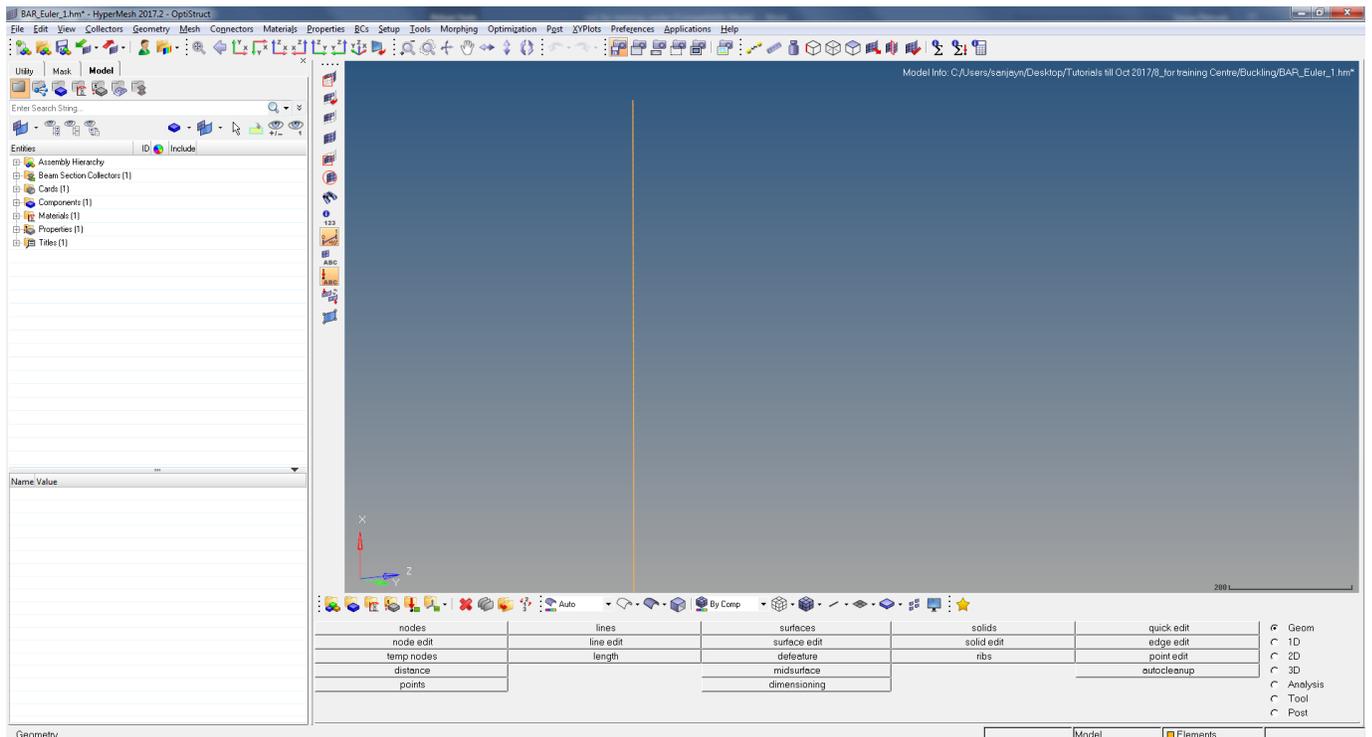


Example: Linear Buckling Analysis of a Beam Structure

Step 1

Launch Hypermesh, select the Optistruct User Profile and retrieve the BAR_EULER_1.hm



Model Information:

$b = 10 \text{ mm}$

$h = 20 \text{ mm}$

$l = 1000 \text{ mm}$

$E = 210,000 \text{ N/mm}^2$, $\nu = 0.3$ (steel)

Area moment of inertia = $1.666,667 \text{ mm}^4$

Analytical Critical Buckling Force $F_{\text{crit}} = 863.59 \text{ N}$ (Euler case 1)

The model is built on 1D elements (of type CBAR) with property card image PBARL (in case you are interested to learn more about 1D elements, please see the corresponding chapter in this book). The material parameters correspond to steel.

In order to better understand the model details, please review the Card images of the property and material collector, respectively.

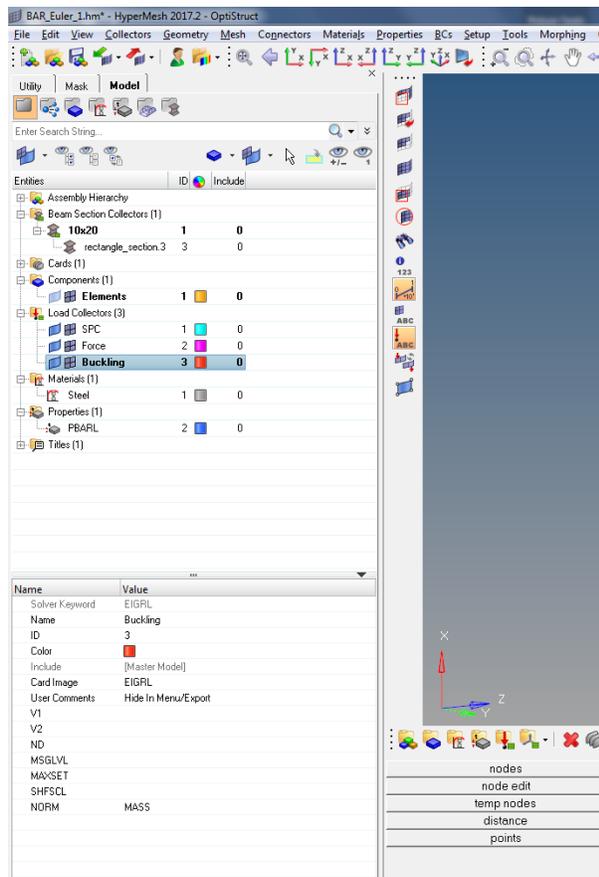
Step 2

Create three load collectors:

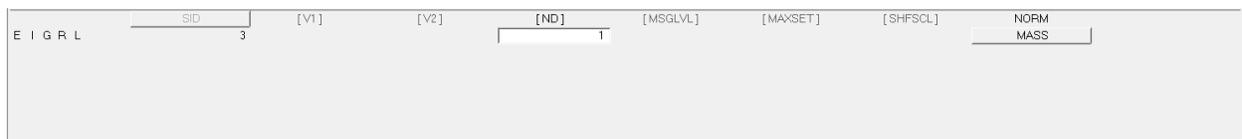
1. SPC (Single Point Constraint, No Card Image)
2. Force (Static Unit Load, No Card Image)
3. Buckling (with Card Image EIGRL)

The load collector can be easily created through right-click in the **Model Browser** window, and activate the option **Create>Load Collector**.

Then simply assign names and colors on your choice, respectively.

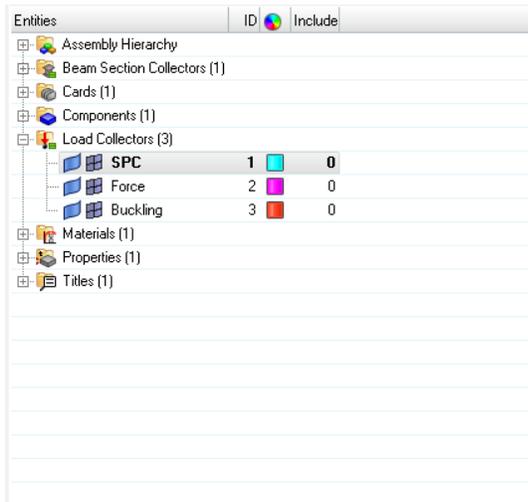


Right Click on the Buckling load collector and select Card Edit. In this enter the value of **ND** as **1**.

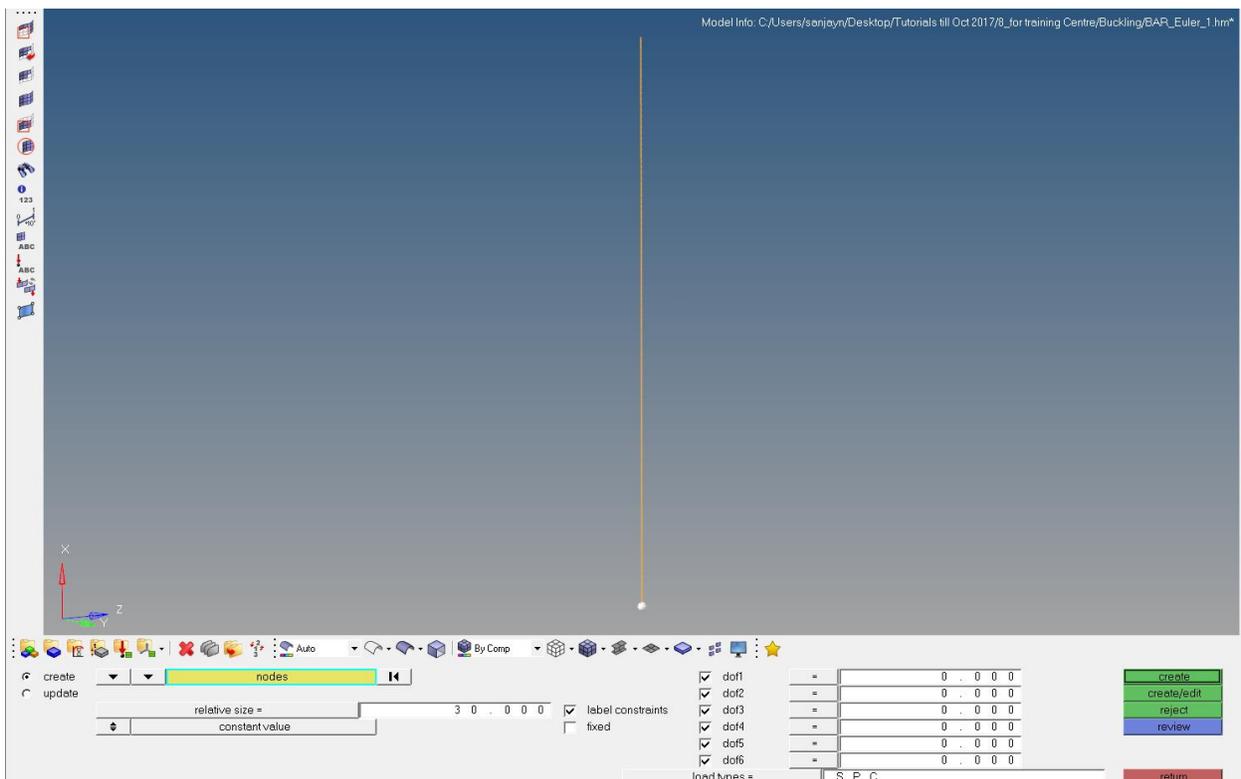


Step 3

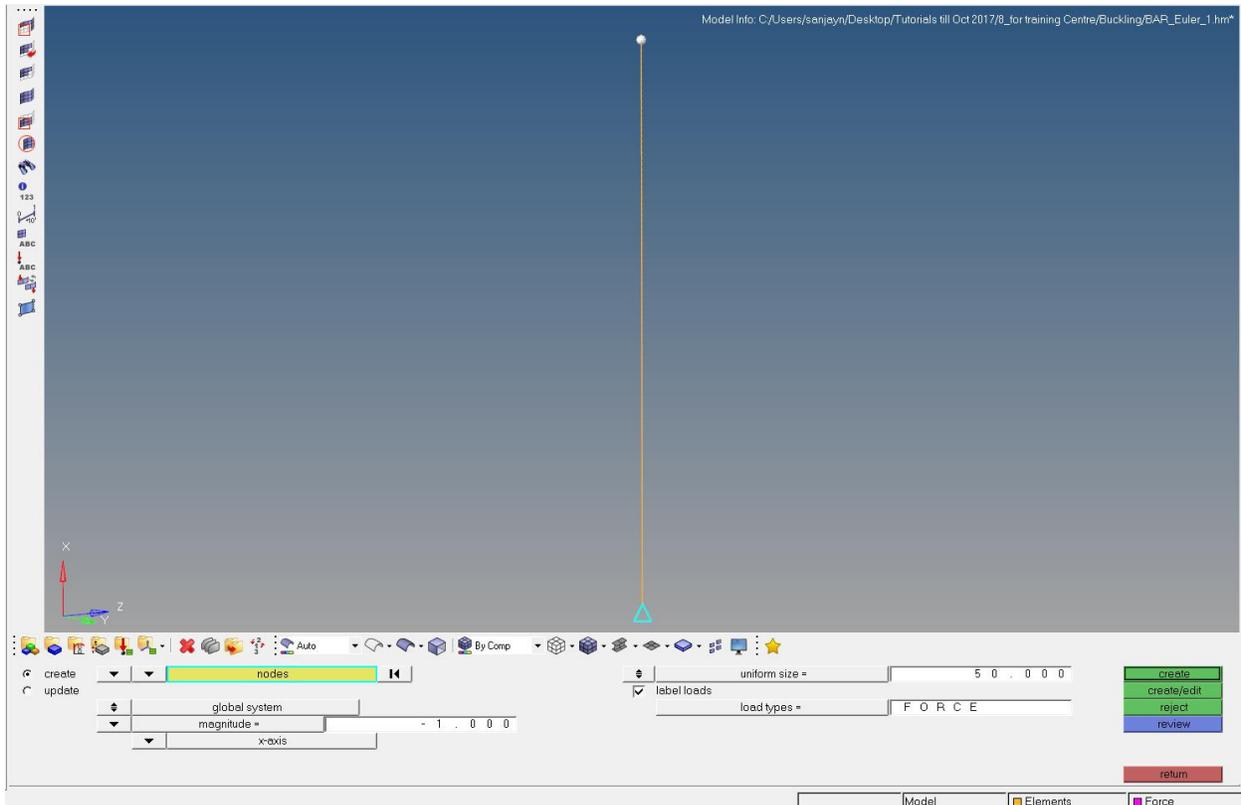
Create the loads and boundary conditions. Make sure that the respective load collector is set to the current load collector (in the **Model Browser** right-click on for instance **SPC** and select **Make Current**).



In the constraints panel (**BCs > Create > Constraints**) select the node on the bottom face of the beam, as shown in the image below. All degrees of freedom are constraint (dof 1-6 =0)



Next set the load collector force as the current load collector and create the reference load (unit load). In the force panel (**BCs > Create > Forces**) the node at the top of the beam is selected, as shown in the figure below:



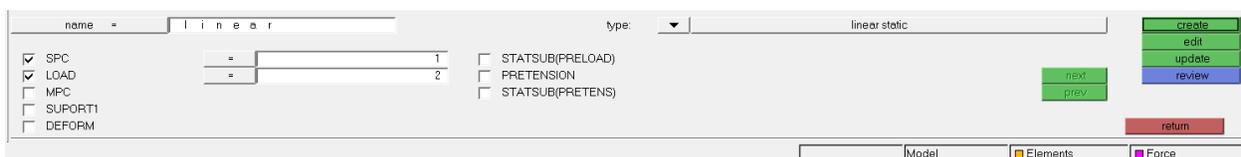
Step 4

Create the loadsteps. For the linear buckling analysis, two loadsteps/subcases are required:

1. Linear Static Loading
2. Linear Buckling Analysis

The subcases are created in the **Loadsteps** panel which can be accessed from the menu bar by selecting **Setup > Create > Loadsteps** which will open a new pop-up menu. This can also be done from **Analysis** page **Loadsteps**.

Make sure the type is set to **linear static**. In here **SPC** references the single point constraint load collector (named **SPC**), and **LOAD** the load collector with the reference load (**load**):



In the second loadstep, the buckling loadstep, make sure the type is set to **Linear Buckling**:



NOTE: STATSUB references the static subcase and **METHOD** the load collector with the information about the buckling modes to be determined.

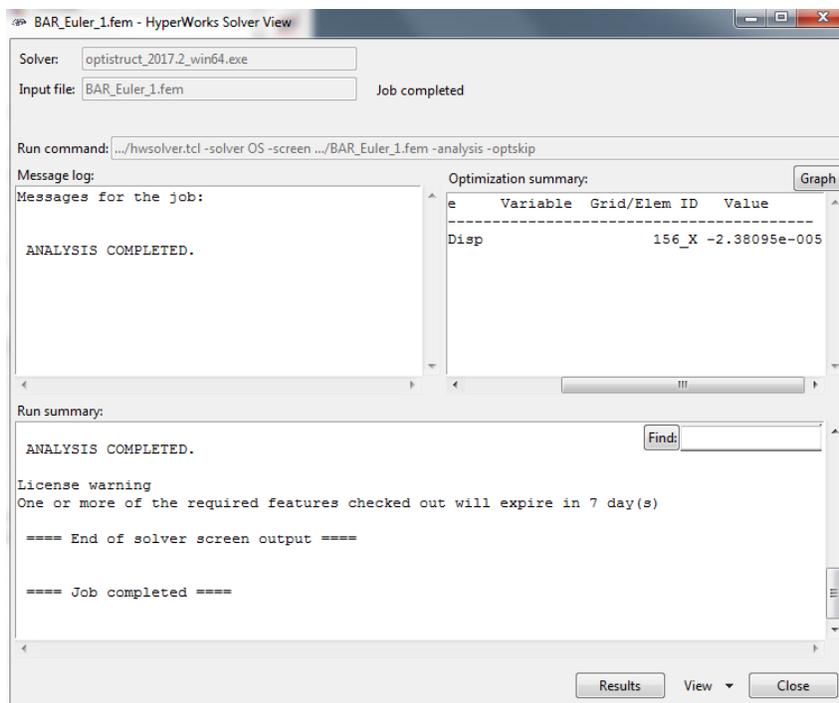
That's all it takes – time to run the analysis.

NOTE: Make sure that the component has been assigned with the property and material, otherwise this will show an error in the run.

Step 5

Start the computation run of OptiStruct. On the **Analysis** page, select the **OptiStruct** panel, choose the working directory and then save the model as .fem. Finally, click **OptiStruct** to execute the calculation.

With the start of the calculation, the Solver View appears. This shows the steps of the calculation and possible errors in the Message log area.



Note: the *.out file includes information from the model check run before the calculation as well as additional base information of the calculated results. Also, check this file error messages and warnings that will help you debug your input deck if any errors are present. In general, always view this file, even if the analysis run was successful.

Step 6

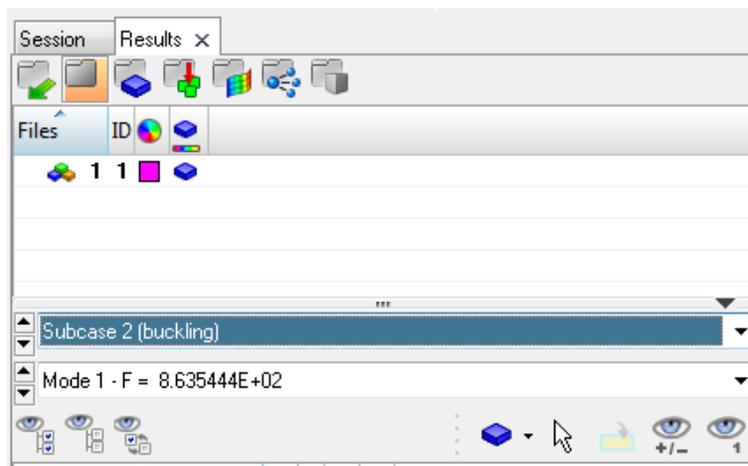
Postprocessing of the analysis. If the job is successful, you should see new result files in the working directory. The working directory is the directory where the model was saved.

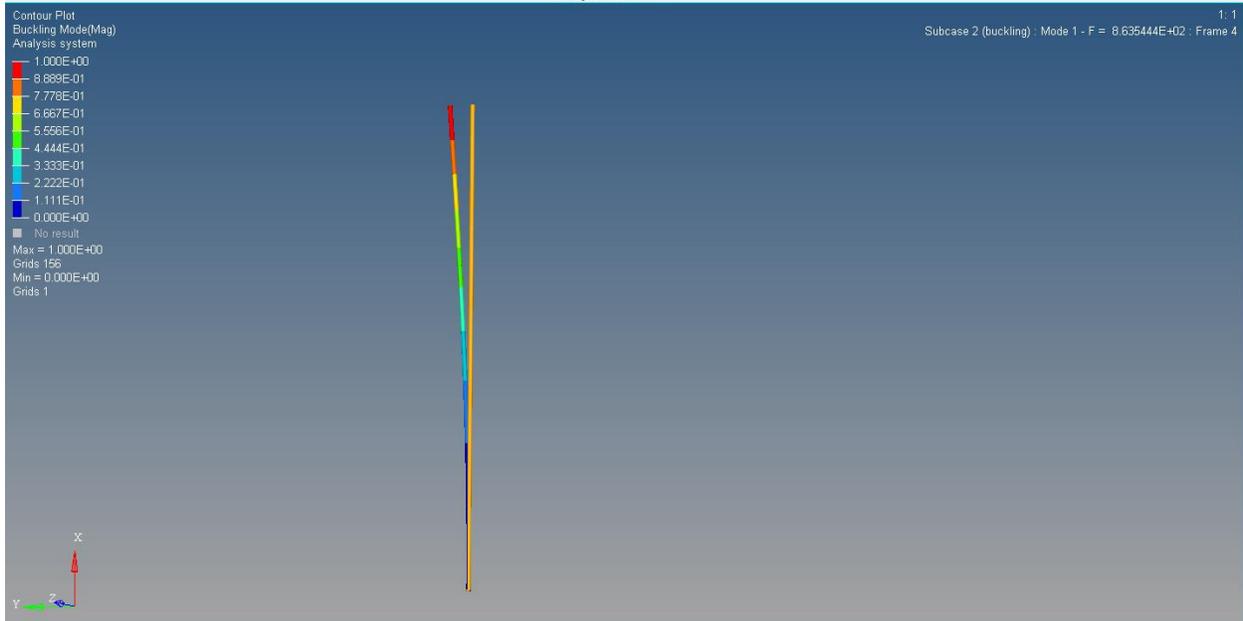
The default files that will be written to your directory are:

| File | Explanation |
|--------|--|
| *.h3d | Binary HyperView result file |
| *.res | Includes all the results of displacement through stress, which can be shown in the Post panel of HyperMesh |
| *.out | ASCII-based output file, which includes information about the model check run before the calculation and additional base information of calculation results. |
| *.stat | Detailed breakdown of the elapsed CPU-time for all important analysis steps |

To view the results, start HyperView

Make sure and pay attention to the loadstep/subcase selected for display. This is controlled in the **Result Browser**.





The critical buckling load for the calculated bar (with unit load 1N) is $F = 863.54$ N (critical buckling factor $\lambda_{cr} = 863.54$). In other words, the model could be loaded 863 times the unit load until buckling would occur. Of course, we need to remember that this is an ideally linear buckling analysis and that real parts tend to buckle at lower loads than the theoretical value due to imperfections.