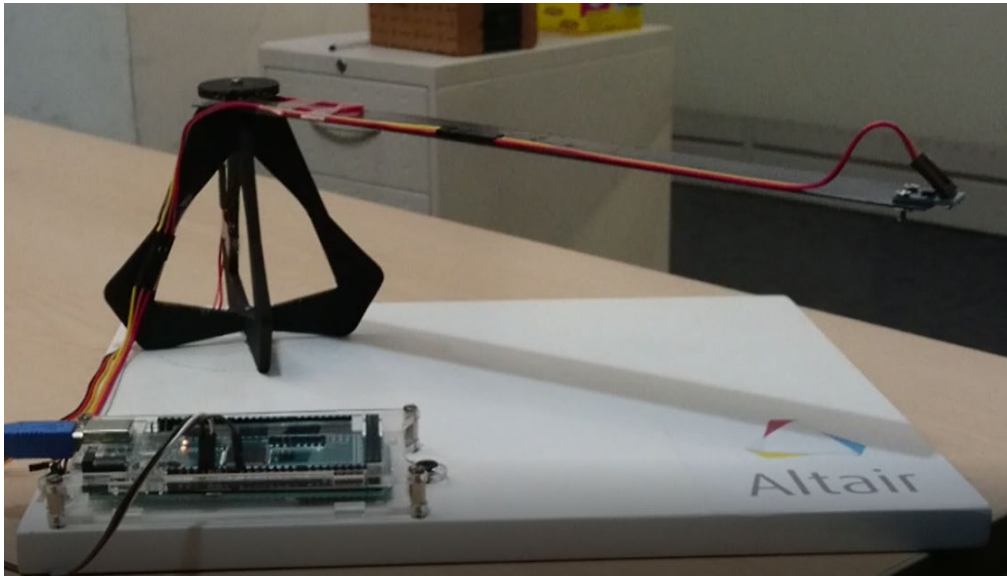


## Linear Static Analysis of the Cantilever beam using Altair Inspire

In this tutorial, we show you how to set-up, run & post-process the finite element analysis of a cantilever beam/structure. CAD design was modelled in such a way that the loads and boundary conditions can be applied smoothly on required region.

### Experimental Setup

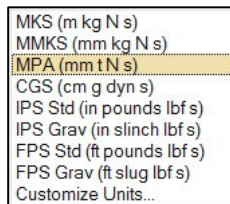


## Step 1: Open the Model

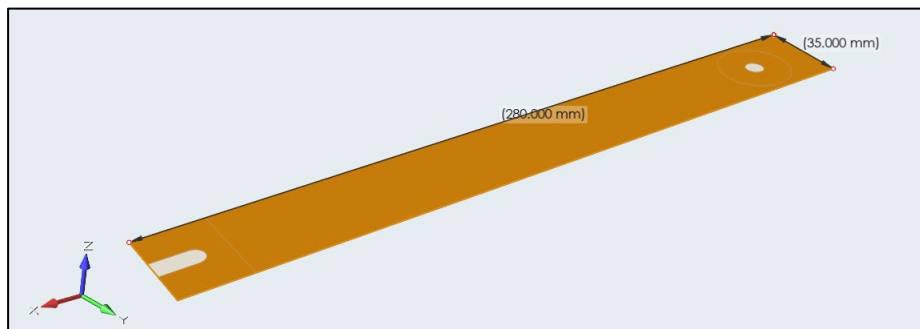
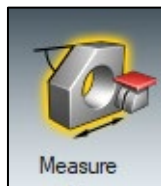
- Start Inspire.
- Click Open Model on the Files icon.



- In the Open File window select the Cantilever Beam.stmod file and click Open.
- Click on the Unit System Selector in the lower right corner of the modeling window, default unit system to MPA (mm t N s).

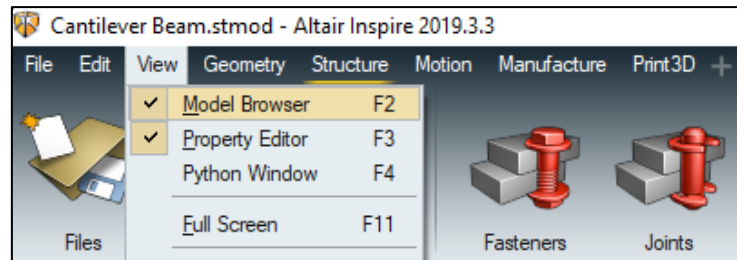


- Use Measure box tool to check the dimension:

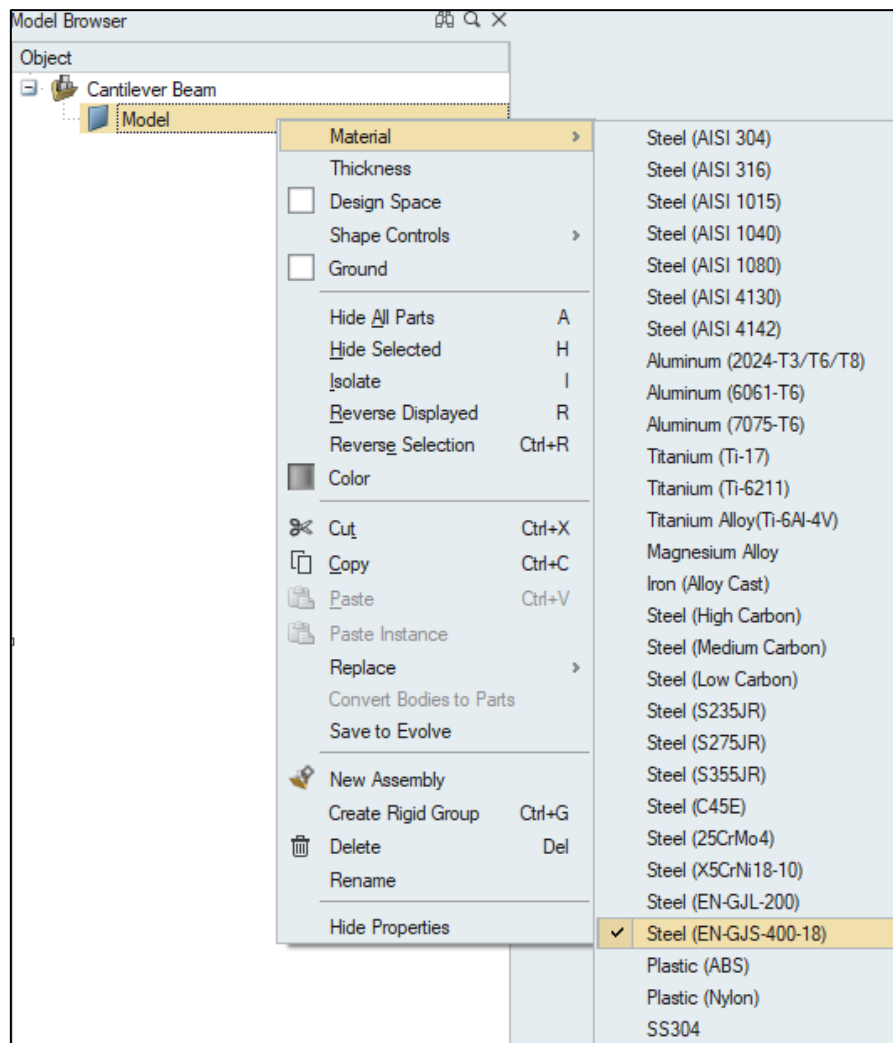


## Step 2: Create Material & Property to the Cantilever beam

- Please make sure to check model browser and property editor from View pull down.

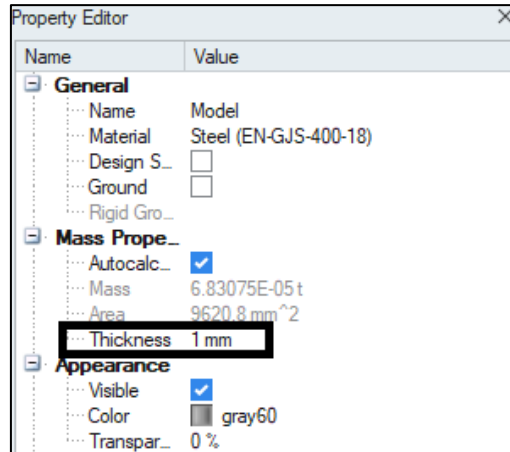


- Right click the model from graphics area or in model browser to assign material from the Inspire Material library.



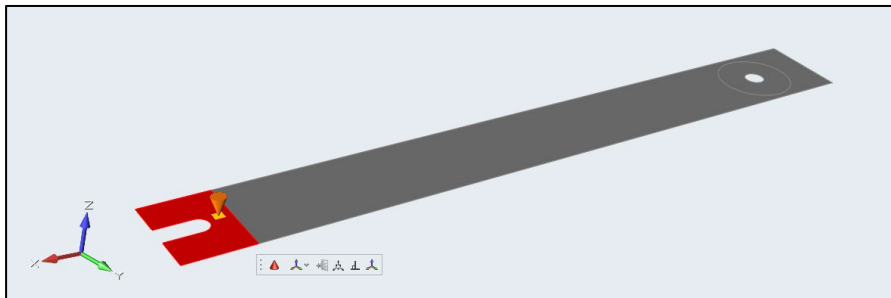
Note: One can also use predefined material or create new material from Inspire material library. To create new material, click Structure>Setup>Materials>My Materials

- To assign thickness right click the model and assign thickness value or one could also use property editor to assign thickness.

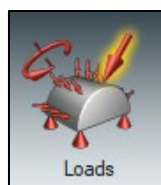


### Step 3: Define loads and boundary condition

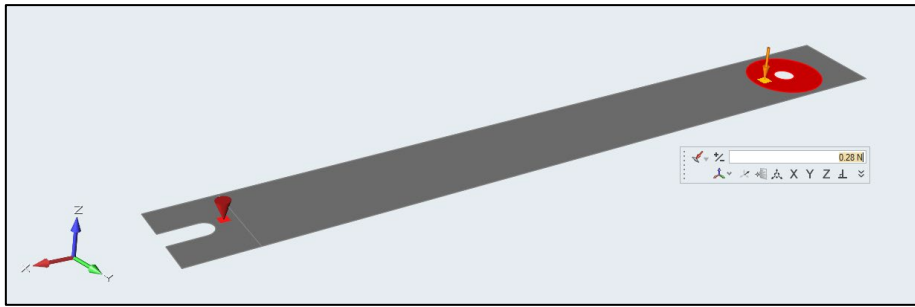
- Click the Structure tab on the ribbon.
- Select the Apply Supports tool on the Loads icon.
- Click on the highlighted part of the cantilever beam to apply the support.



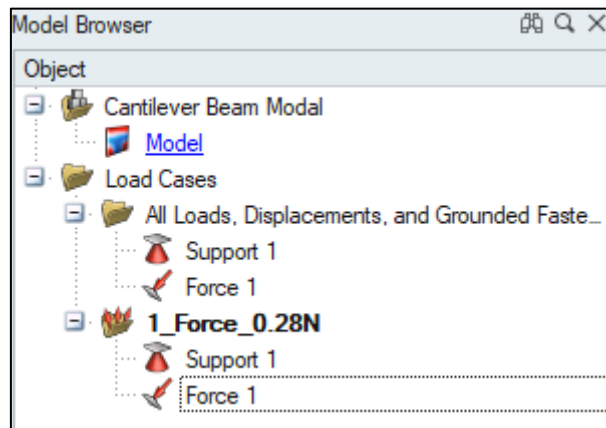
- Now select the Apply forces tools on the load icon



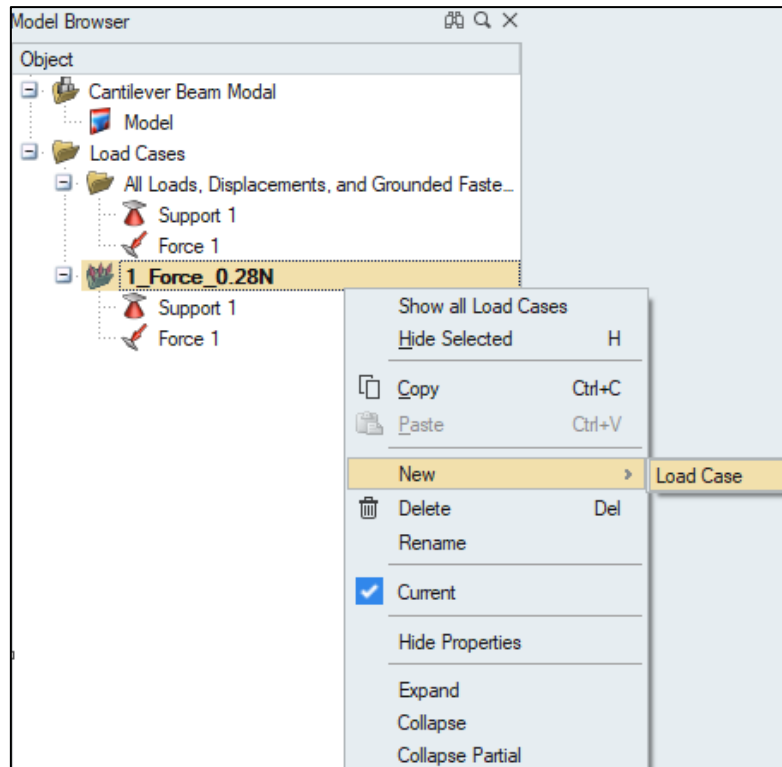
- Click on the highlighted part of the cantilever beam to apply the force of .28 N.



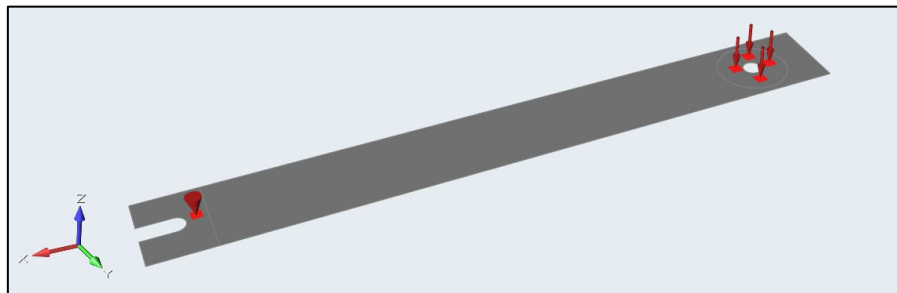
- In the Model Browser, two new folders are created, one called Load Case 1, which we rename as **1\_Force\_0.28N** and the other called All Loads, Displacements & Grounded Fastener. Support 1 is added to both. The name of the load case is shown in bold, indicating that it is the current load case.



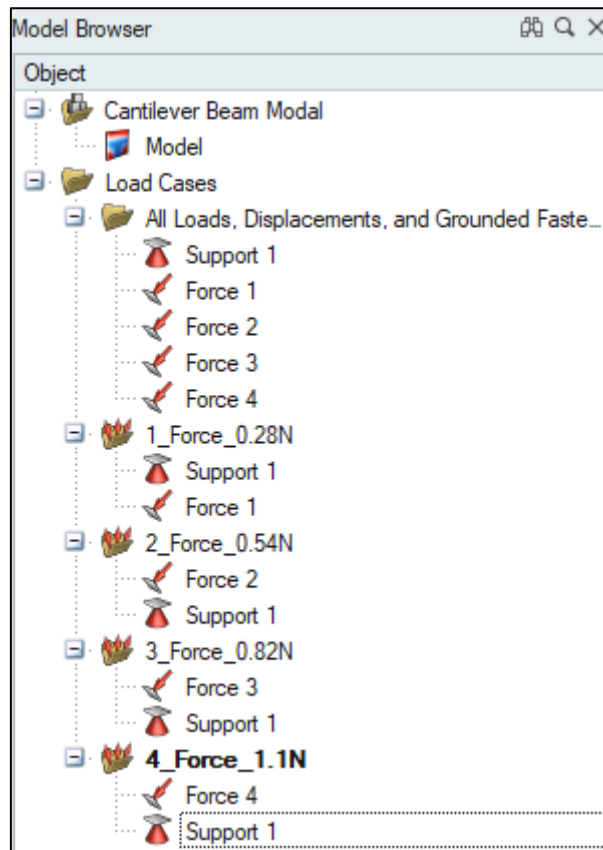
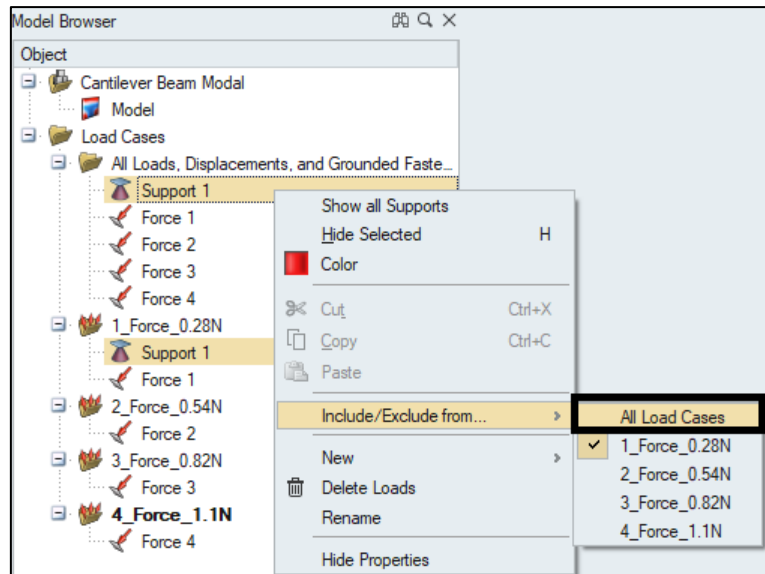
- For loading, we have 4 weights for the experimental setup which we are applying simultaneously. To replicate same in finite element analysis, we will create 4 loadcase.
- First loadcase is already created. To create new loadcase right click **loadcase 1 or 1\_Force\_0.28N** > new > loadcase



- Name the newly created loadcase as **2\_Force\_0.54N** & create force of .54 N
- Repeat same approach to create **3\_Force\_0.82N** & **4\_Force\_1.1N** with force of .82N & 1.1N respectively.



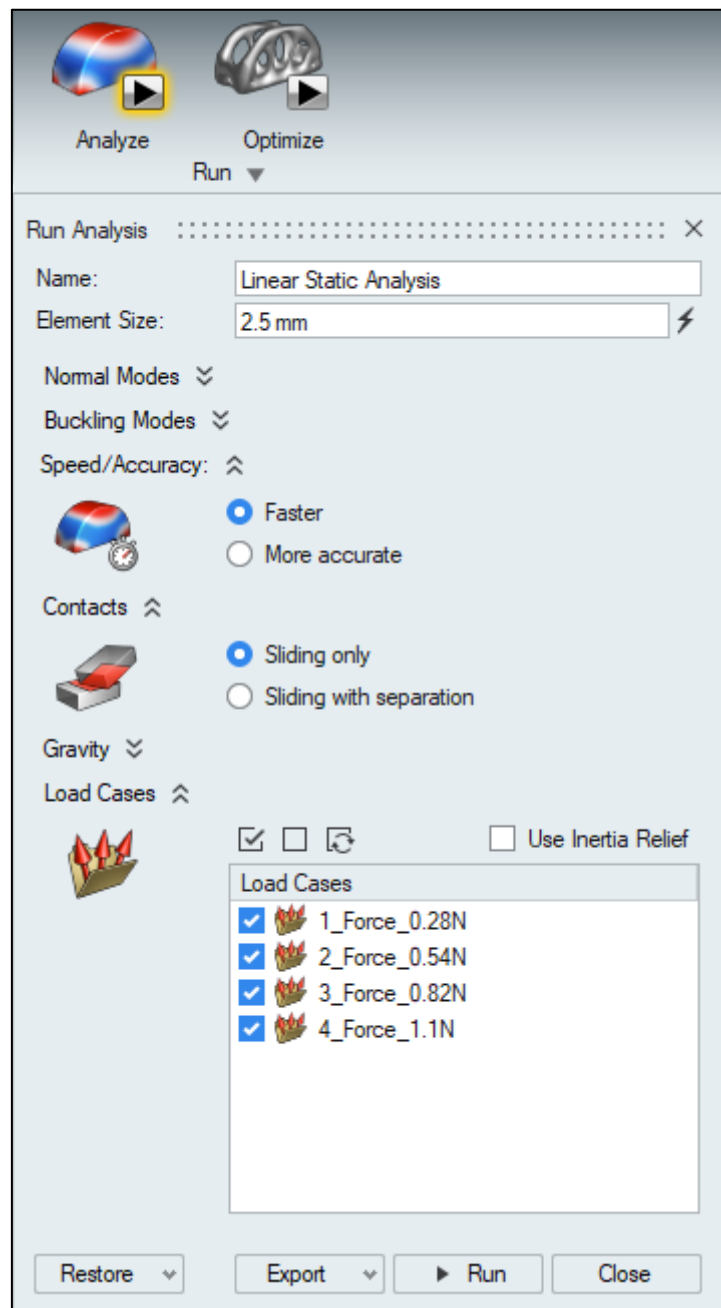
- Since support1 is common for all loadcase. Just right click support1 and include it to all loadcases.



#### Step 4: Run a Linear Static Analysis

- Click on Analyze ribbon
- Name it as **Linear Static Analysis** & set the element size as **2.5mm**

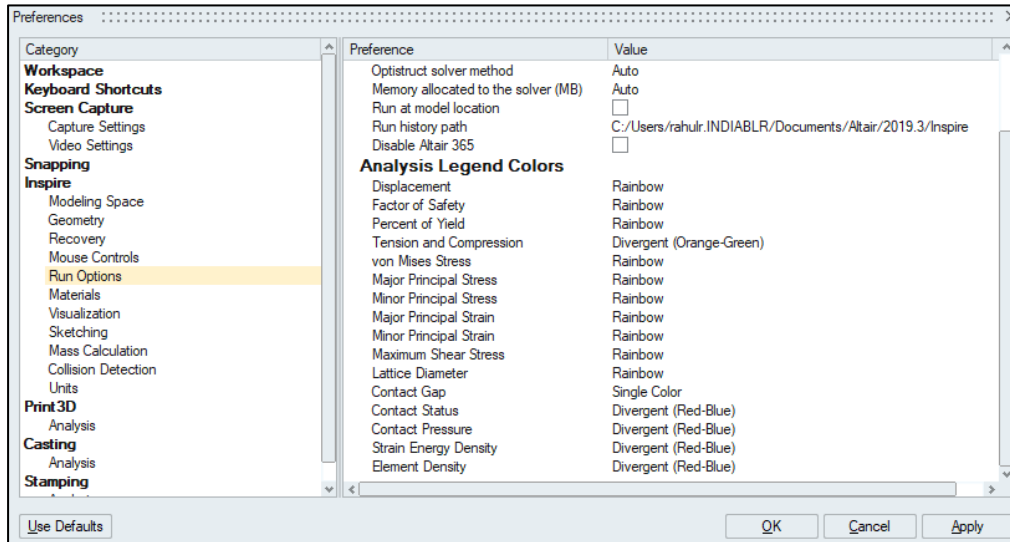
- Make sure that Speed/Accuracy is set to Faster & contacts set to sliding only
- Click Load Cases and verify that all the Load Case is selected.
- Click Run to perform the analysis. The Run Status window appears. A green check mark will appear when the analysis is complete (this may take several minutes).
- Double-click on the name of the run to view the results. The results are displayed in the Analysis Explorer.



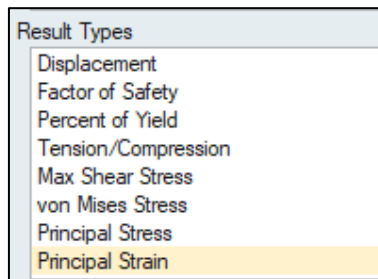


## Step 5: View the Results

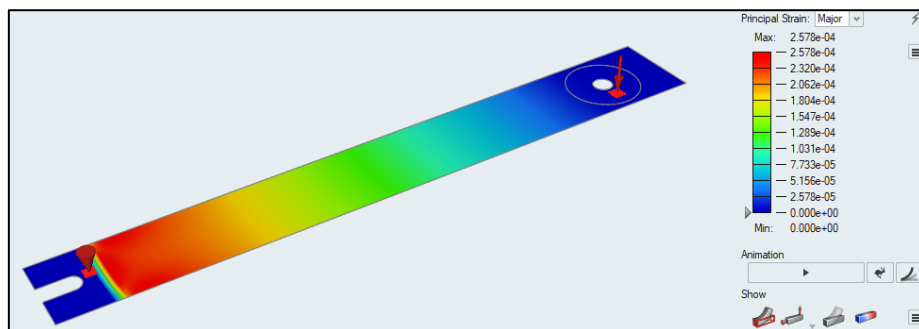
Most of the time you are looking at so called contour plots, i.e. results such as displacements or stresses are displayed using colors (either single color ranging from bright to dark or rainbow colors). To change the Analysis Legend Colors from single or divergent color to Rainbow select **File > Preferences > Run Options** as shown below



- In the Analysis Explorer under Result Type, select Principal Strain.





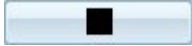


- The maximum principal strain for the model is approximately  $2.57e-4$  mm, as shown in the legend.



Always very helpful is to animate the results as it helps to better understand the overall behaviour of the model. In this way you may see that the model was not properly constraint (fixed).

For your information - the program interpolates the results between the initial (zero displacements) and the final state.

- Click the Show/hide deformed state  icon to show the deformed state of the model. Click the icon again to hide it.
- Click the  button under Animation to visualize the displacement.
- Click the  button and use the slider to change the speed of the animation.
- Inspire automatically scales the displacement animation to make it easier to see.
- Click the  button, deselect Auto, and enter a value to change the scale of the animation.
- Click the  button to stop the animation.