



Finite Element Analysis

For Stage 6 Engineering Studies, this experiment covers the following syllabus points: P/H1.2, P/H2.1 and P/H3.1

With advances in computing power and mathematical modelling, most materials and structural testing is now conducted using Finite Element Analysis (FEA) software. FEA first breaks components into small units like triangles and cubes, forming a mesh (picture below) which divides a complex system into discrete areas. Equations such as the heat equation or in our case the Euler-Bernoulli beam equation can then be applied to these finite elements and upon completion, compiled to represent the behaviour of the system.

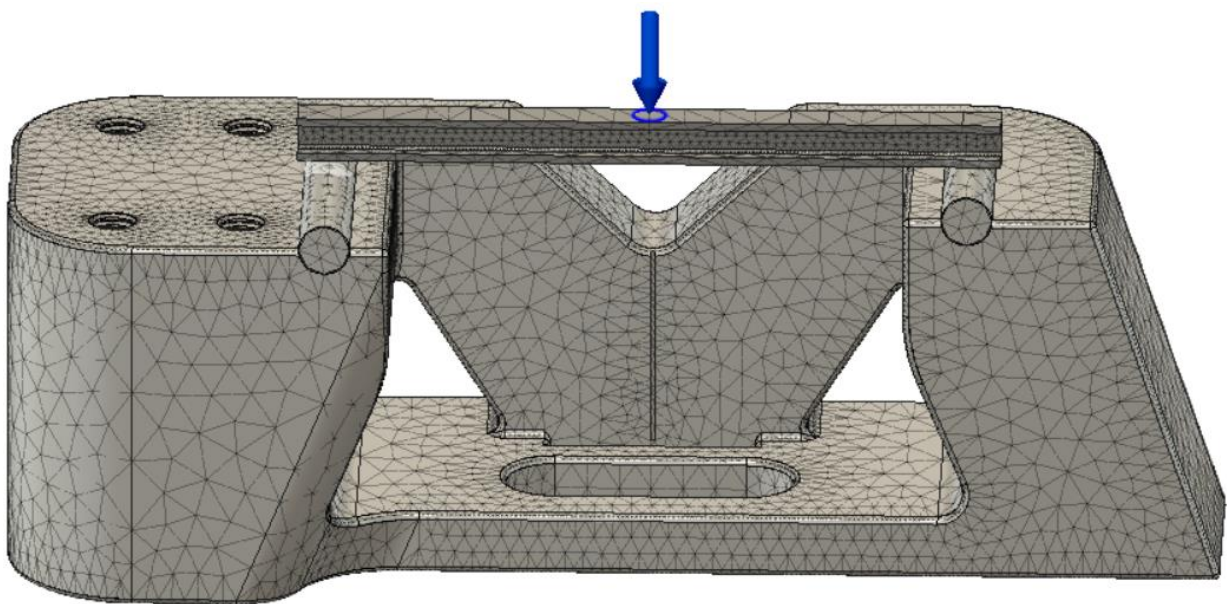


Figure 1: Meshed view of the 3 point bending test

While FEA is a powerful tool which saves a large amount of money and time compared to physically testing early prototypes, it is very important to set the correct conditions to simulate the actual operating environment. This is done by applying loads, constraints between components, contact sets and materials properties which all feed into the algorithms. For example the simulation in figure 1 below obviously doesn't reflect the results we would expect (figure 2) if we physically tested the sample (blue represents a load), however incorrect results can be far more subtle.

FEA set ups

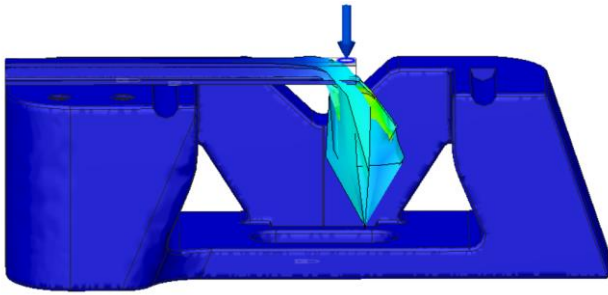


Figure 2: FEA stress analysis of a cantilever bending test showing unrealistic results

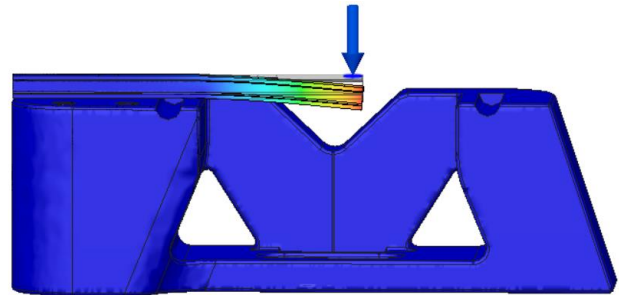


Figure 3: FEA stress analysis of a cantilever bending test showing the predicted results

Another consideration when running FEA is the level of consistency and isotropy of the materials being tested. Many metal treatments like cold working and hot forging can cause lead to the alignment of grains resulting in anisotropic properties which the FEA software won't take into account. Furthermore, fibre reinforced composites like carbon fibre reinforced polymer (CFRP) are incredibly anisotropic and require specialised software which allows the fibre layer (ply) directions to be inputted.

Method:

In this experiment we will use FEA to model bending tests in a range of materials and configurations including 3 point tests, 4 point tests and cantilever tests. The FEA environment will be set up to subject the beams to typical forces and stresses experienced during application. The results will provide information about the mechanical properties of the material of that specific cross-section. Comparisons can then be drawn between the same material with different cross-section geometries.

What you need:

- A student version of Autodesk Fusion 360 – free download [here](https://www.autodesk.com/products/fusion-360/students-teachers-educators) (link address: <https://www.autodesk.com/products/fusion-360/students-teachers-educators>)
- The Fusion 360 file 'Bending_test_UNSW_School_of_Materials_Science__Engineering.f3d' which you can access in the downloads section of from [Thingiverse](#)
- An internet connection will also speed up the testing process as you can conduct the stress analysis on the cloud and run multiple simulations concurrently

Activity 1: Materials testing

This test will make use of the two models, 'Standardised specimen 3 and 4 point test' and 'Standardised specimen cantilever test'

1. Loads:

- Navigate to the 'Simulation Environment' and then activate the simulation you wish to run (see **Figure 4** for an example of the activated 4 point test within the 3 and 4 point test environment)
- Select the loads you would like to edit and then input an appropriate force (e.g. 20N)

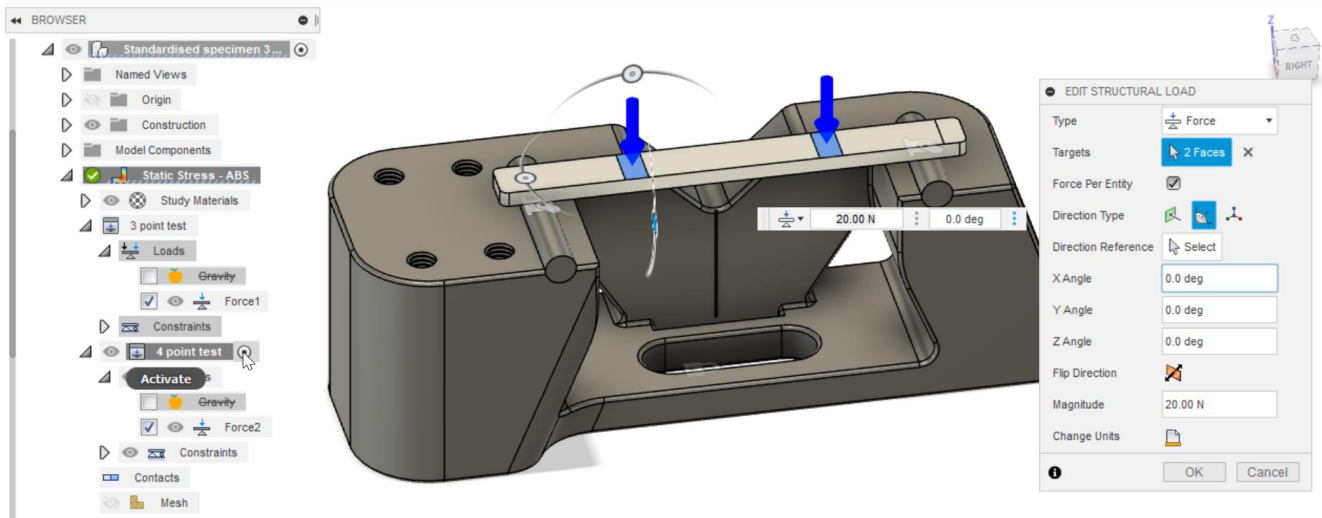


Figure 4: Application of a loads to the 4 point bending test

2. Materials

- Select the 'Study Materials' menu and then select edit. In the resulting window navigate down to the component called standardised sample and then edit the 'Study Material'. You will have a wide selection of materials available – select ones that you have used in the real life bending test or those that are interesting to you

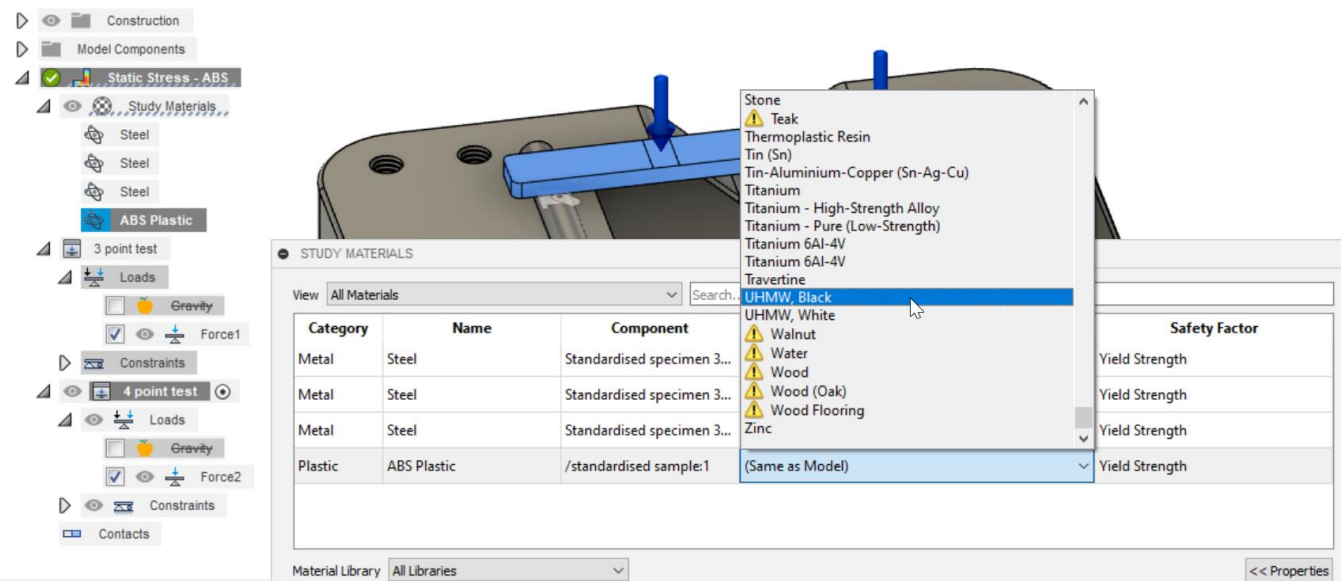


Figure 5: Choosing a new material for the standardised sample

3. Simulation

- a. Once you are happy with the loads and materials applied, click the 'Solve' button. When the computation is complete the results of the test should be displayed. Using the toggle to the left of the scale bar you can explore the range of outputs provided including displacement (shown below), safety factor and stress

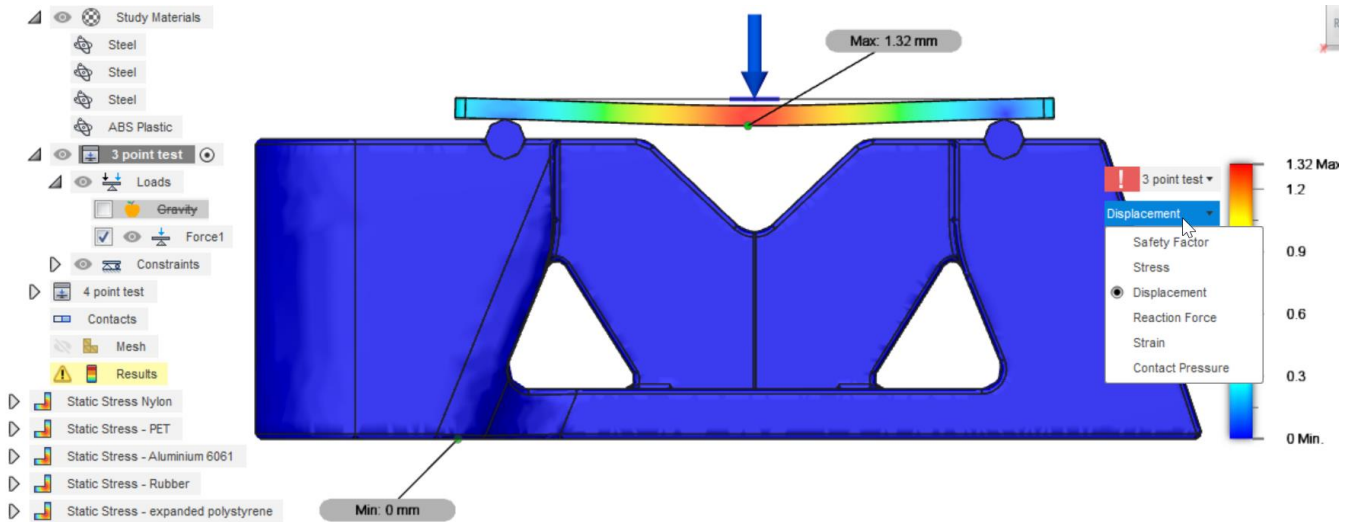


Figure 6: Results section, displaying the displacement of the standardised specimen in the 3 point test

4. Comparison

- a. Note down the displacements of any material you tested in the first lesson plan and then enter in the those from the simulations your ran

Material	Physical test displacement (mm)	FEA simulation displacement (mm)

Discussion:

Question time!

Question 1:

What could explain the differences in values between your physical test and the simulation?

Question 2:

How do differences between real world and simulation values affect design choices?

Question 3:

If 3d printed samples with varying print orientations were used, would you expect differing results (if yes, then why)?

Question 4:

Can FEA analysis be used for all materials including metals, composites, ceramics etc? Explain your answer:

Activity 2: Simulation set up (advanced):

1. Load the file and familiarise yourself with the available components in the 'Design' workspace
2. Switch to the simulation workspace found in the top left corner (Figure 7)

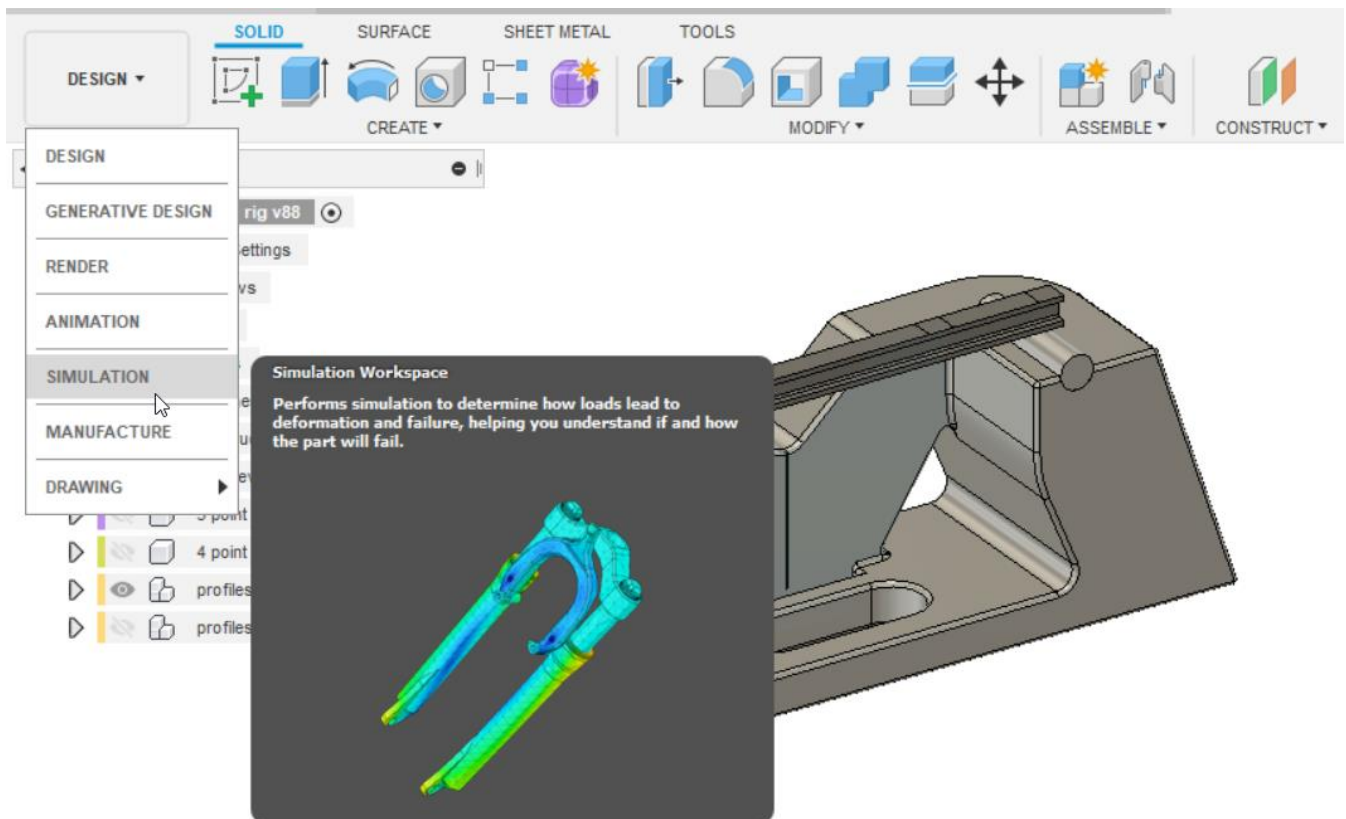


Figure 7. Activating the 'Simulation' environment

3. When setting up an FEA it is important to remove all the components which do not contribute or will interrupt the results. This can be done in the simplify environment and in this file, we have two main environments set up, which include the 3 and 4 point tests which require the sample to move freely on two dowels in the middle of the testing rig and the cantilever test which sees the sample bonded to the larger surface of the testing jig. You can access each environment by activating them and checking out the selected models involved. From here you can also see the individual tests done within the environment
 - a. To set up a new environment, select the *Simplify* option from the top ribbon, near to the left. From here, remove all the components unnecessary for the simulation, including the 3 and 4 point applicators and the cantilever plate (we will use proxies for these components)
 - b. If you are setting up a cantilever test, remove 'profiles 1' and then remove all of the unneeded beams (just leave the one you want to work on). You can use the same process for the 3 and 4 point bending test, however you will be removing 'profile 2'
 - c. Once you are happy with the environment, close the simplify environment using the top right button on the ribbon

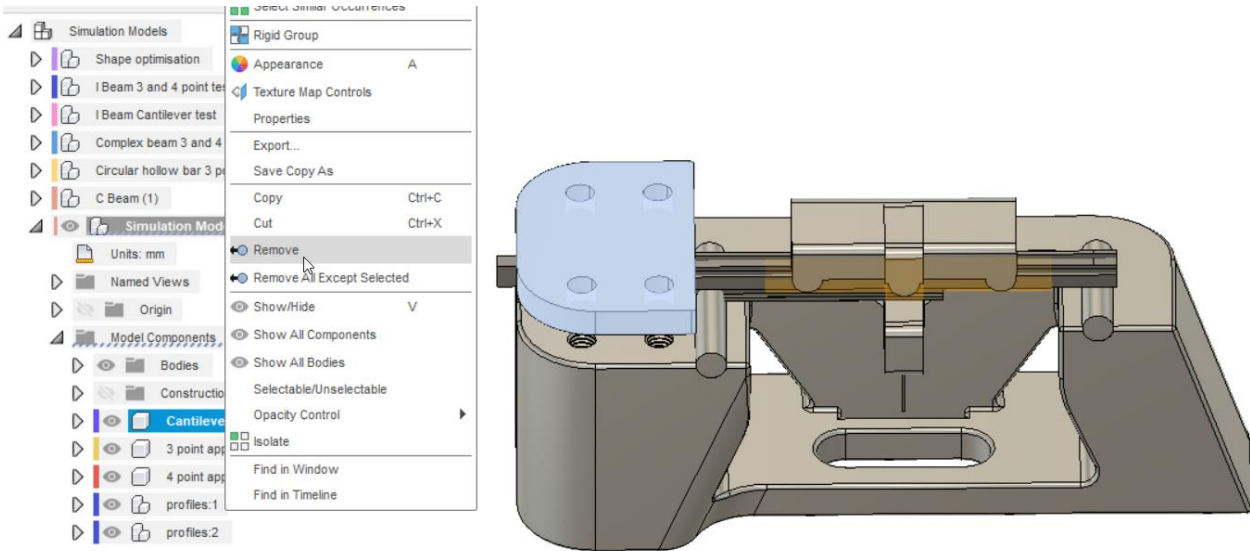


Figure 8: Remove all of the unnecessary components by right clicking on the in the browser or window and then selecting remove.

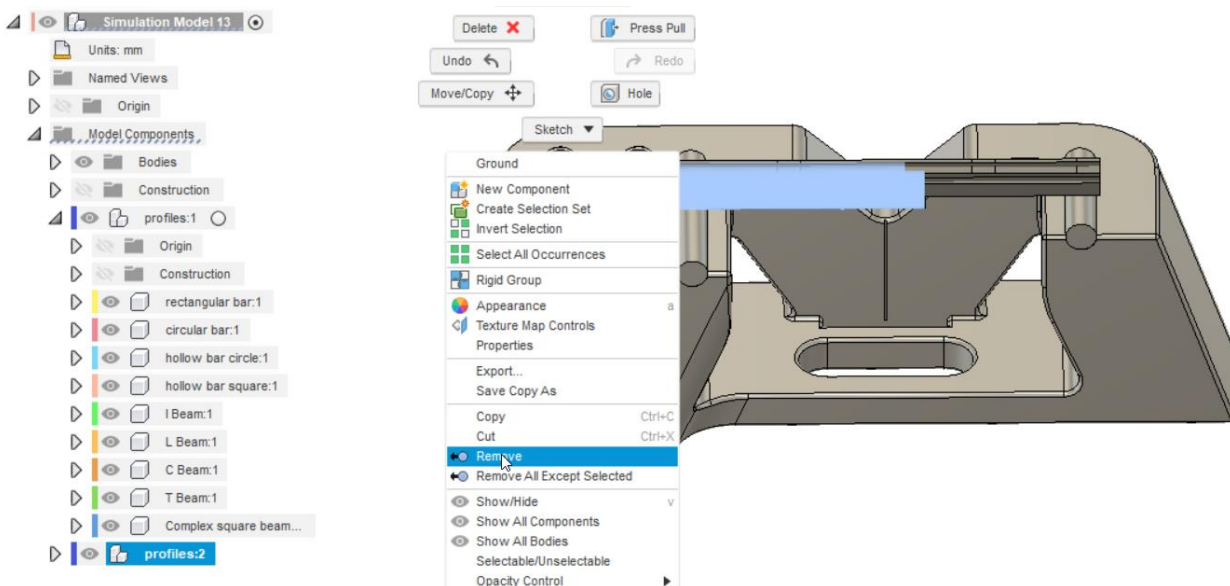


Figure 9: If setting up a cantilever test, remove 'profiles 1'. If you are setting up the 3 or 4 point bending test remove 'profiles 2'.

- To set the degrees of freedom and algorithm parameters we now enter the constraints and contact sets. The first constraint is a fixed constraint on the base of the model to ground it (Figure 10):

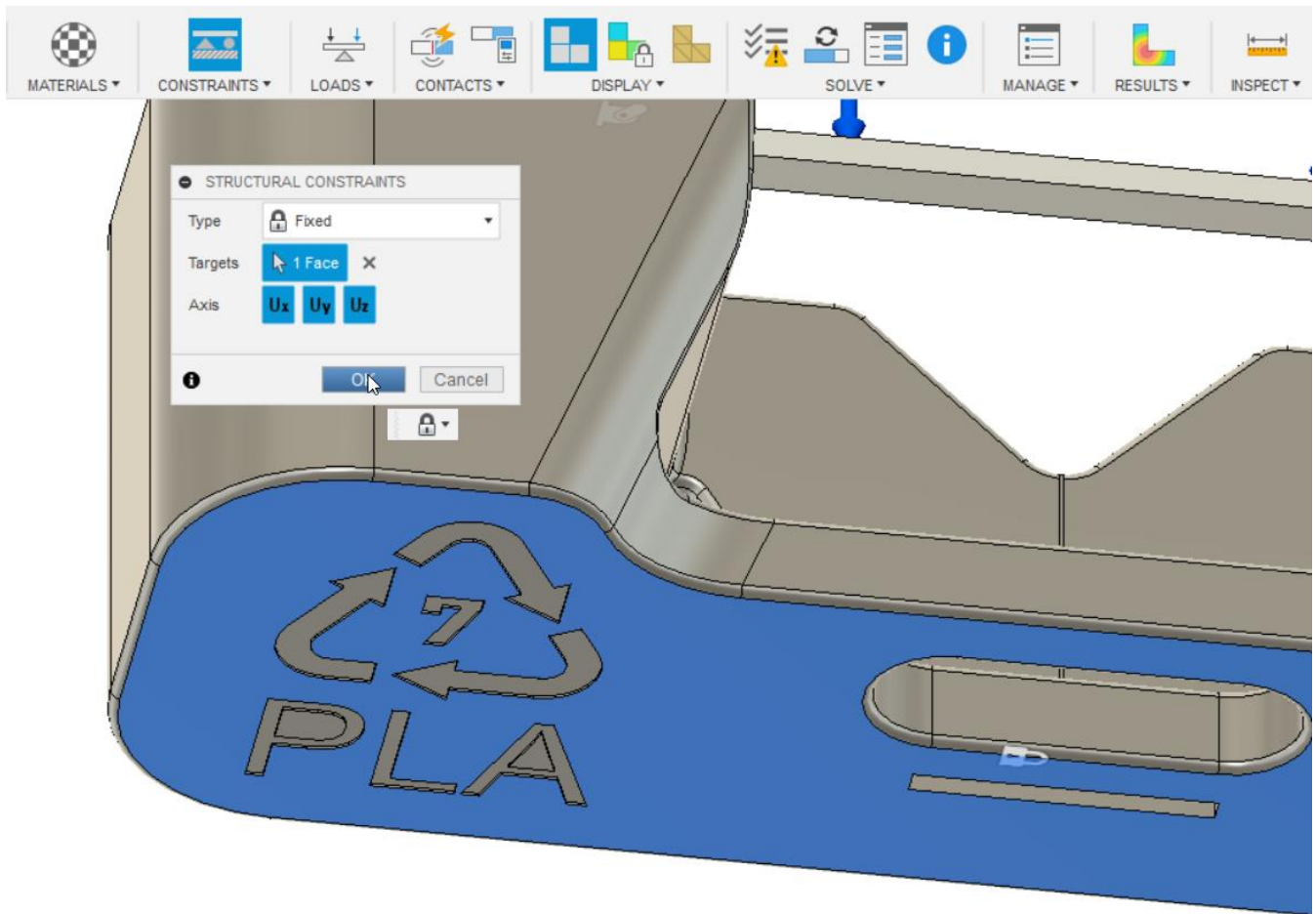


Figure 10: A fixed constraint is used to fix the bottom of the model and provide a point of reference.

- a. **3 and 4 point bending tests:** These require the beam to be able to slide over the dowels so you will need to set the constraints of the dowel faces as pin constraint

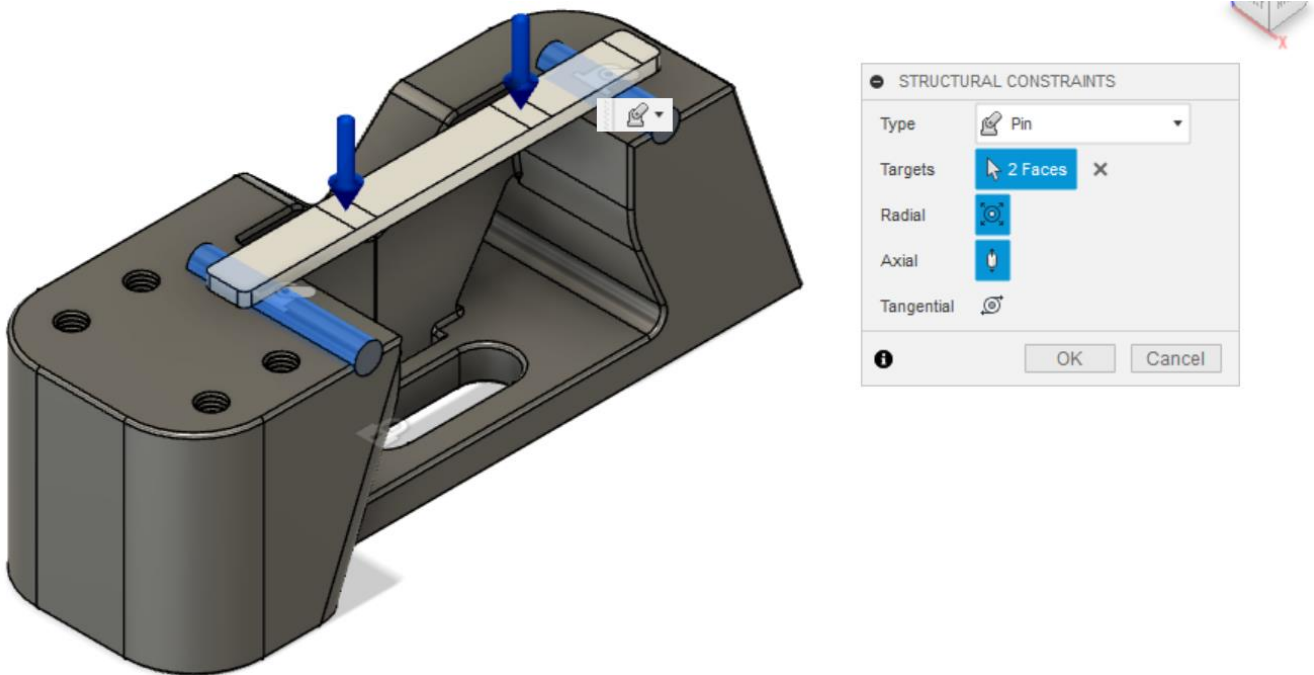


Figure 11: A pin constraint is added to the dowels of the 3 and 4 point cantilever test

- 5. **Contact sets:** Once the constraints are added, click the 'Automatic Contacts' button to set up the constraint set
 - a. **3 and 4 point bending tests** - Setting up the contacts sets will not pick up the sliding behaviour of the specimen over the dowels, so you will need to modify them using the 'Manage Contacts' menu. Select the contacts connecting the sample to the dowels and change them from 'Bonded' to 'Sliding'

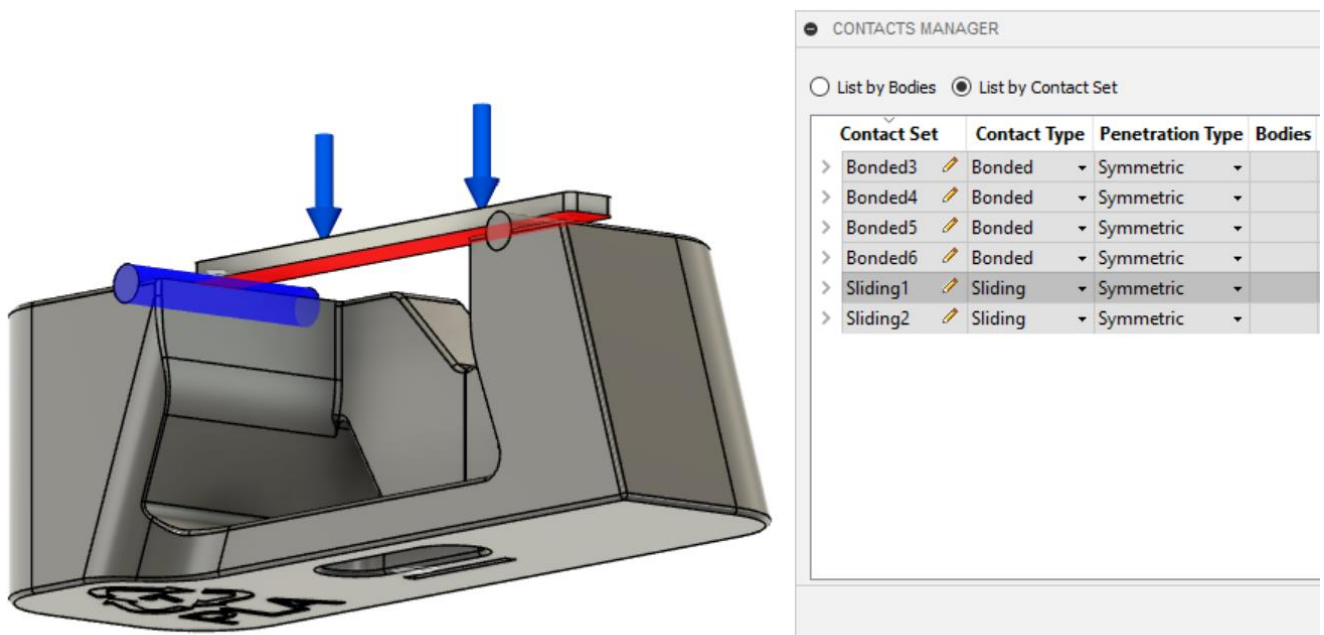


Figure 12: Sliding contacts are set for the surfaces where the specimen touches the dowels

6. **Adding loads:** We will be adding a static force to the model in varying locations to simulate the stresses experienced in a 3 point, 4 point and cantilever bending test
- 3 point bending test** - select the 'Loads' menus and then select the middle face and change the workflow to a directed force. From here, change the size of the force being applied to 2.5mm and then set the load (e.g 20N) coming from the above the specimen

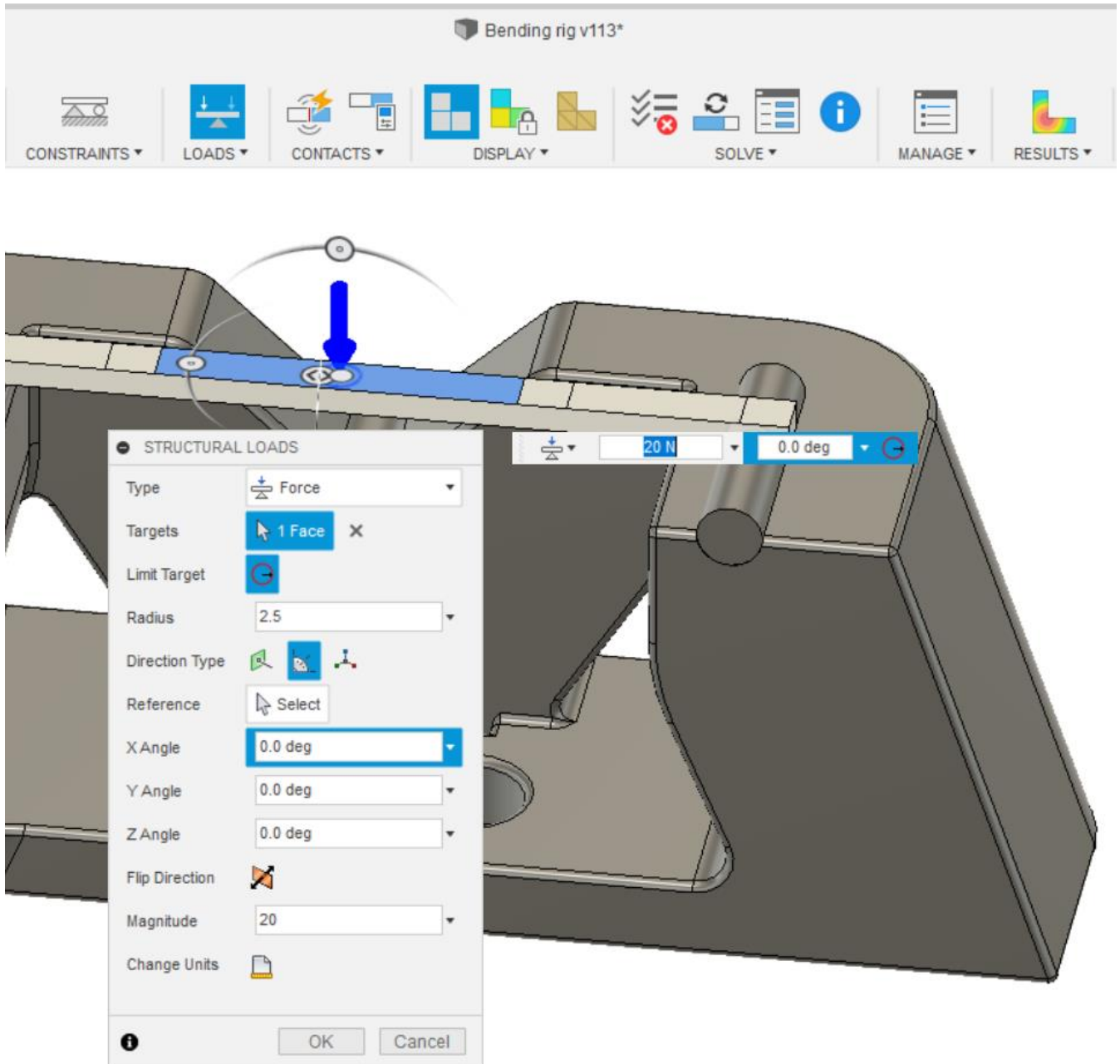


Figure 13: Application of a targeted 20N load for the 3 point bending test

- b. **4 point test:** Repeat the steps above in the 3 point test, however this time select the two middle faces and you will not need to limit the target (see below)

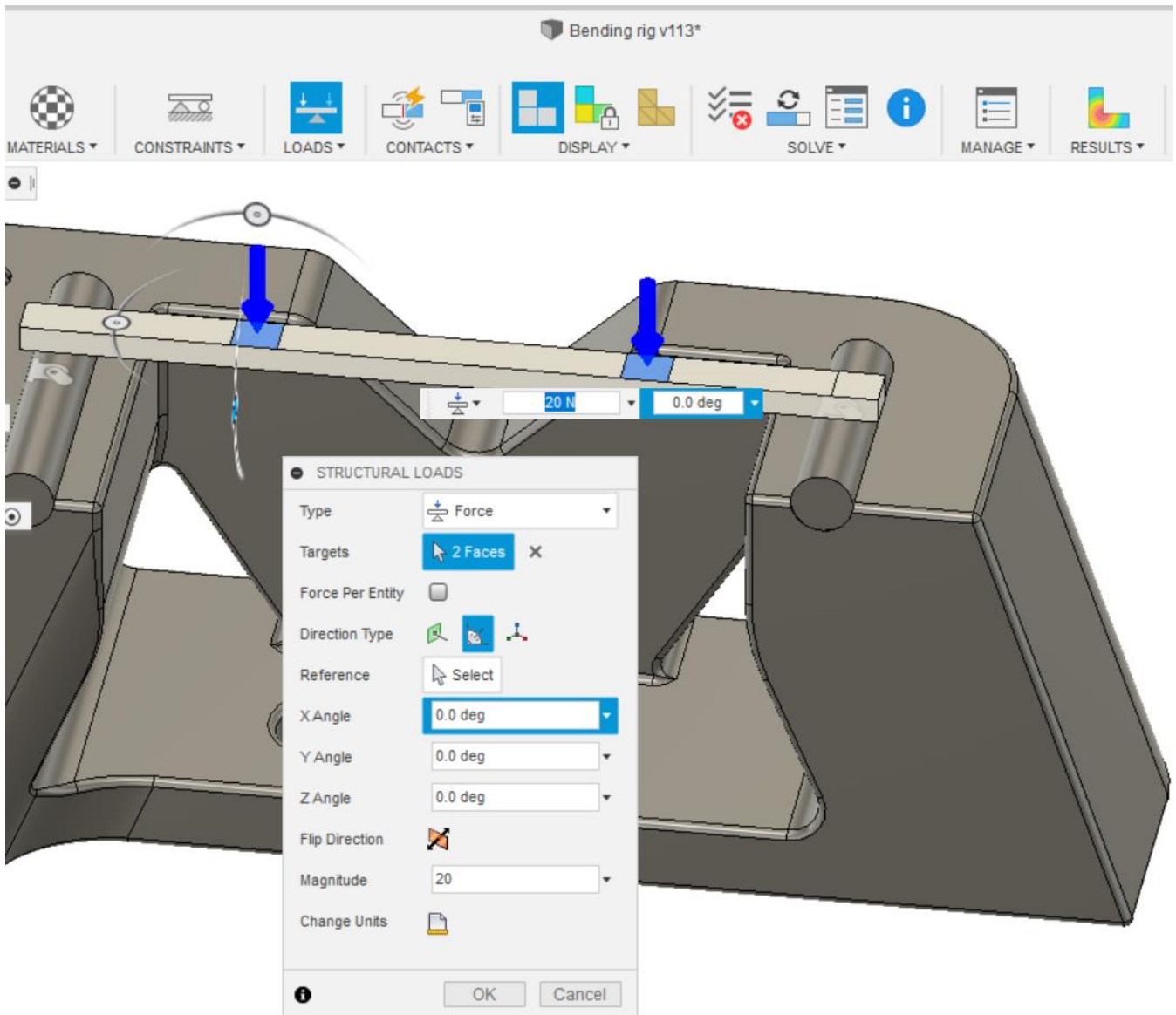


Figure 14: Loads are placed on the middle faces of the specimen in the 4 point bending test

- c. **Cantilever test:** Activate the cantilever test environment and then repeat the steps above to add a load, but this time to the end most face

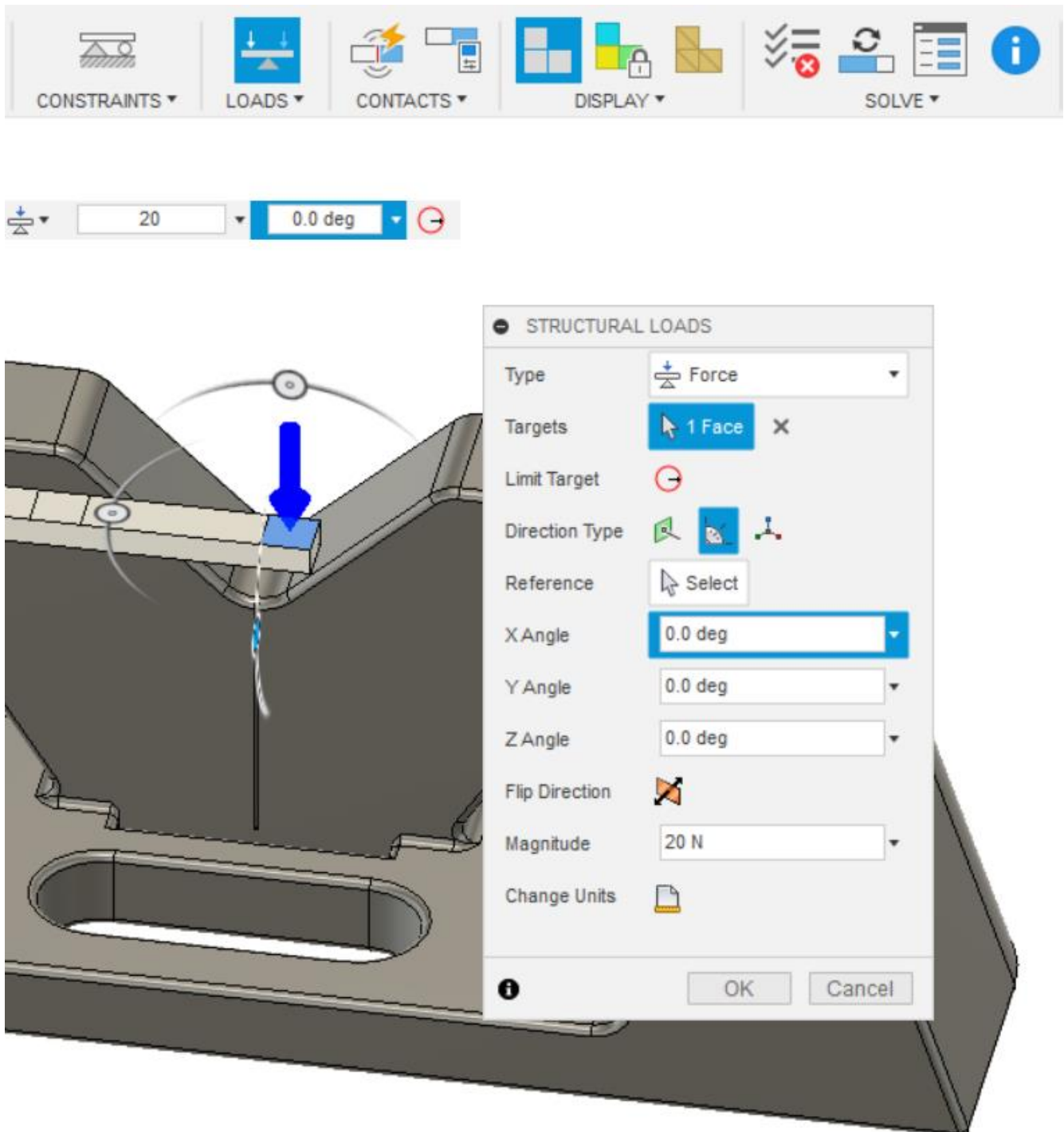


Figure 15: A load is placed on the end face of the specimen in the cantilever bending test

7. Once you have completed the set up, and added the desired materials (found in the first activity) you are ready to compute the results!

Questions:

- What would have happened if you maintained all of the contacts as bonded in the 3 and 4 point bending tests (Figure 12 for a prompt)?
- Can all geometries and materials types be tested in an FEA environment?

Follow up tasks:

- If you are familiar with the design environment of Fusion 360, set up the remaining samples and perform bending tests on these
- If you still want some more, design new samples based around available items (paddle pop sticks etc), run bending tests simulations on these and then compare the results to those of the real world items on the 3d printed bending test

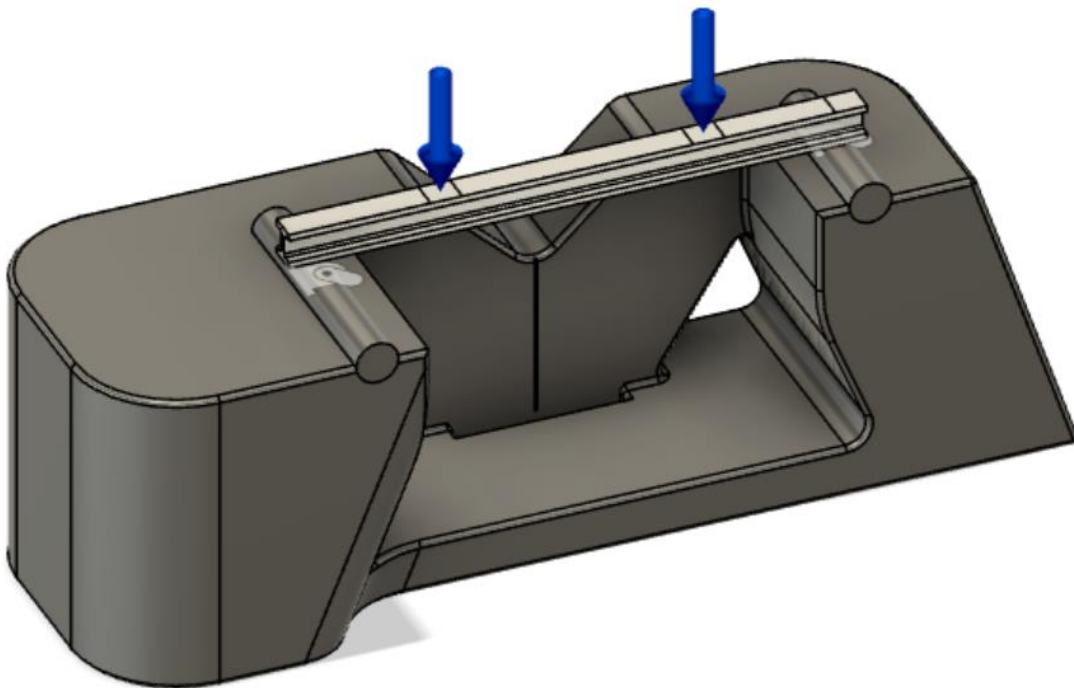


Figure 16: 4 point bending test set up of an I beam sample

Acknowledgements

This project builds on the open source 3 point bending test platform created by Stefan Hermann of CNC kitchen. You can download the original files at the link below and check out his YouTube channel for some awesome material science videos relating to 3d printing!

Original files at: <https://www.thingiverse.com/thing:3142077>

Stefan's YouTube channel: <https://www.youtube.com/watch?v=te0Wwf7Dxj4&feature=youtu.be>