct

Step 7: Review the `.out` file wrt warnings and errors and check if the lowest λ for each buckling subcase is > 1.5

Tip: buckling factors λ_4 = 10.0, λ_5 = 3.0, λ_6 = 2.9, so λ_i > 1.5

Step 8: Review contours of the buckling modes in HyperView

Tip: Buckling modes 1 for load steps 4-6 shown below



Chapter 6

Thermal Stress Steady State Analysis

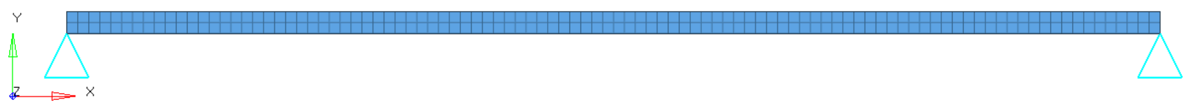
Exercise 6a: Thermal Stress Analysis of a Beam

This exercise runs a thermal stress analysis on a simple supported beam modeled with shell elements, known from exercises 3b and 5b.

The objective is to calculate the deformed shape and maximum displacement due to the SPCs and the following temperature load

Reference temperature of 20

Lower/middle/upper row of nodes has a temperature of 20/30/40



File Name and Location

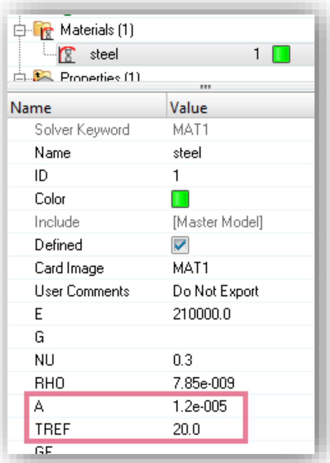
```
…\STUDENT-EXERCISE\6a_Simple_Beam\beam.hm
```

Step 1: Open the model in HyperMesh Desktop

Step 2: Review the model

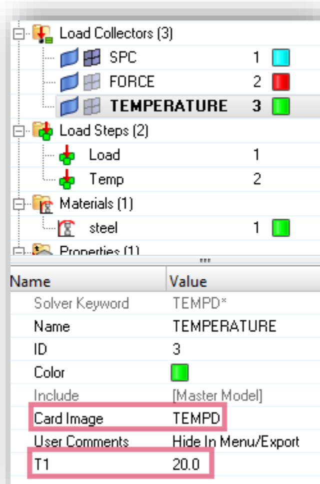
Step 3: Add the thermal expansion coefficient and a reference temperature to the material card

Tip: As the material is steel, take `1.2e-5` as `A`. `TREF` is given as `20`

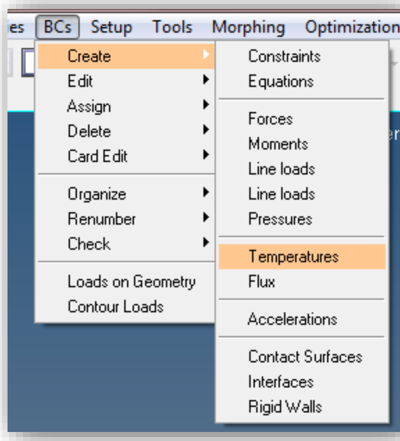


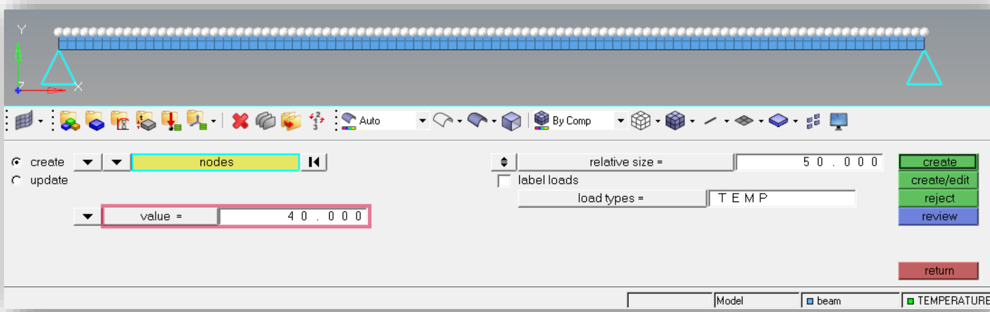
Step 4: Create a load collector `TEMPERATURE` (card image `TEMPD`) with `T = 20` and create `TEMP` loads with a value of `30` and `40` for the middle and upper row of nodes respectively.

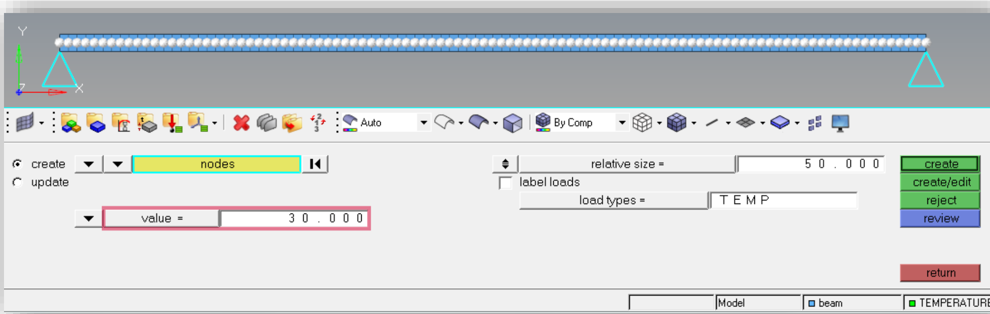
Tip: Use HyperMesh's Quick Access Tool (`Crtl+f`) to create a `TEMPD` card



Due to `TEMPD` card with `T = 20`, there is no need to create `TEMP` loads for the lower row of nodes

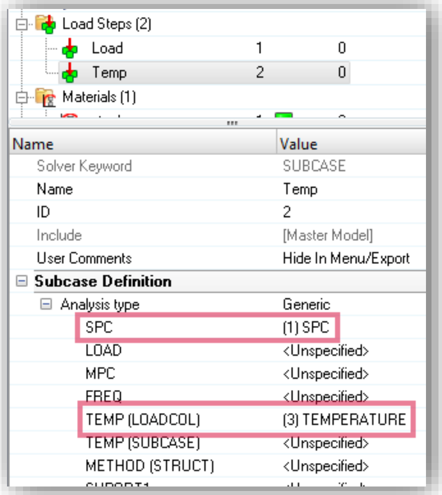






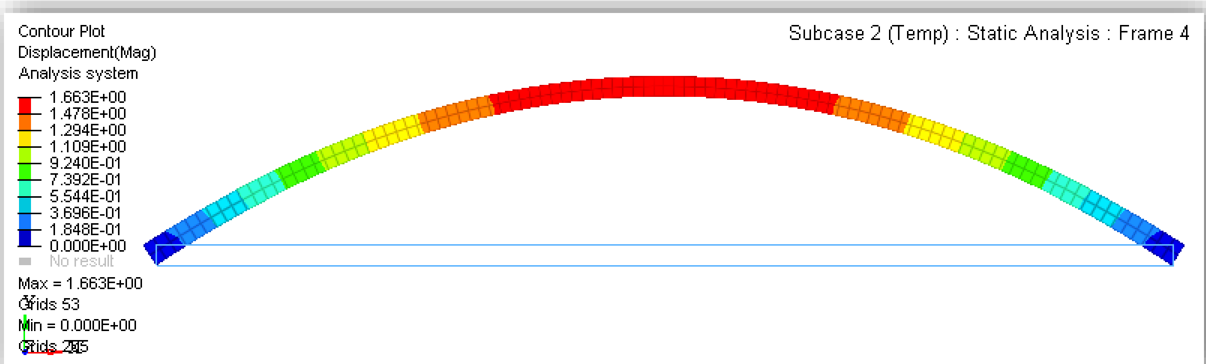
Step 5: Create a load step `Temp` with SPC referencing `SPC` load collector and TEMP(LOADCOL) referencing new `TEMPERATURE` load collector

Tip: Use Generic as Analysis type.



Step 6: Run the analysis with OptiStruct, review the results in HyperView and check the maximum displacement

Tip: Maximum displacement is 1.663 (deformed shape inflated by 100)



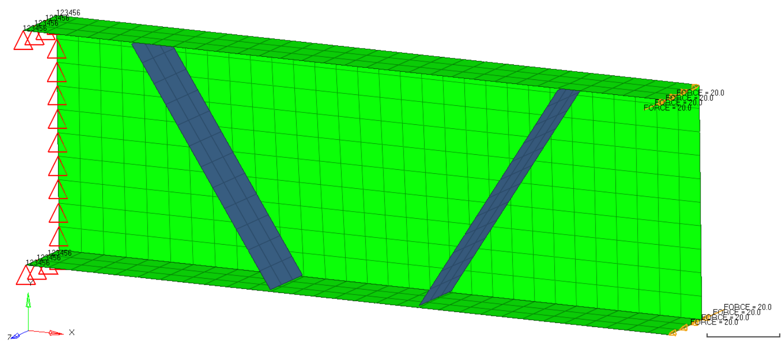
Chapter 7

Advanced Topics

Exercise 7a (optional): Static Analysis using Freeze Contact

This exercise runs a linear static analysis on a simple C beam modeled with shell elements. Additional ribs will be added and connected to the beam using freeze contact.

The objective of this project is to determine without any mesh changes if the additional ribs will reduce the maximum total displacement to below 0.38.



File Name and Location

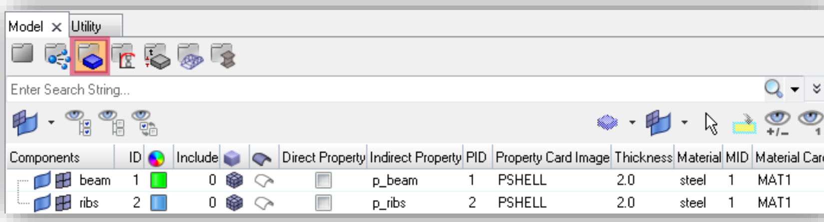
`…\STUDENT-EXERCISE\7a_Beam_Rib\beam_rib.hm`

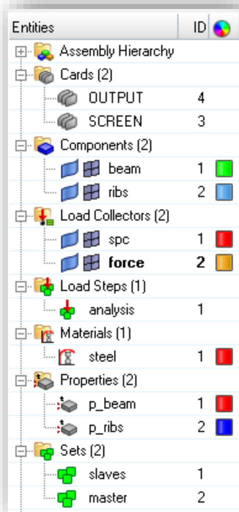
Step 1: Open the model in HyperMesh Desktop

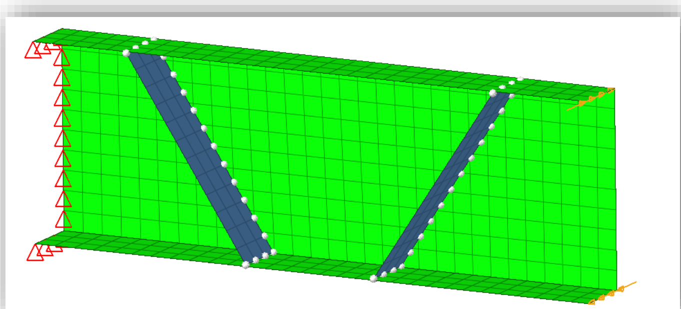
Step 2: Review the model (control cards, components, properties, materials, sets, load collectors & step)

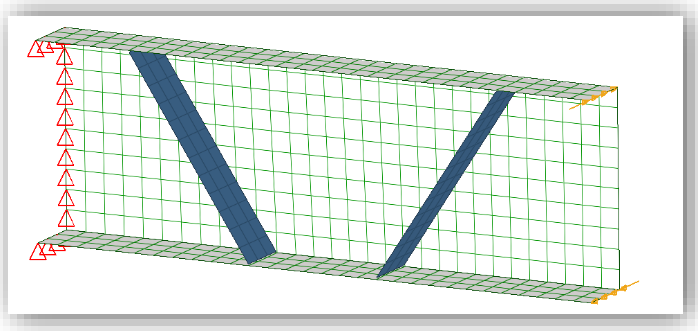
Tip: To review the model wrt components, properties and materials the component view in the **Model Browser** is well suited.

Common control cards requests are set:`SCREEN` and `OUTPUT`. There is one static load step defined: `analysis` There are two sets defined: `slaves` as grid set (ID 1) `master` as element set (ID 2)





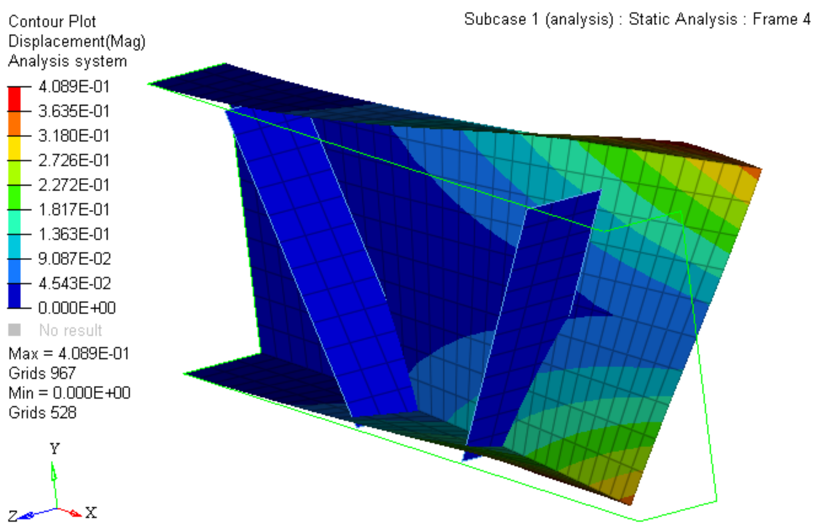




Step 3: Run the static analysis in OptiStruct, review the results in HyperView and check maximum static displacement

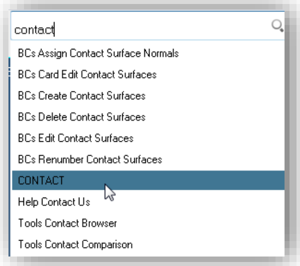
Tip: Max. displacement = 0.4089 (deformed shape inflated by 100)

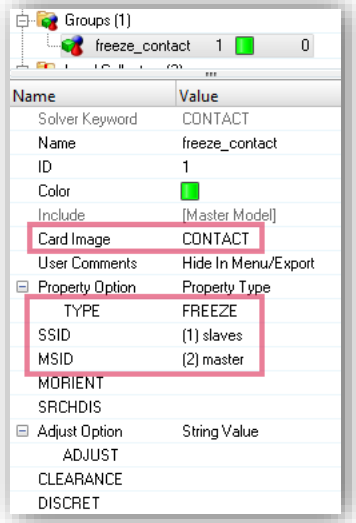
The two ribs are not connected to the beam



Step 4: Create a group collector `freeze_contact` (card image `CONTACT`) with TYPE = `FREEZE`, SSID = slaves (grid) set (ID `1`) and MSID master (element) set (ID `2`)

Tip: Use HyperMesh's Quick Access Tool (`Crtl+f`) to create a `CONTACT` card

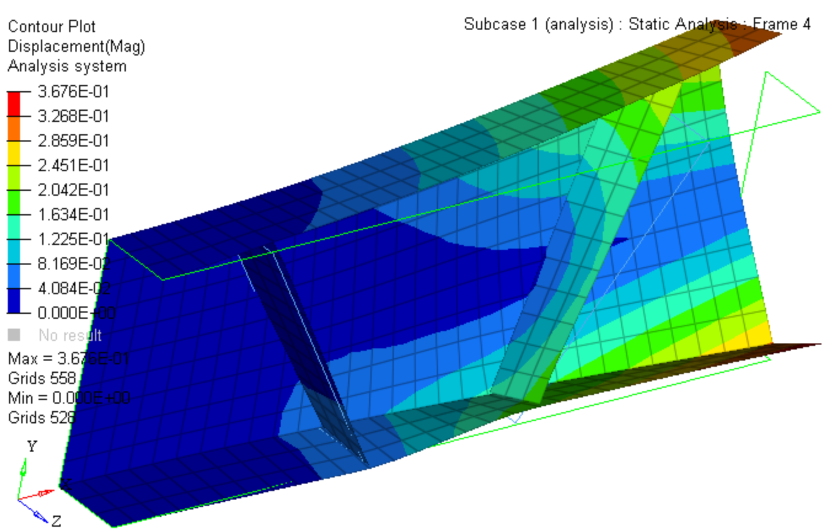




Step 5: Rerun the analysis with OptiStruct, review the results in HyperView and check if maximum static displacement is below 0.38

Tip: Max. displacement = 0.3676 < 0.38 (deformed shape inflated by 100)

The two ribs are connect to the beam

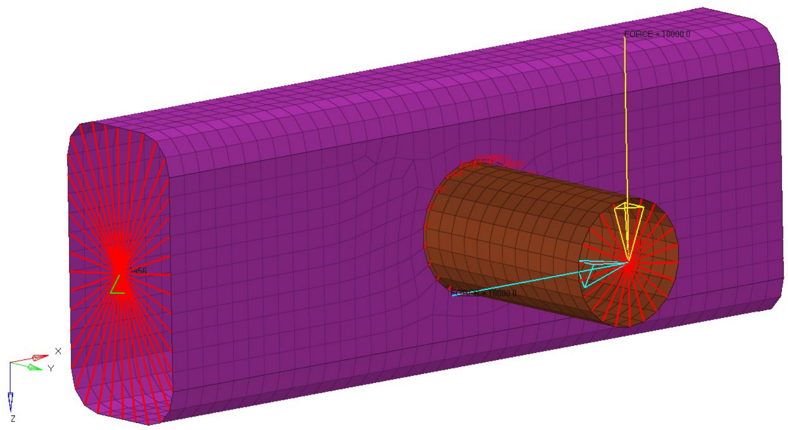


Chapter 8

Optimization in Linear Analysis

Exercise 8a: Size Optimization of a Rail Joint

The purpose of this exercise is to set up a property parameter optimization on an automobile rail joint modeled with shell elements. The deflection at the end of the tubular cross-member should be limited. The optimal solution would use as little material as possible



Problem Definition

Design Variables

- *Property parameter thickness of tube1 and tube2 with initial values of 1.0, lower bounds of 0.1 and upper ones of 5.0*

Responses

- *volume*
- *U_x at load application point for load step LOAD_X*
- *U_z at load application point for load step LOAD_Z*

Objective: *min volume*

Constraints

- *$U_x < 0.9$*
- *$U_z < 1.6$*

File Name and Location

…\STUDENT-EXERCISE\8a_Rail_Joint\joint_size.fem

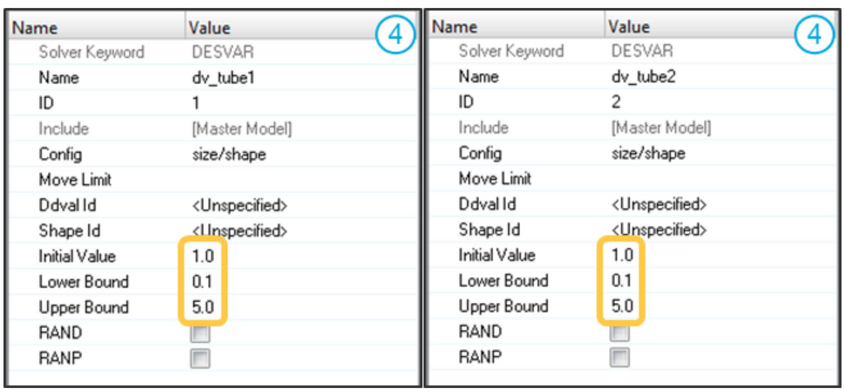
Step 1: Import the model in HyperMesh Desktop with OptiStruct user profile selected

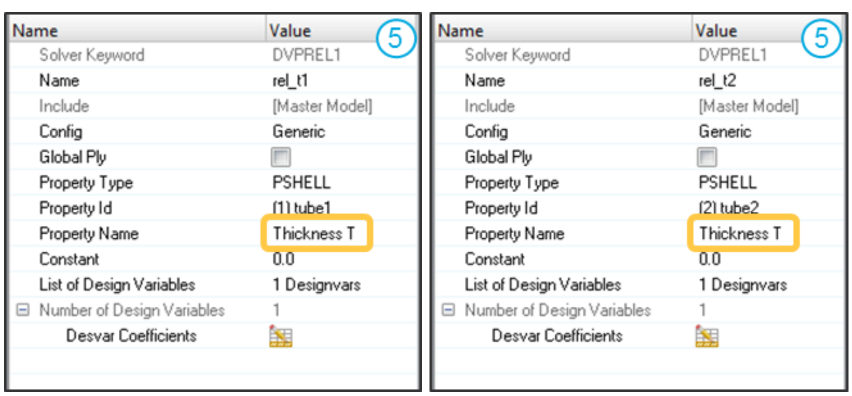
Step 2: Review the model and check the constraints and load steps

Step 3: Run an analysis with OptiStruct and review displacements U_x and U_z according to the later constraints:
$U_x = 1.273$ and $U_z = 2.144$

Step 4: Create two size/parameter design variables with initial values of `1.0`, lower bounds of `0.1` and upper ones of `5.0`, empty (default) move limit and no ddval.

Step 5: Create two generic relationships to link the property parameters thickness to the according design variables





Step 6: Create a volume response

Name	Value
Solver Keyword	DRESP1
Name	r_volume
ID	1
Include	[Master Model]
Response Type	volume
Property	PROP_TOTAL
Property Type	total
Region Identifier	
DREPORT	

6

Step 7: Create a static displacement response in x-direction for the load application point (ID 5555), U_x

Ste 8: Create a static displacement response in z-direction for the load application point (ID 5555), U_y

Name	Value
Solver Keyword	DRESP1
Name	r_ux
ID	2
Include	[Master Model]
Response Type	static displacement
Property	PROP_TOTAL
List Of Nodes	1 Nodes
Region Identifier	
	dof1
COORD	
DREPORT	

7

Name	Value
Solver Keyword	DRESP1
Name	r_uz
ID	3
Include	[Master Model]
Response Type	static displacement
Property	PROP_TOTAL
List Of Nodes	1 Nodes
Region Identifier	
	dof3
COORD	
DREPORT	

8

Step 9: Create a constraint for U_x response to be lower than `0.9` for load step `LOAD_X`

Step 10: Create a constraint for U_z response to be lower than `1.6` for load step `LOAD_Z`

Name	Value
Solver Keyword	DCONSTR
Name	c_ux
ID	1
Include	[Master Model]
Lower Bound	
Upper Bound	0.9
Response	(2) r_ux
List of Loadsteps	1 Loadsteps
PROB	

9

Name	Value
Solver Keyword	DCONSTR
Name	c_uz
ID	2
Include	[Master Model]
Lower Bound	
Upper Bound	1.6
Response	(3) r_uz
List of Loadsteps	1 Loadsteps
PROB	

10

Step 11: Define the objective function to minimize volume response

Name	Value
Solver Keyword	DESOBJ(MIN)
Name	objective
ID	1
Include	[Master Model]
Objective Type	Minimize
Response Id	(1) r_volume

11

Step 12: Export and review `.fem` file wrt optimization cards

12

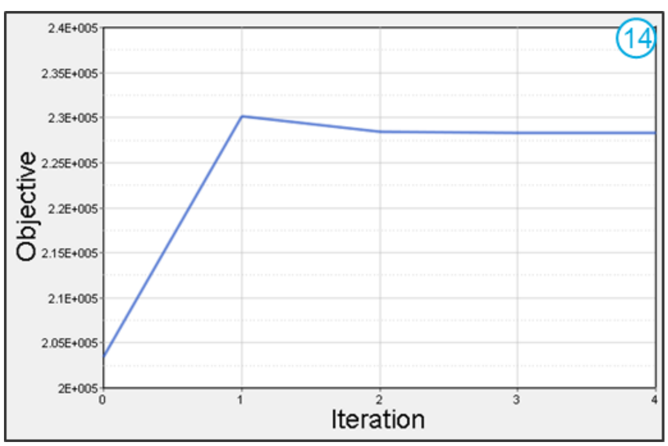
```
DESOBJ(MIN)=1
SUBCASE       1
  LABEL Force_X
  SPC =         1
  LOAD =         2
  DESSUB =        3
SUBCASE       2
  LABEL Force_Z
  SPC =         1
  LOAD =         3
  DESSUB =        4
$
BEGIN BULK
DESVAR         1dv_tube11.0     0.1     5.0
DESVAR         2dv_tube21.0     0.1     5.0
DVPREL1 1       PSHELL          1       4                0.0
+       1       1.0
DVPREL1 2       PSHELL          2       4                0.0
+       2       1.0
DRESP1  1       r_volumeVOLUME
DRESP1  2       r_ux    DISP                           TX            5555
DRESP1  3       r_uz    DISP                           TZ            5555
DCONSTR        1       2        0.9
DCONSTR        2       3        1.6
DCONADD        3       1
DCONADD        4       2
[…]
PSHELL         1       11.0            1               1        0.0
PSHELL         2       10.8            1               1        0.0
```

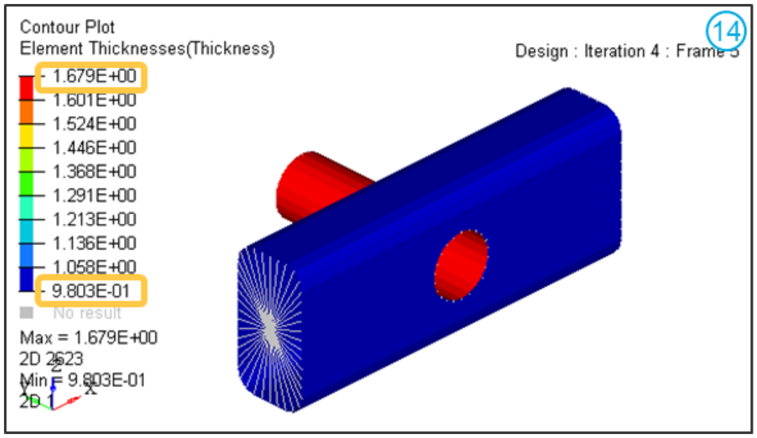
Step 13: Run optimization with OptiStruct, review `.out` file

```
ITERATION    4
  the 2nd satisfied convergence ratio =  2.7998E-05

Objective Function (Minimize VOLUM) =  2.28294E+05   % change =        -0.00
Maximum Constraint Violation %      =  0.92458E-01
Volume                              = 2.28294E+005   Mass     = 0.00000E+000

 Subcase    Compliance      Epsilon
       1  3.534265E+03 -1.106546E-14
       2  8.007397E+03  1.515182E-13

Note : Epsilon = Residual Strain Energy Ratio.


                           RETAINED RESPONSES TABLE
------------------------------------------------------------------------------------
Response Type  Response     Subcase Grid/     DOF/   Response      Objective    Viol.
User-ID           Label     /RANDPS Element/  Comp     Value       Reference/     %
                            /Model  MID/PID/  /Reg                 Constraint
                            +Frqncy Mode No.                         Bound
                            /Times
------------------------------------------------------------------------------------
       1 VOLUM r_volume          --       --  TOTL   2.283E+05 MIN
       2 DISPL r_ux               1     5555    TX   7.069E-01  <  9.000E-01     0.0
       3 DISPL r_uz               2     5555    TZ   1.601E+00  <  1.600E+00     0.1 A
------------------------------------------------------------------------------------
```

```
--------------------------------------------------------
 Design     Design     Lower       Design      Upper
Variable   Variable    Bound       Variable    Bound
   ID        Label
--------------------------------------------------------
       1  dv_tube1   1.000E-01   9.803E-01   5.000E+00
       2  dv_tube2   1.000E-01   1.679E+00   5.000E+00
--------------------------------------------------------



                DESIGNED PROPERTY ITEMS TABLE
-------------------------------------------------------------
 DVPREL1/2  USER-ID  PROP-TYPE  PROP-ID  ITEM-CODE  PROP-VALUE
-------------------------------------------------------------
 DVPREL1          1    PSHELL         1    T          9.803E-01
 DVPREL1          2    PSHELL         2    T          1.679E+00
-------------------------------------------------------------

************************************************************************

 OPTIMIZATION HAS CONVERGED.

 FEASIBLE DESIGN (ALL CONSTRAINTS SATISFIED).
```

Step 14: Review results of `_hist.mvw` and `.mvw` files





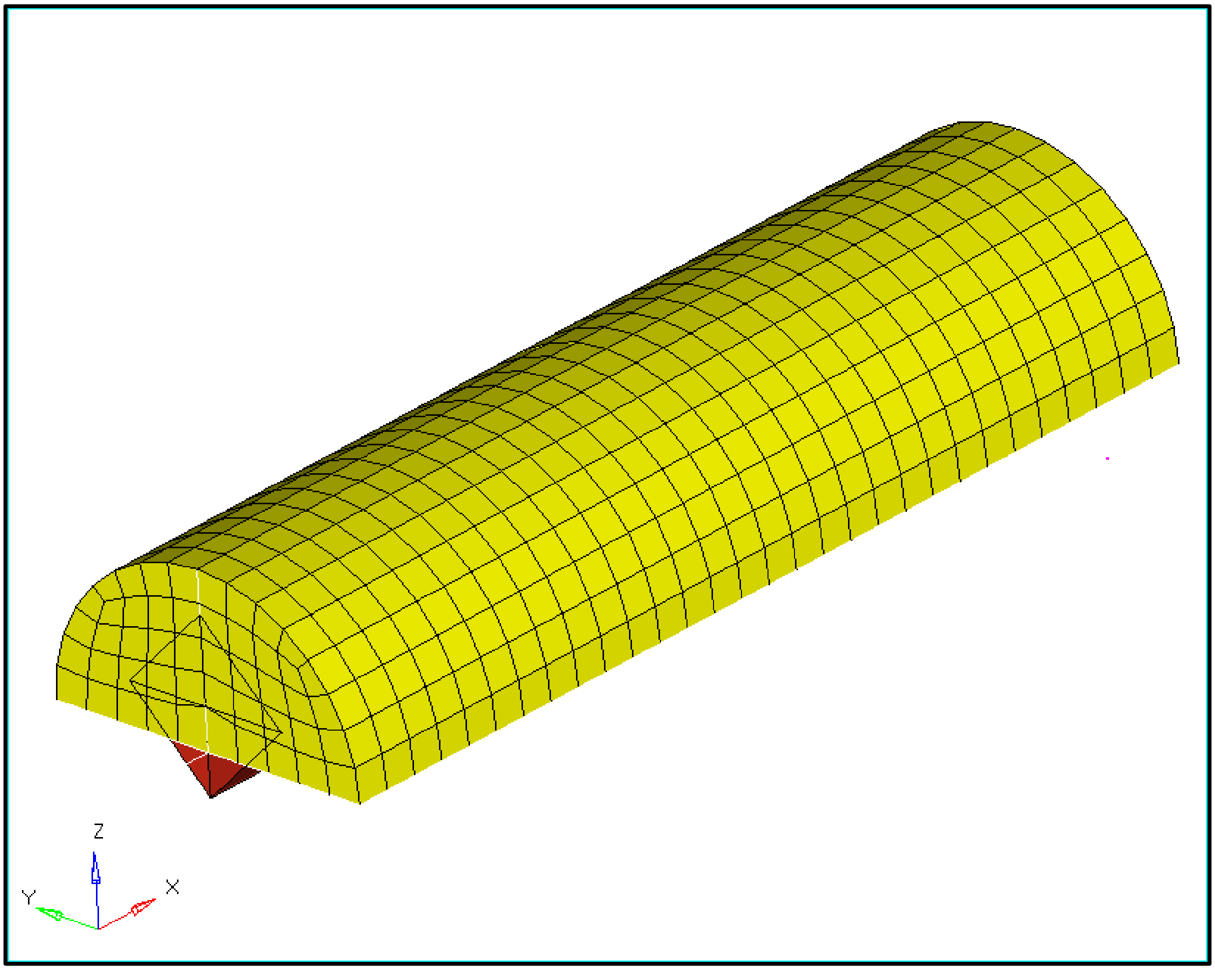
Step 15: Import `.prop` file in HyperMesh Desktop

```
$ ----------------------------------------
$ PROPERTIES AND MATERIALS AT ITERATION
$ ----------------------------------------
PSHELL, 1, 1, .98030871461  1, 1.0, 1,, 0.0
PSHELL, 2, 1, 1.6789640958  1, 1.0, 1,, 0.0
```

Exercise 8b: Size Optimization of a Shredder with Modal Loadcases

A size optimization involves the changing of the properties of either 1D or 2D elements. These properties include area, moments of inertia of the 1D elements, and the thickness of 2D elements and is performed when it is not necessary to remove materials, generate beads or change the shape of the structure. Size optimizations are highly flexible, able to modify cross-sectional properties of one-dimensional elements, material properties, thicknesses of shell and composite elements, and other selected card entities. Values for such entities are changed within given bounds to meet the necessary objective. Properties are linked with design variables (DESVAR) using DVPREL cards.

This exercise goes through the steps involved in defining a size optimization for a model comprised of shell and bar elements. You will update the PBARL property to simulate the properties of the bar elements and then link that to the design variable. The resulting design will have higher frequencies and updated element properties.



Problem Setup

You should copy this file: `shredder.hm`

Step 1: Open the file `shredder.hm` in HyperMesh Desktop and enter the Optimization View of the Model Browser

Step 2: Create the size design variables for the shredder components

Name	Value
Solver Keyword	DESVAR
Name	cover_thk
ID	1
Include	[Master Model]
Config	size/shape
Move Limit	
Ddval Id	<Unspecified>
Shape Id	<Unspecified>
Initial Value	3.0
Lower Bound	0.5
Upper Bound	8.0
RAND	☐
RANP	☐

Name	Value
Solver Keyword	DESVAR
Name	hat_height
ID	2
Include	[Master Model]
Config	size/shape
Move Limit	
Ddval Id	<Unspecified>
Shape Id	<Unspecified>
Initial Value	100.0
Lower Bound	75.0
Upper Bound	125.0
RAND	☐
RANP	☐

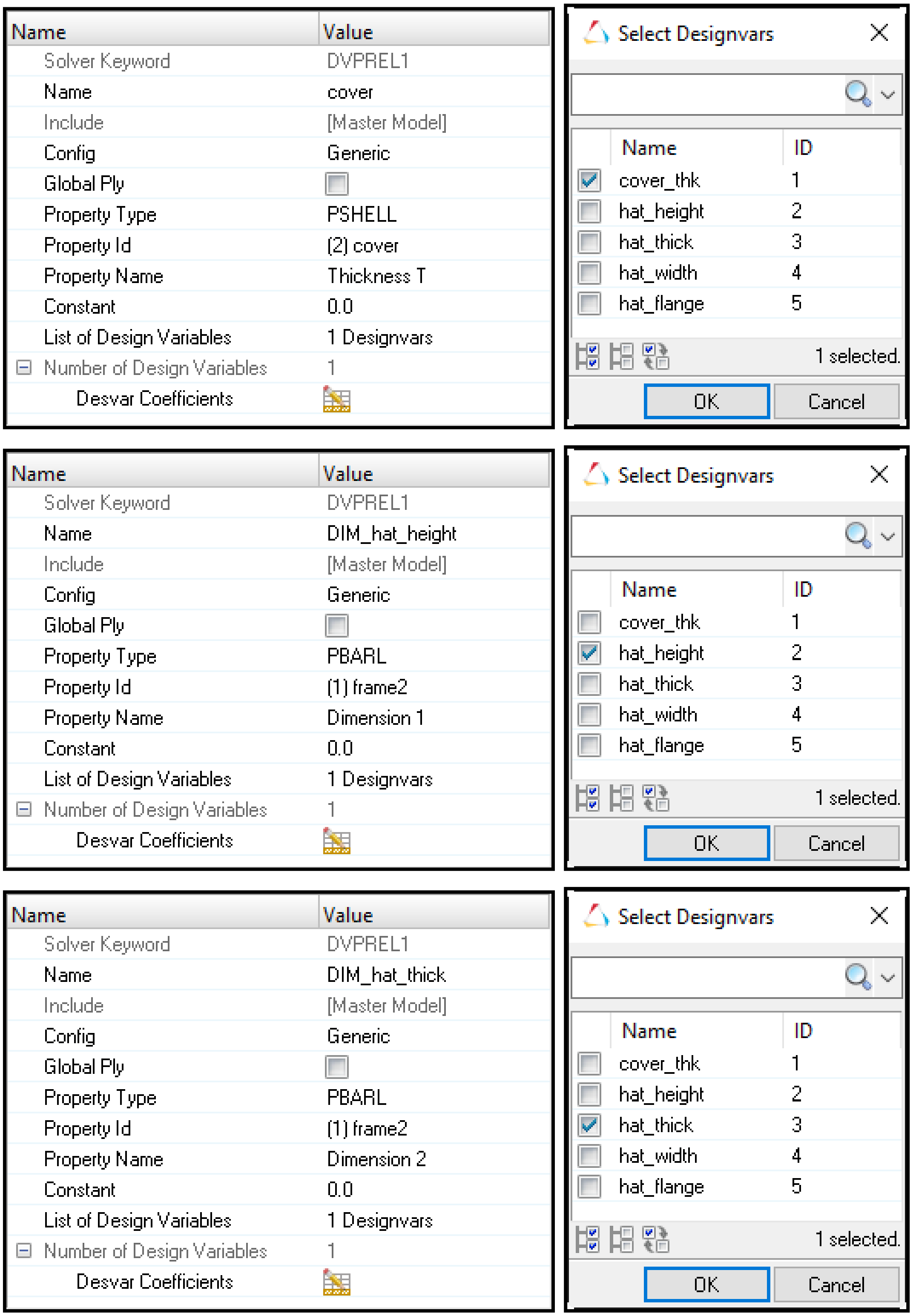
Name	Value
Solver Keyword	DESVAR
Name	hat_thick
ID	3
Include	[Master Model]
Config	size/shape
Move Limit	
Ddval Id	<Unspecified>
Shape Id	<Unspecified>
Initial Value	10.0
Lower Bound	3.0
Upper Bound	12.0
RAND	☐
RANP	☐

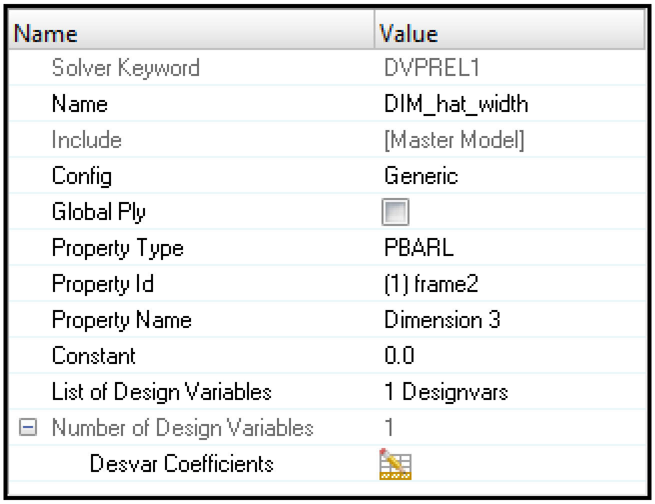
Name	Value
Solver Keyword	DESVAR
Name	hat_width
ID	4
Include	[Master Model]
Config	size/shape
Move Limit	
Ddval Id	<Unspecified>
Shape Id	<Unspecified>
Initial Value	60.0
Lower Bound	40.0
Upper Bound	80.0
RAND	☐
RANP	☐

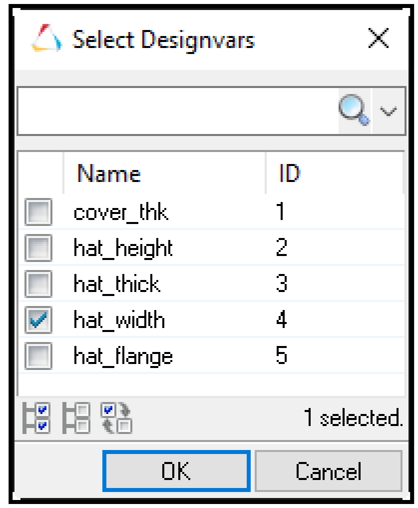
Name	Value
Solver Keyword	DESVAR
Name	hat_flange
ID	5
Include	[Master Model]
Config	size/shape
Move Limit	
Ddval Id	<Unspecified>
Shape Id	<Unspecified>
Initial Value	30.0
Lower Bound	1.0
Upper Bound	40.0
RAND	☐
RANP	☐

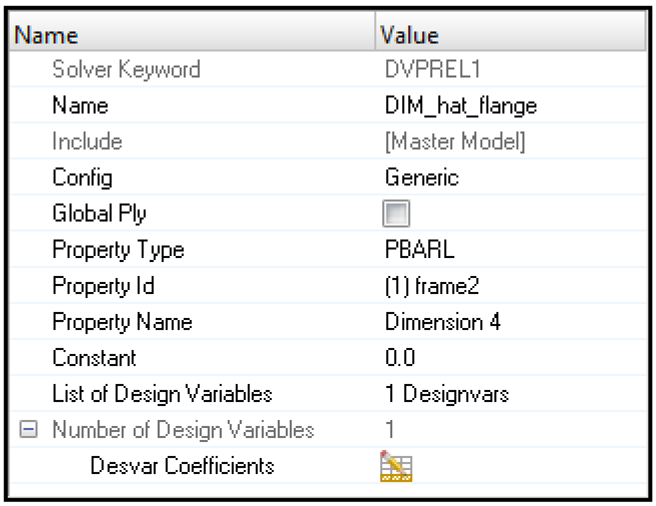
Step 3: Create relationships between the design variables and properties

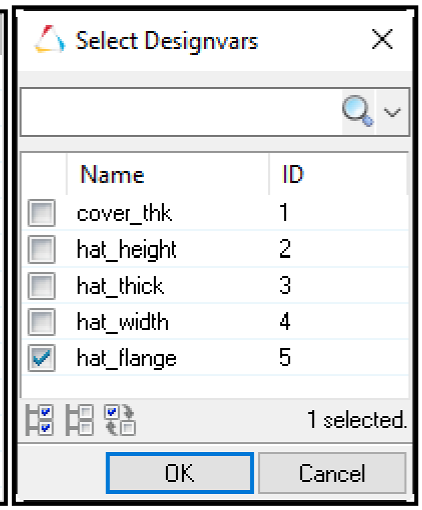
Tip: ***Create > Design Variable Relationships > Generic***



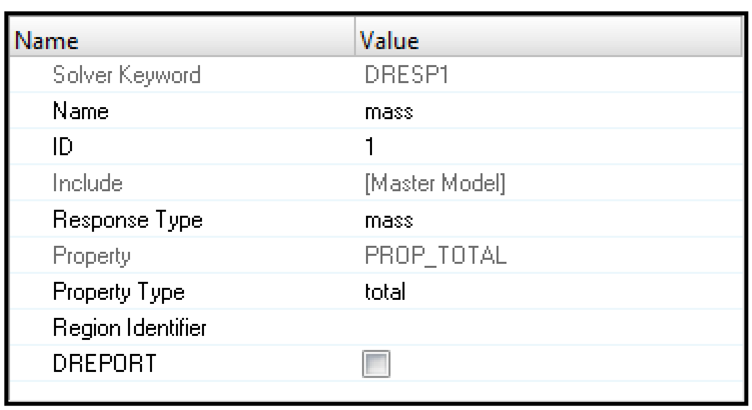








Step 4: Create three responses to track the mass and third and fourth modes



Name	Value
Solver Keyword	DRESP1
Name	freq_mode_3
ID	2
Include	[Master Model]
Response Type	frequency
Property	LOADSTEPS
Region Identifier	
Mode Number:	3
Subcase Mode Tracking	☐
DREPORT	☐

Name	Value
Solver Keyword	DRESP1
Name	freq_mode_4
ID	3
Include	[Master Model]
Response Type	frequency
Property	LOADSTEPS
Region Identifier	
Mode Number:	4
Subcase Mode Tracking	☐
DREPORT	☐

Step 5: Create two constraints: a lower bound for the 4th mode of 6 Hz and an upper bound for the mass of 1 . 8

Name	Value
Solver Keyword	DCONSTR
Name	C_freq_mode_4
ID	1
Include	[Master Model]
Response	(3) freq_mode_4
List of Loadsteps	1 Loadsteps
Lower Options	
Lower Options	Lower bound
Lower Bound	6.0
Upper Options	<OFF>
PROB	

Name	Value
Solver Keyword	DCONSTR
Name	C_mass
ID	2
Include	[Master Model]
Response	(1) mass
Lower Options	<OFF>
Upper Options	
Upper Options	Upper bound
Upper Bound	1.8
PROB	

Step 6: Create the objective to `max` the `f3` response for the `ld1` loadstep

Name	Value
Solver Keyword	DESOBJ(MAX)
Name	Objective
ID	1
Include	[Master Model]
Objective Type	Maximize
Response Id	(2) freq_mode_3
Loadstep Id	(1) ld1

Step 7: Run the optimization in OptiStruct

Step 8: Post-process the results by viewing the `shredder.out` and `shredder.prop` files

```
 DVPREL1/2  USER-ID  PROP-TYPE  PROP-ID  ITEM-CODE  PROP-VALUE
-------------------------------------------------------------
 DVPREL1          1   PSHELL          2   T          6.312E-01
 DVPREL1          2   PBAR            1   DIM1       1.250E+02
 DVPREL1          3   PBAR            1   DIM2       1.153E+01
 DVPREL1          4   PBAR            1   DIM3       8.000E+01
 DVPREL1          5   PBAR            1   DIM4       4.000E+01
-------------------------------------------------------------


ITERATION    7
  TRACKED OBJECTIVE FUNCTION/CONSTRAINT MODE #  4 GOES OUT OF BOUND,
  RETURNED TO THE PREVIOUS GOOD DESIGN.

Objective Function (Maximize FREQ ) =  6.34153E+00   % change =        0.00
Maximum Constraint Violation %      =  0.00000E+00
Volume                              = 2.33075E+008   Mass     = 1.79468E+000

 Subcase Mode   Order     Weight      Frequency     Eigenvalue    Weight/Eigen
       1     1      1   1.000E+00   1.031023E+00   4.196590E+01   2.382887E-02
       1     2      2   0.000E+00   1.534846E+00   9.300133E+01   0.000000E+00
       1     3      3   0.000E+00   6.341527E+00   1.587623E+03   0.000000E+00
       1     4     10   0.000E+00   8.145025E+00   2.619055E+03   0.000000E+00
       1     5      4   0.000E+00   7.358200E+00   2.137484E+03   0.000000E+00
       1     6      5   0.000E+00   7.358540E+00   2.137682E+03   0.000000E+00
       1     7      6   0.000E+00   7.359955E+00   2.138504E+03   0.000000E+00
       1     8      7   0.000E+00   7.361666E+00   2.139498E+03   0.000000E+00
       1     9      8   0.000E+00   7.363115E+00   2.140341E+03   0.000000E+00
       1    10      9   0.000E+00   7.423498E+00   2.175590E+03   0.000000E+00
-------------------------------------------------------------------------------
~ implies mode is ambiguously tracked    * implies mode cannot be tracked.
```

Tip: Some information about the design history can be reviewed visually by animating a contour of the thickness results in HyperView. Since most of the optimization in the model was performed on 1D element properties, the `*.prop` file is the best way to examine the final results.

```
$ PROPERTIES AND MATERIALS AT ITERATION    7
$ ----------------------------------------
$
$HMNAME PROP                  1"frame2" 3
$HWCOLOR PROP                 1      38
PBARL    1              1         HAT
+, 125.0, 11.525224906, 80.0, 40.0, 0.0
$ A=4459.681 I1=8799222. I2=7028952. I12=     0.0 J=197461.0 K1=.5269398 K2=.3539209
$ C1~F2 = +       65.87983    40.0-59.1202    80.0-59.1202   -80.065.87983   -40.0
$M1A~N2B= +            0.0      0.0      0.0      0.0      0.0      0.0      0.0      0.0
$HMNAME PROP                  2"cover" 4
$HWCOLOR PROP                 2       3
PSHELL, 2, 1, .63122871619, 1, 1.0, 1, .833333, 0.0
```

Since the `*.prop` file contains the optimization results in OptiStruct FEM format, it can be imported over an existing model to replace the property information in that file, allowing easy update of models to the latest optimization results.