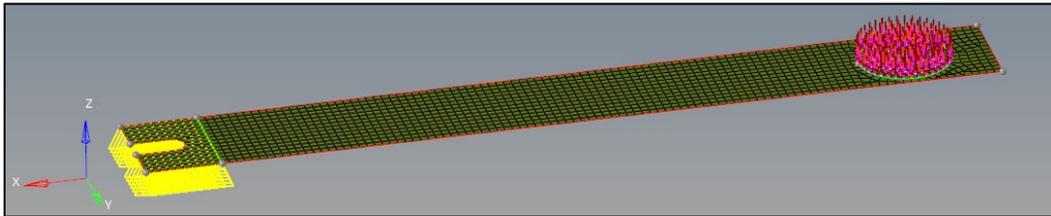


Static Analysis of the Cantilever

In this tutorial, we show you how to set-up and run the finite element analysis of a cantilever beam/structure. Post-processing tool used is HyperView, to determine the natural frequency of the cantilever beam.



1. Launching HyperMesh and Setting the OptiStruct

User Profile

1. Launch HyperMesh

The User Profile dialog appears.

2. Select OptiStruct and click OK

This loads the user profile. It includes the appropriate template, macro menu, and import reader, paring down the functionality of HyperMesh to what is relevant for generating models for OptiStruct.

2. Opening the Model

1. Click File > Open > Model

2. Select the cantilever.hm file you have as the input
3. Click Open

The cantilever.hm database is loaded into the current HyperMesh session, replacing any existing data. The database only contains geometric data. The geometry has four components namely the cantilever, constrain, load area and strain gauge.

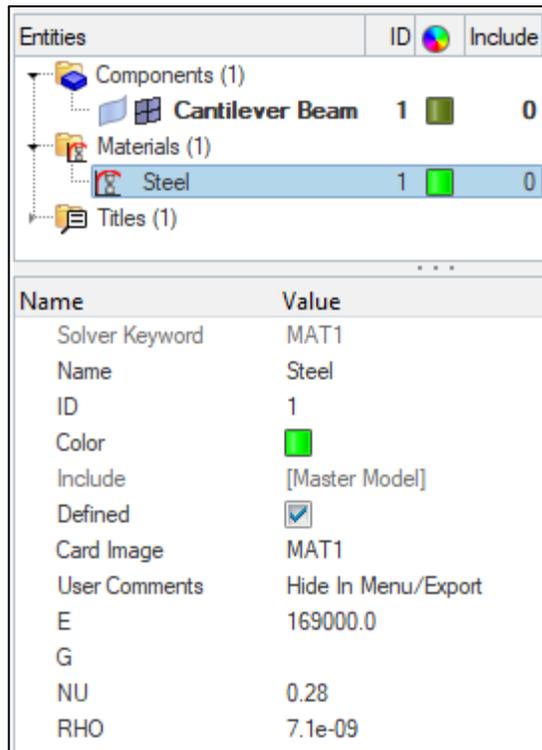
3. Setting Up the Model

3.1 Creating the Mesh

1. Go to 2D > automesh and select the cantilever all component/surface
2. Set the mesh size to 2.5 mm and set the toggle element to surface component
3. Click on mesh
4. A mesh is created on the surface with an element size of 2.5 mm

3.2 Creating the Material

1. In the Model Browser, right-click and select Create > Material from the context menu. A default material displays in the Entity Editor
2. For Name, enter Steel
3. Set Card Image to MAT1
4. Enter the material values next to the corresponding fields
 - a. For E (Young's Modulus), enter 169000 N/mm²
 - b. For NU, (Poisson's Ratio), enter 0.28
 - c. For RHO (Mass Density), enter 7.1e-9 T/mm³



Material Values for steel

New material steel has been created. The material uses OptiStruct's linear isotropic material model, MAT1.

3.3 Creating the Property

1. In the Model Browser, right-click and select Create > Property from the context menu.

A default property displays in the Entity Editor.

2. For Name, enter 1mm Thickness.

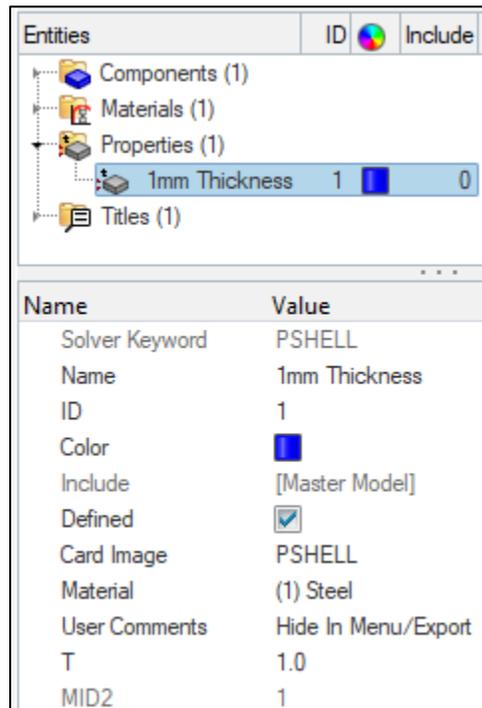
3. Set Card Image to PSHELL.

Enter the property values next to the corresponding fields.

An empty Value field indicates that it is turned off. To edit these properties, click on the blank Value fields next to them and enter the required values.

- a. For Material, click Unspecified. In the Select Material dialog, select steel and click OK.
- b. For T (thickness of the plate), enter 1

Property Values



Name	Value
Solver Keyword	PSHELL
Name	1mm Thickness
ID	1
Color	
Include	[Master Model]
Defined	<input checked="" type="checkbox"/>
Card Image	PSHELL
Material	(1) Steel
User Comments	Hide In Menu/Export
T	1.0
MID2	1

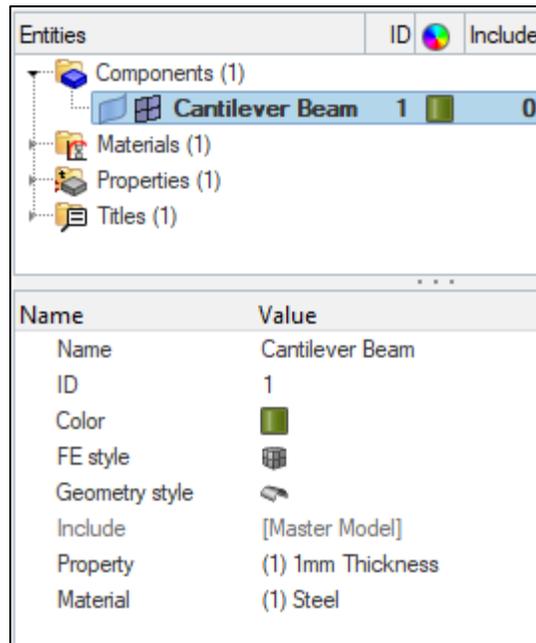
A new property shell has been created as a 2D PSHELL. Material information is also linked to this property.

3.4 Updating the Components

1. In the Model Browser, select the component of the model.

The component fields are displayed in the Entity Editor below.

Assign the material (steel) and property (Pshell) in the required fields.



The components have been updated now. The components use the shell property definition with a thickness value of 1mm ,the material steel is referenced by the components.

4. Applying Loads and Boundary Conditions

For static analysis loading, we have 4 weights from the experimental setup which we are applying simultaneously. To replicate same in finite element analysis we will create 4 load collector for forces 0.28 N, 0.54 N ,0.82N & 1.1 N

4.1 Creating Load Collectors

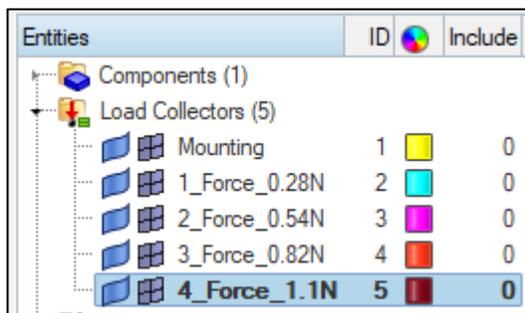
1. In the Model Browser, right-click and select Create > Load Collector from the context menu.

A default load collector displays in the Entity Editor.

- For Name, enter Mounting
- Click Colour and select a colour from the colour palette
- Set Card Image to None. A new load collector, Mounting is created.



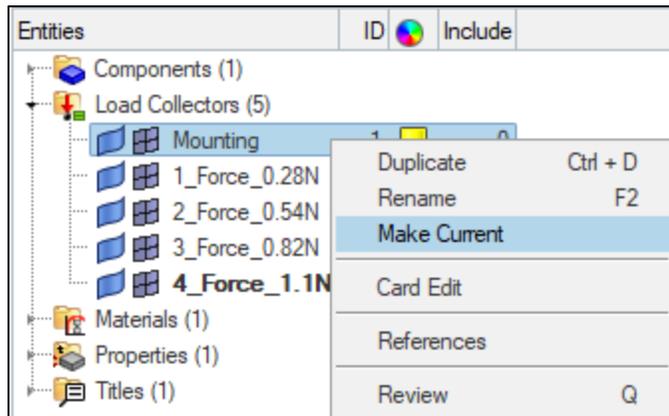
- Creating load collector for forces. Repeat step 1-step4 and create another load collector 0.28 N, 0.54 N ,0.82N & 1.1 N



4.2 Creating Constraints

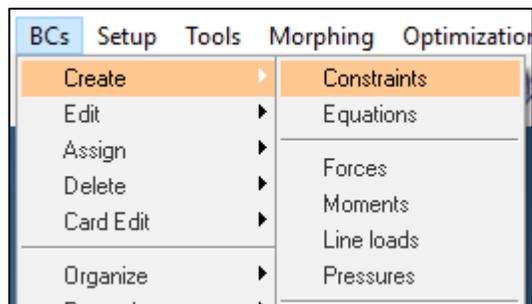
- In the Model Browser, Load Collectors folder, right-click on Mounting and select Make Current to set Mounting as the current load collector.

Setting Mounting as the Current Load Collector



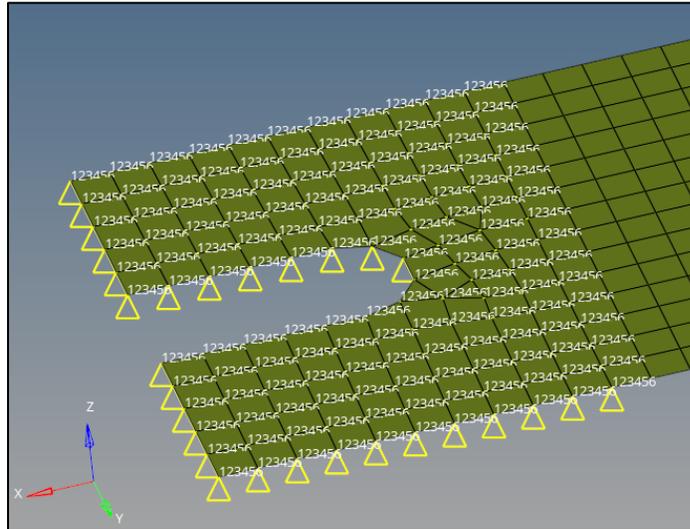
- From the menu bar, click BC's > Create > Constraints to open the Constraints panel.

Accessing the Constraints Panel



- Make sure nodes is selected from the entity selection switch
Geometry is trimmed in such a way that one could easily pick the nodes of mounting location (nodes>by geometry>surface selection)
- Constrain dof1, dof2, dof3, dof4, dof5, and dof6 and set all of them to a value of 0.0.
 - DOFs with a check will be constrained while DOFs without a check will be free.
 - DOF's 1, 2, and 3 are x, y, and z translation degrees of freedom
 - DOF's 4, 5, and 6 are x, y, and z rotational degrees of freedom

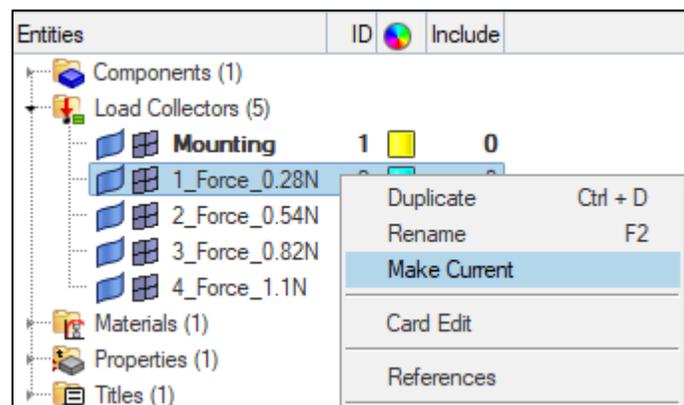
5. Click create. Constraints are applied to the selected nodes
6. Click return to go back to the main menu



4.3 Creating Forces

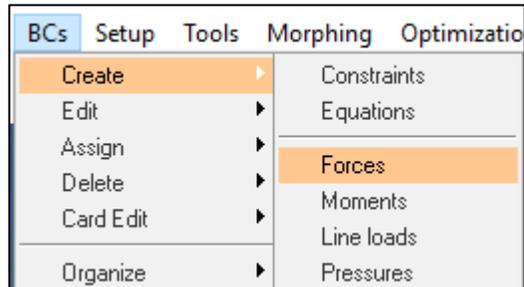
1. In the Model Browser, Load Collectors folder, right-click on Mounting and select Make Current to set **1_Force_0.28N** as the current load collector.

Setting 1_Force_0.28N as the Current Load Collector



- From the menu bar, click BC's > Create > Forces to open the Forces panel.

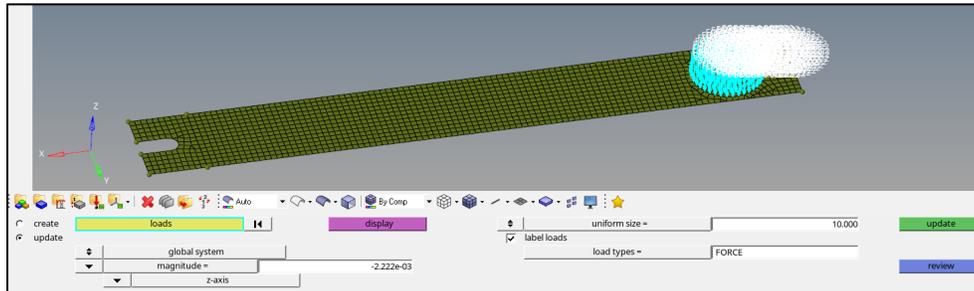
Accessing the Forces Panel



- Make sure nodes is selected from the entity selection switch

Geometry is trimmed in such a way that one could easily pick the nodes of loading location (nodes>by geometry>surface selection)

Magnitude of the force used is calculated by 0.28 (Force in newton) divided by 126 (no of nodes selected in that region) = $-2.222e-03$

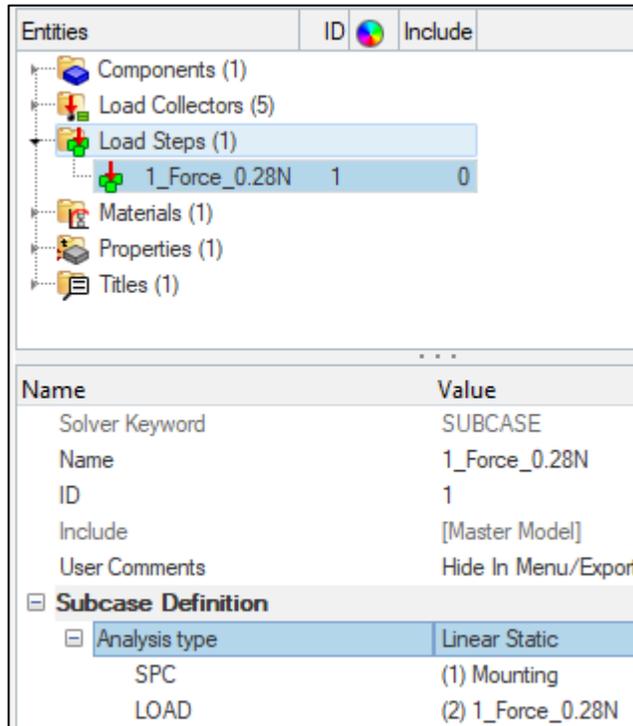


- Similarly create forces by repeating step1-step3 **2_Force_0.54N**, **3_Force_0.82N** & **3_Force_1.1N**

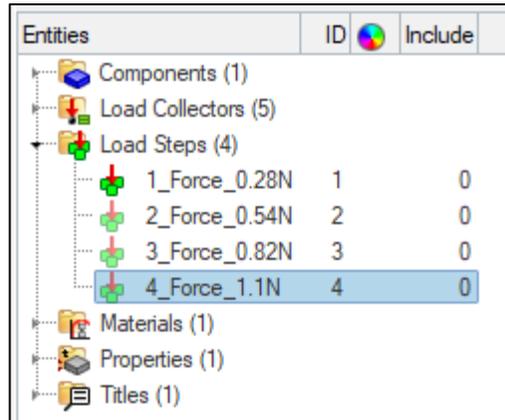
5. Creating Load Steps

- In the Model Browser, right-click and select Create > Load Step from the context menu.

2. Create load step named **1_Force_0.28N**
3. Set Analysis type to Linear Static
4. For SPC, click Unspecified > select the load collector Mounting
5. For LOAD select the **1_Force_0.28N** load collector

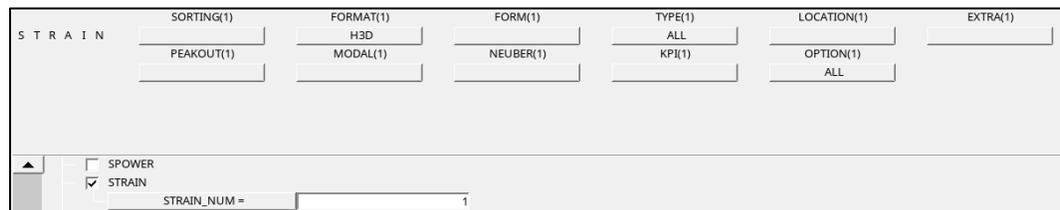


6. Similarly create Load Steps by repeating step1-step5 **2_Force_0.54N**, **3_Force_0.82N** & **3_Force_1.1N**



6. Creating output requests

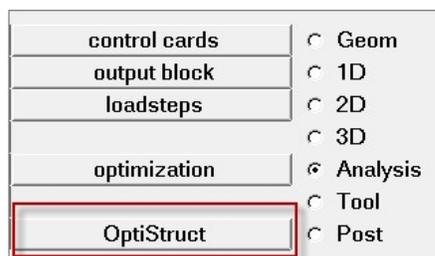
1. Click on Analysis > control cards > Global_Output_Request and select strain
2. Toggle the selection as described in the image



7. Submitting the Job

1. From the Analysis page, click the OptiStruct panel

Accessing the OptiStruct Panel



2. Click save as



3. In the Save As dialog, specify a location to write the OptiStruct model file and enter cantilever for the filename

For OptiStruct input decks, .fem is the recommended extension.

4. Click Save

The input file field displays the filename and location specified in the Save As dialog

Set the export options to toggle to all

5. Set the run options to toggle to analysis

6. Set the memory options to toggle to memory default

7. Click OptiStruct to launch the OptiStruct job

If the Job is successful, new results files should be in the directory where the cantilever.fem was written. The cantilever.out file is a good place to look for error messages that could help debug the input deck if any errors are present.

The default files written to the directory are:

cantilever.html	HTML report of the analysis, providing a summary of the problem formulation and the analysis results.
cantilever.out	An OptiStruct output file containing specific information on the file setup, the setup of your optimization problem, estimates the amount of RAM and disk space required for the run, information for each of the optimization iterations, and compute time information. Review this file for warnings and errors.
cantilever.h3d	HyperView binary results file.
cantilever.res	HyperMesh binary results file.
cantilever.stat	Summary, providing CPU information for each step during the analysis process.

8. Viewing the Result

We are interested in the natural frequency of the modal analysis subcase.

HyperView is a complete post-processing and visualization environment for finite element analysis (FEA), multibody system simulation, video, and engineering data.

8.1 Viewing the Contour Plot of Stresses & Strains

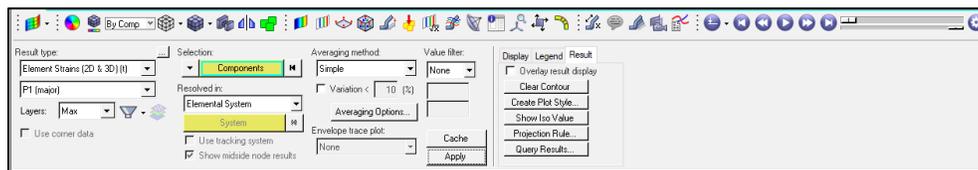
1. From the OptiStruct panel, click HyperView.

HyperView is launched and the results are loaded. A message window appears to inform of the successful model and result files loading into HyperView.

2. On the Results toolbar, click  to open the Contour panel.
3. Define settings in the Contour panel
 - a. Under Result type, set the first pull-down menu to Element Strains (2D & 3D) (t) and set the second pull-down menu to XX

This shows the elemental strains along the strain gauge in the X direction
 - b. Under the component selection option, select the strain gauge component
 - c. Set the Averaging method to simple

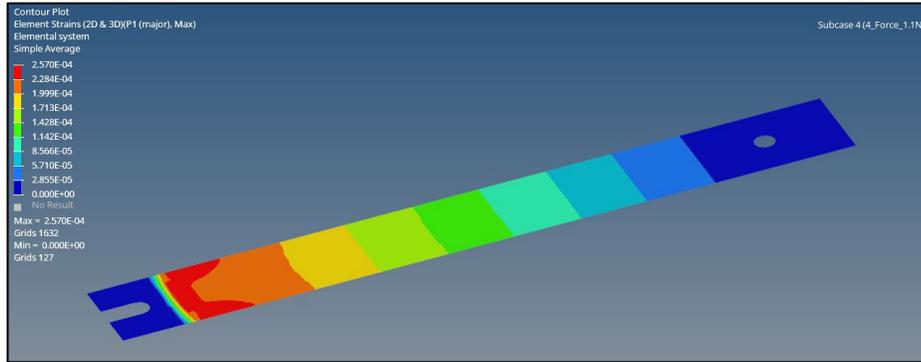
The Contour Panel



4. Click Apply

A contoured image representing direct strains in the X direction along strain gauge should be visible. Each element in the component is assigned a legend colour,

indicating the strain value for that element, resulting from the applied loads and boundary conditions.



Principal Strain for 1.1N (loadcase4): 2.57e-04