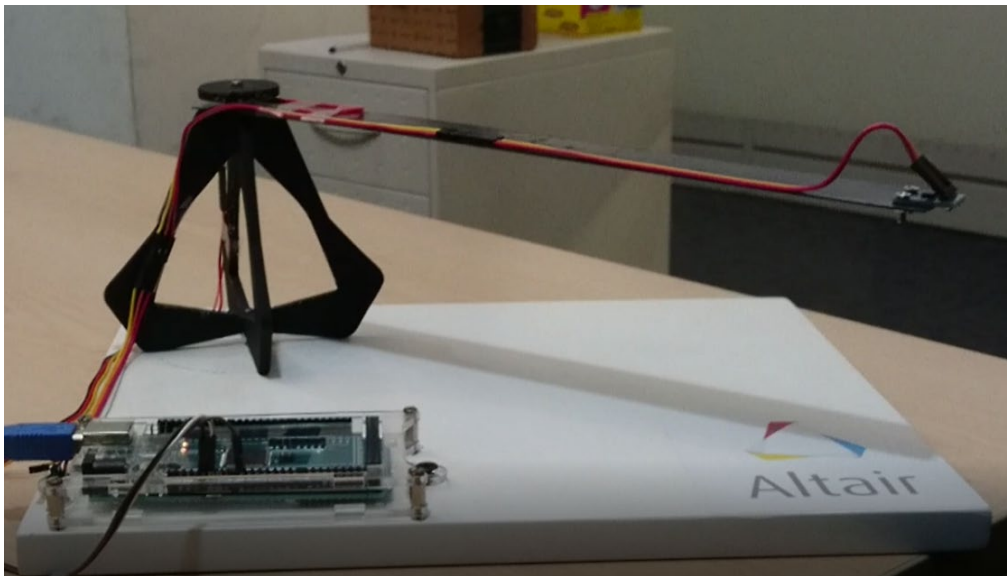


Modal Analysis of the Cantilever beam using Altair

SimSolid

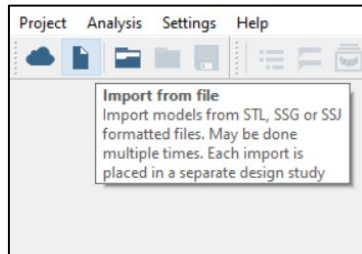
In this tutorial, we show you how to set-up, run & post-process the SimSolid analysis of a cantilever beam/structure. CAD design was modelled in such a way that the loads and boundary conditions can be applied smoothly on required region.

Experimental Setup

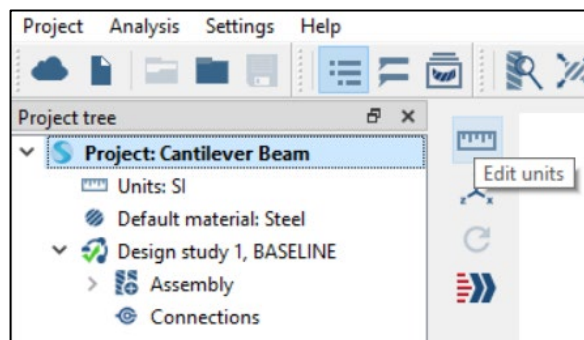


Step 1: Open the Model

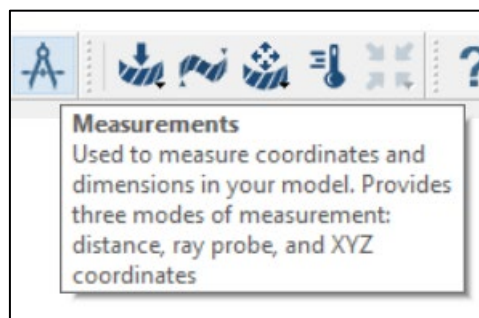
- Start SimSolid.
- Click on “Import from file”



- 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

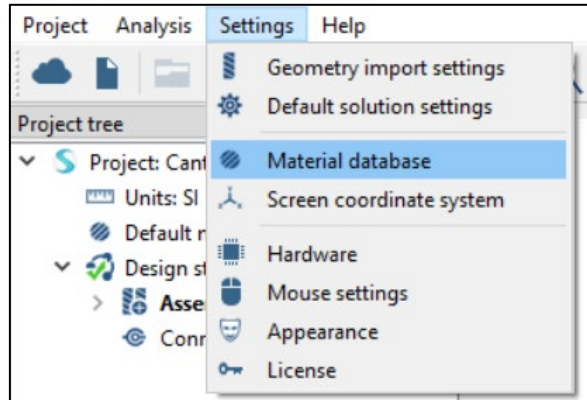


- Use Measure box tool to check the dimension:

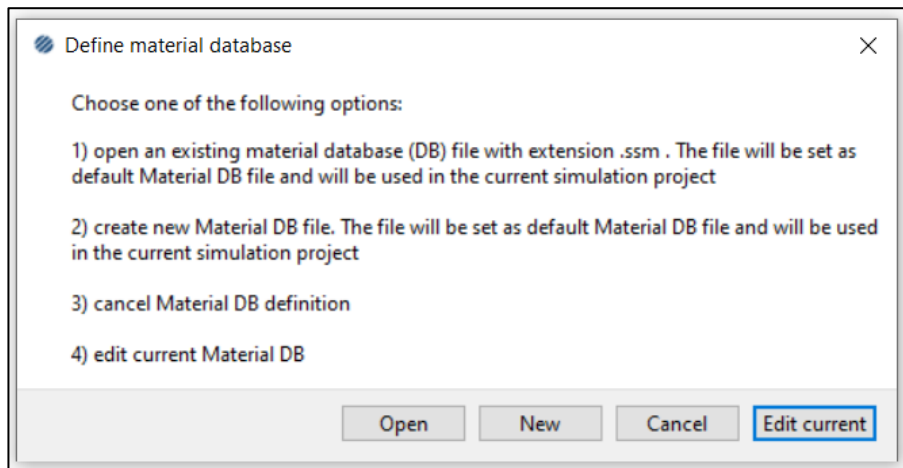


Step 2: Create Material properties for the Cantilever beam

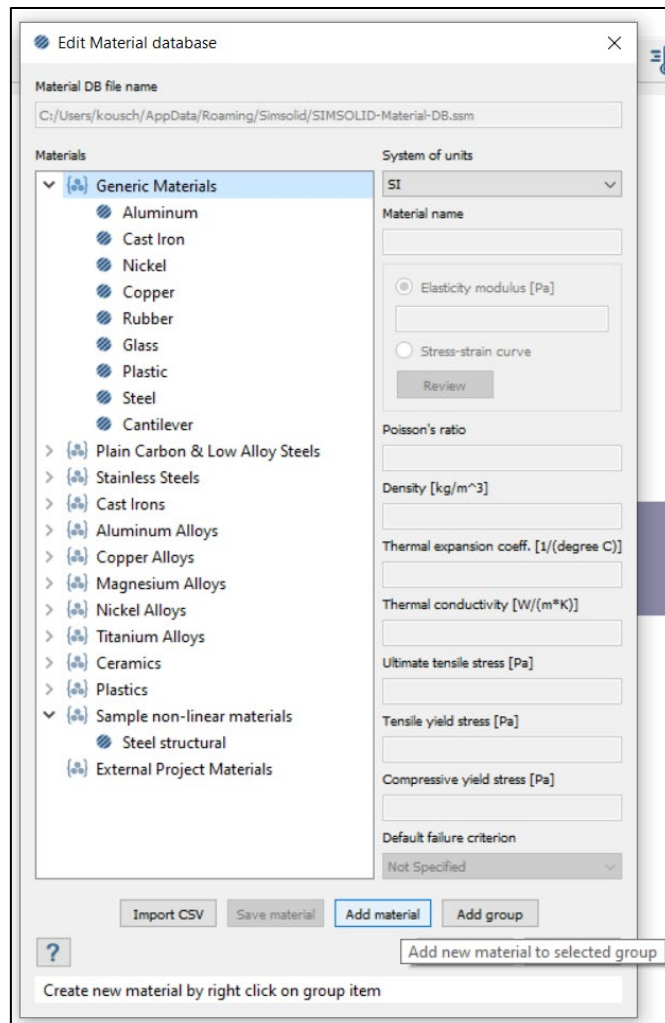
- Add 'Steel (EN-GJS-400-18)' material in the Material database in Settings menu.



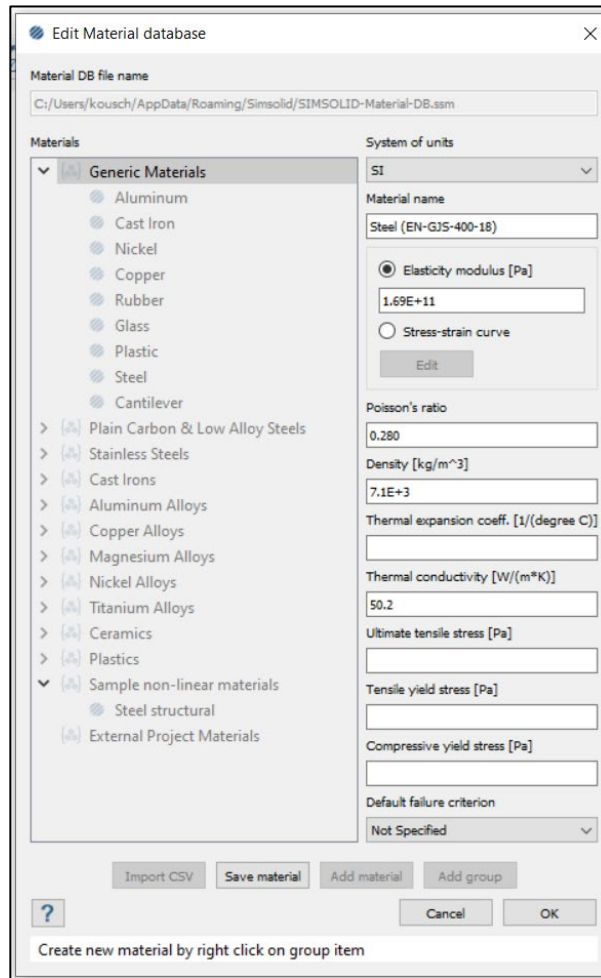
- Click on 'Edit current'



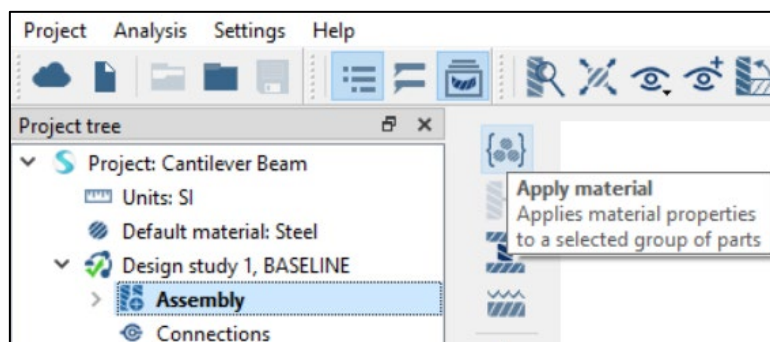
- Click on 'Generic Materials' and then click on 'Add material'



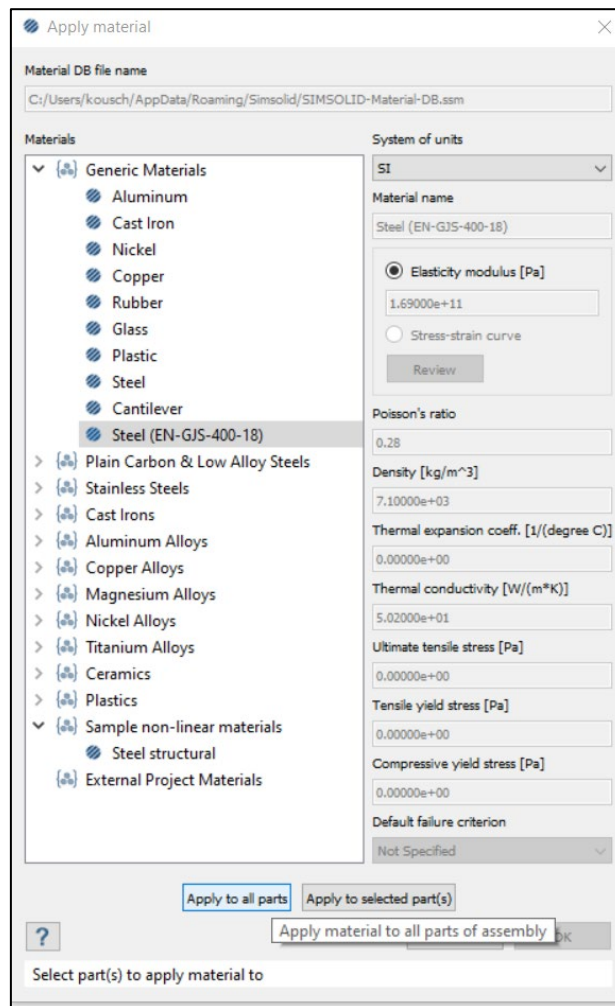
- Enter the material properties as shown below, then ‘Save material’ and click OK



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

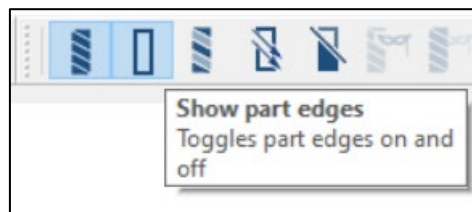


- Select the created material, click on 'Apply to all parts' and click OK

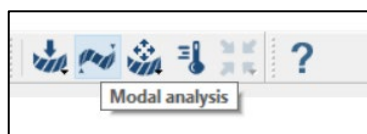


Step 3: Set up Modal Analysis and apply boundary conditions

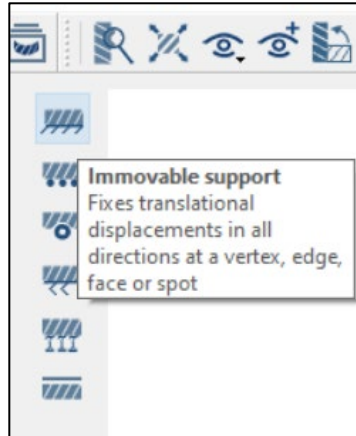
- First, make sure that the partition lines are visible by clicking on 'Show part edges'



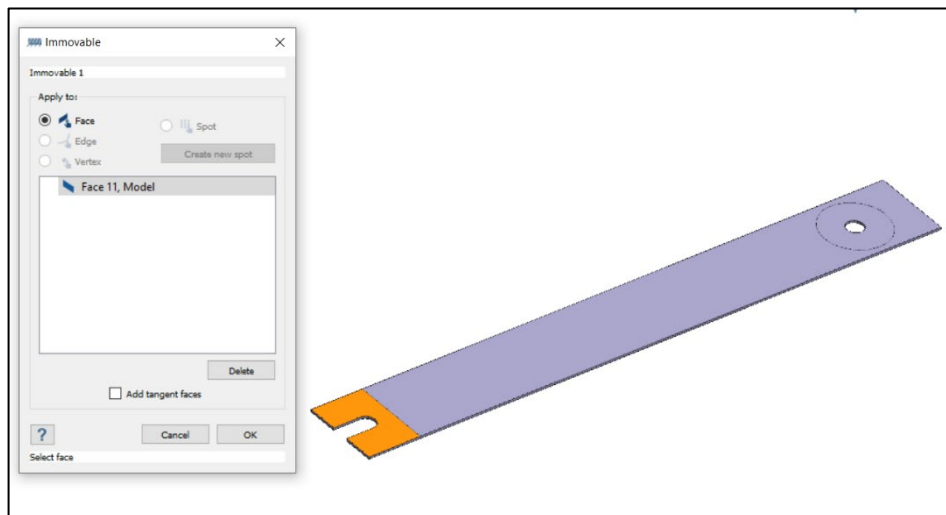
- Click on 'Modal analysis' icon in the horizontal toolbar at the top



- Set the number of normal modes to 10 and click OK
- Click on 'Immovable support' icon as shown below

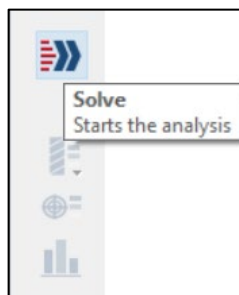


- Click on the highlighted part of the cantilever beam to apply the support.

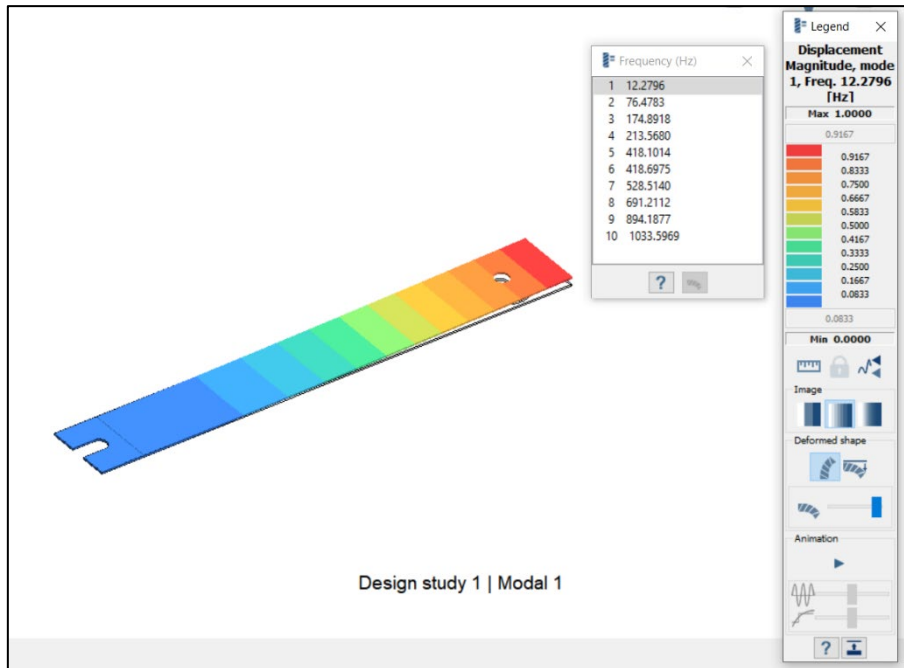


Step 4: Run Modal Analysis

- Click on 'Solve' icon to run the modal analysis





Step 5: View the Results



The fundamental frequency (1st frequency) for the cantilever beam is approximately 12.3 Hz, 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.