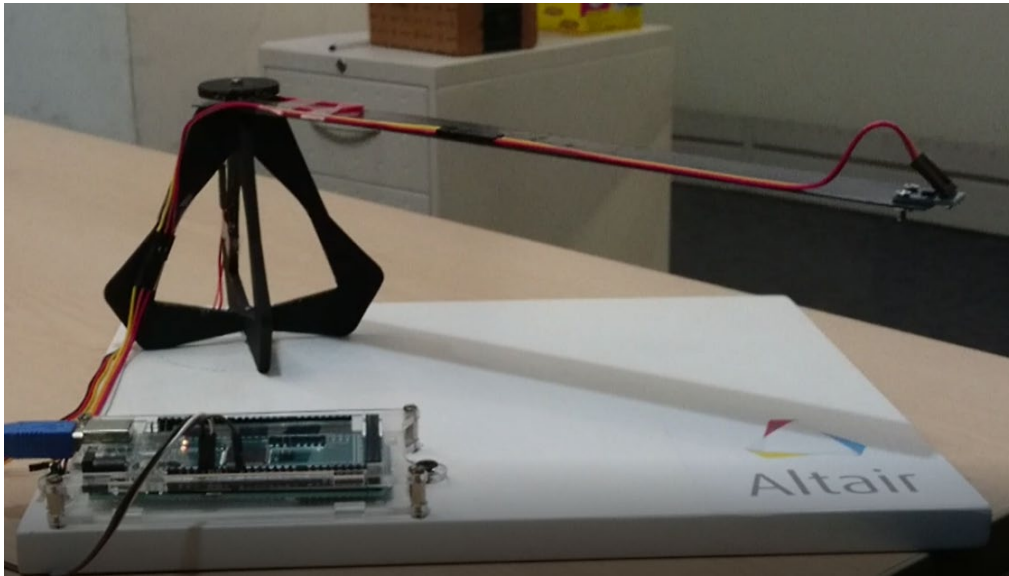




## Linear Static Analysis of the Cantilever beam using Altair SimSolid

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

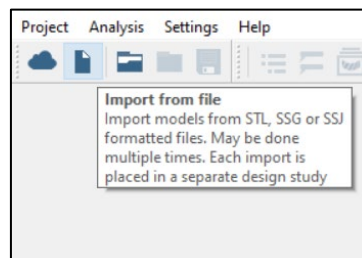
### Approach 1:

If the modal analysis is done prior to this, directly click on structural linear solver to avoid the pre-processing steps and continue from Step 3.

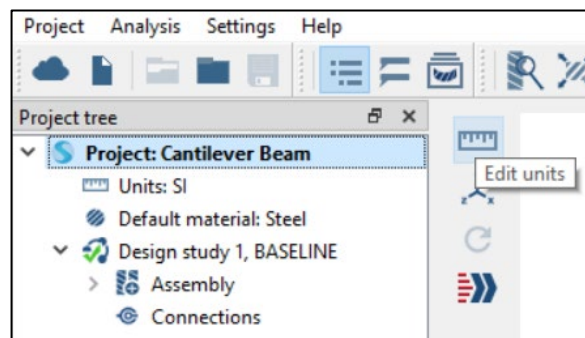
Or

### Approach 2:

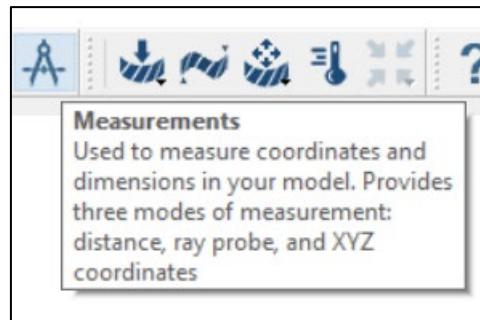
- Start SimSolid.
- Click on “Import from file”



- In the Open File window select the Cantilever Beam.step file and click Open.
- Input and Output units can be set in ‘Edit Units’ icon

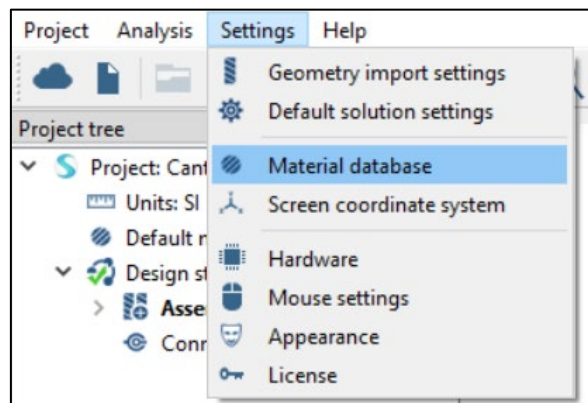


- Use Measure box tool to check the dimension:

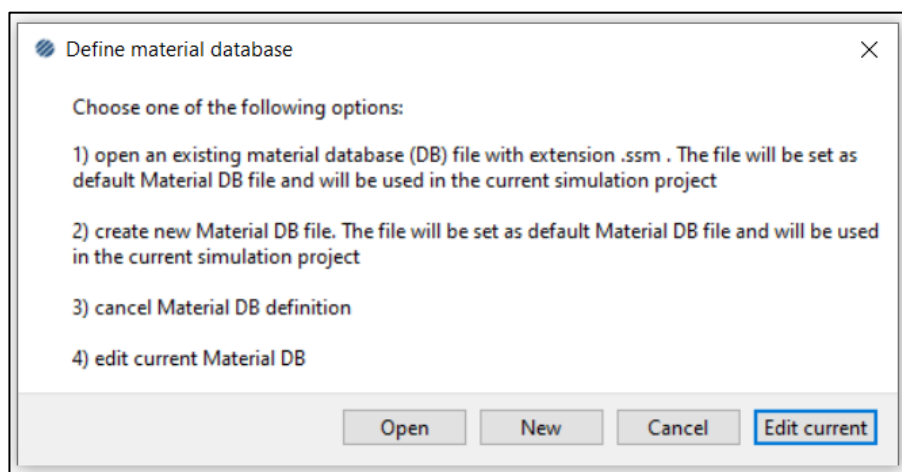


## Step 2: Create Material & Property to the Cantilever beam

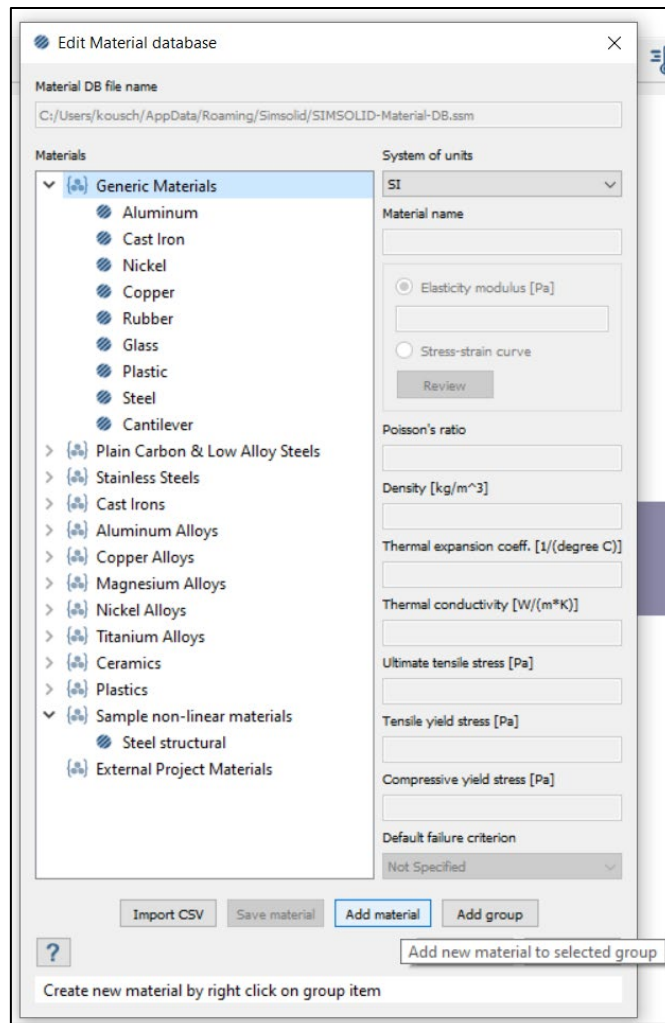
- Add 'Steel (EN-GJS-400-18)' material in the Material database in Settings menu.



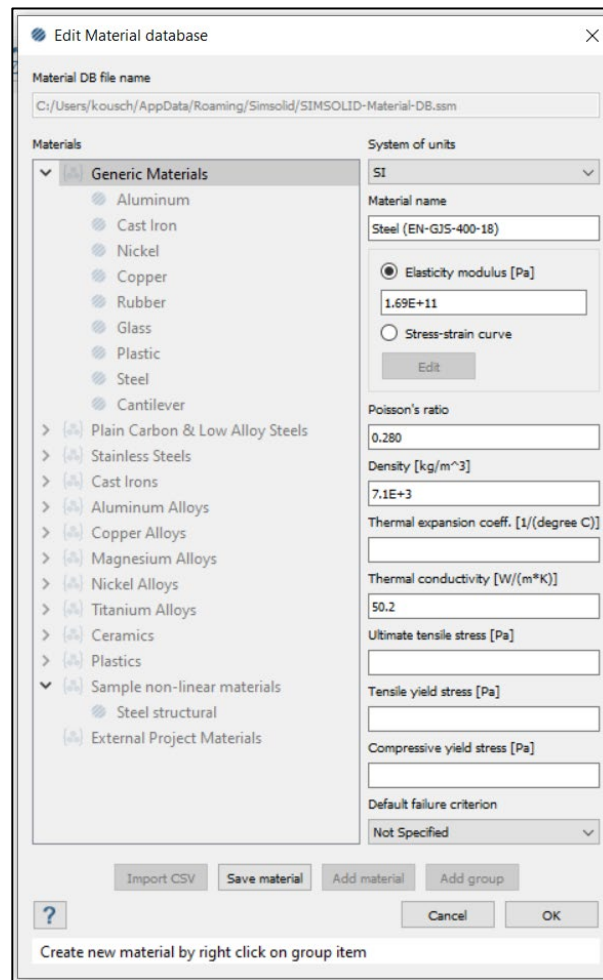
- Click on 'Edit current'



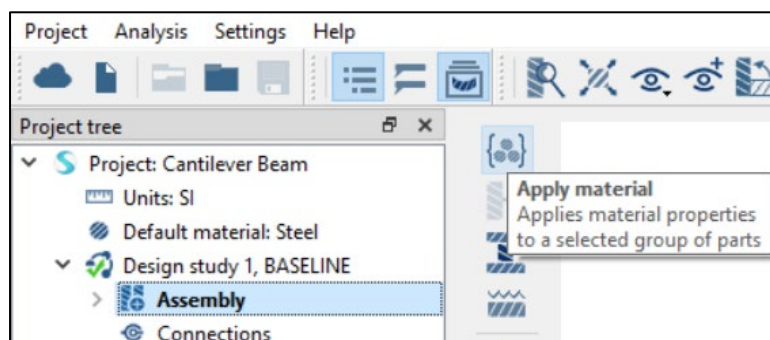
- Click on 'Generic Materials' and then click on 'Add material'



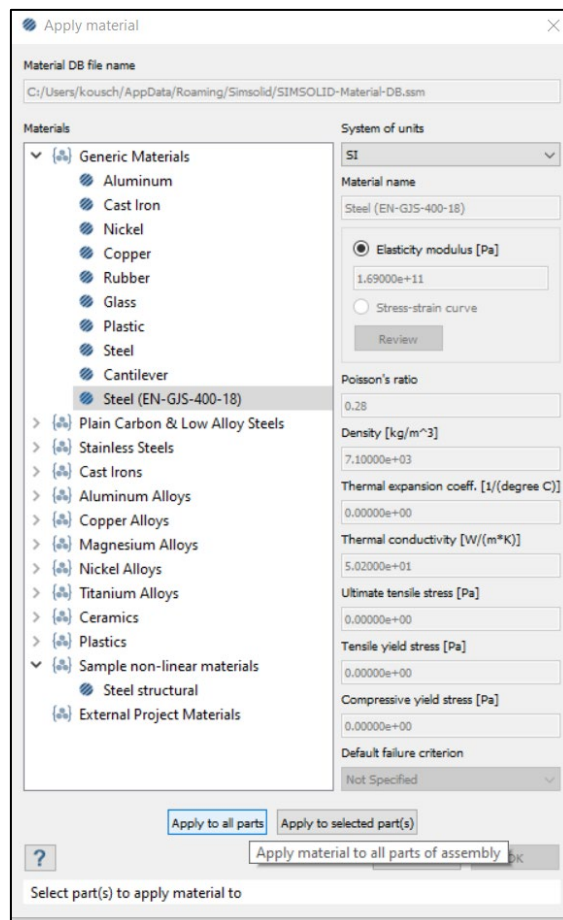
- Enter the material properties as shown below, then ‘Save material’ and click OK



- Click on ‘Assembly’ in the Project tree and click on ‘Apply material’ icon as shown below

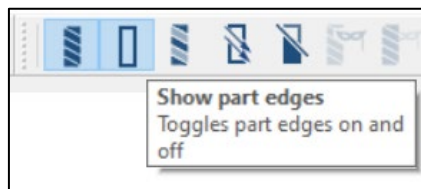


- Select the created material, click on 'Apply to all parts' and click OK

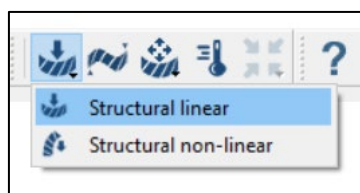


### Step 3: Set up Linear Static Analysis and apply boundary conditions

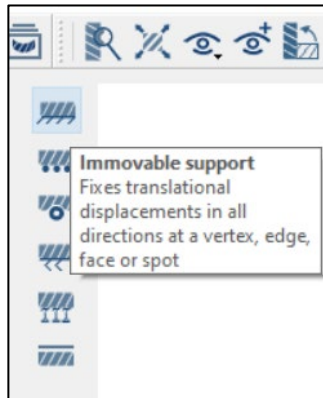
- First, make sure that the partition lines are visible by clicking on 'Show part edges'



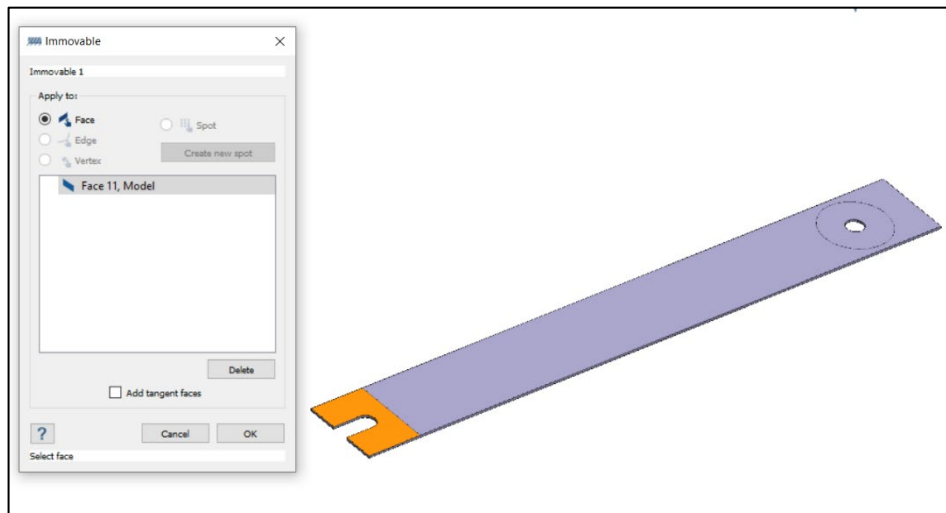
- Click on 'Structural linear' icon in the horizontal toolbar at the top



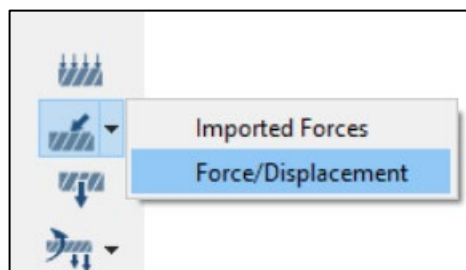
- Click on 'Immovable support' icon as shown below



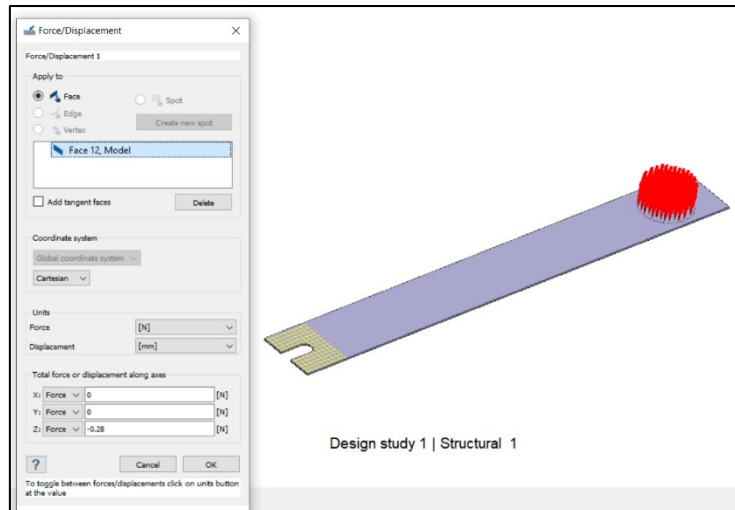
- Click on the highlighted part of the cantilever beam to apply the support.



- Now, click on 'Force/Displacement icon' to apply force

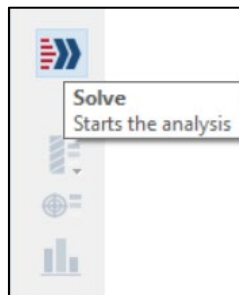


- Click on the highlighted orange part of the cantilever beam to apply the force of .28 N in Z direction (Vertically downwards).

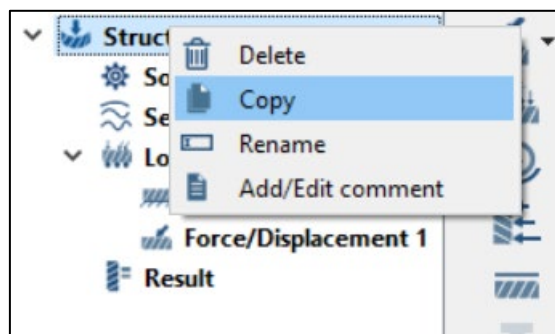


#### Step 4: Run Linear Structural Analysis and repeat the procedure

- Click on 'Solve' icon to run the analysis



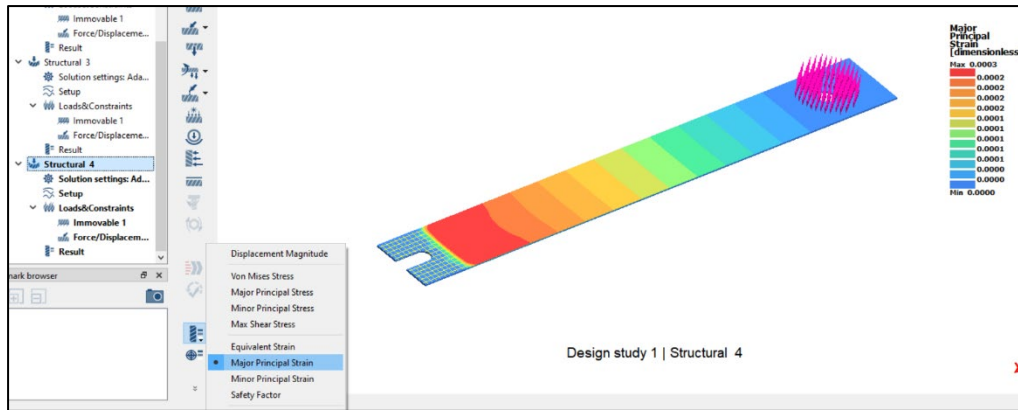
- Copy the structural analysis 3 times and just change the magnitude of the load as: 0.54, 0.82N & 1.1N and run all the analyses.







## Step 5: View the Results

- In the Result plot icon, select Major Principal Strain and view the results as shown below



- The maximum principal strain for the model in 4<sup>th</sup> load case is approximately 0.0003, 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).

- Click the Show deformed shape 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  to stop the animation.