

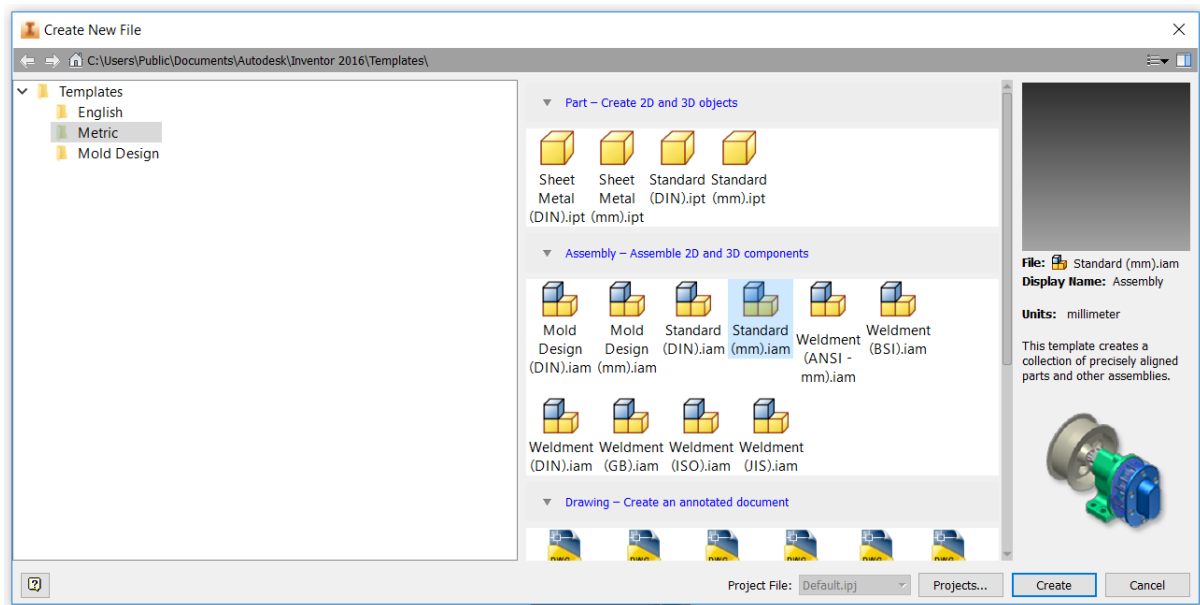
# Performing a Strain Simulation on the Crossboard Sample Holder Using Autodesk Inventor 2016

## Introduction

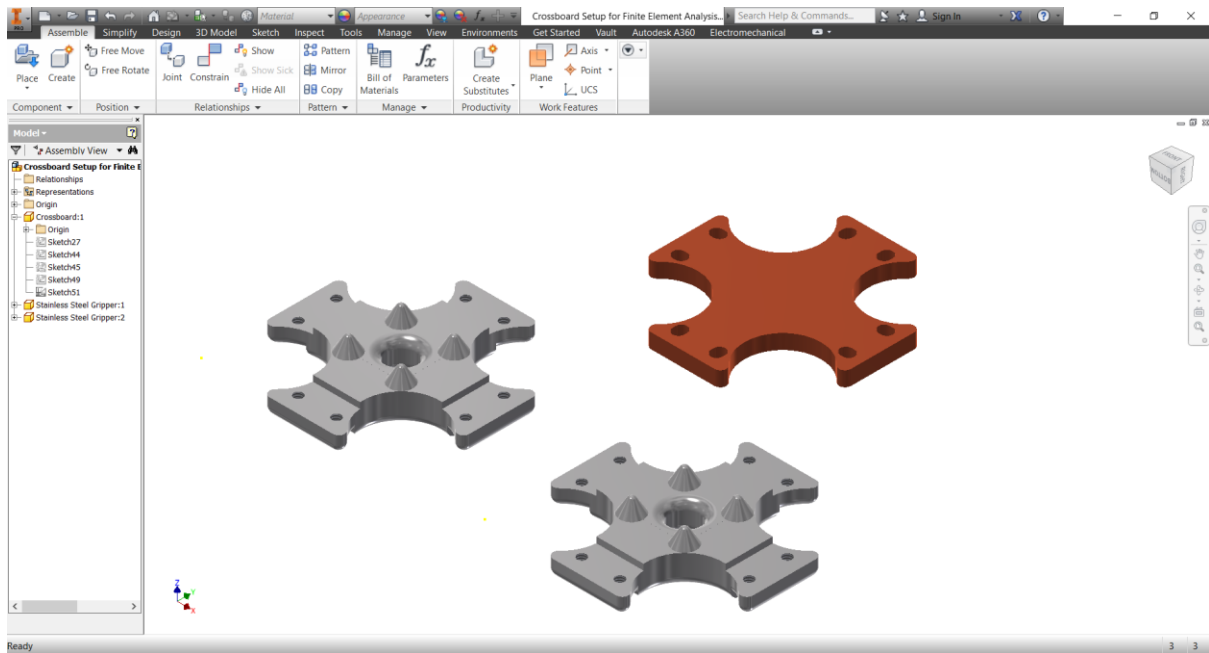
This file provides a guide for reproducing the simulation results depicted in Fig. 2 of the paper titled “Biaxial Strain Measurements of  $J_C$  on a (RE)BCO Coated Conductor”. The simulations are performed using Autodesk Inventor 2016 CAD software which has a built-in Finite Element Analysis (FEA) module. An assembly file named “Finite Element Analysis Results- Figure 2” is provided if the user wishes to bypass steps 1-10.

## Method

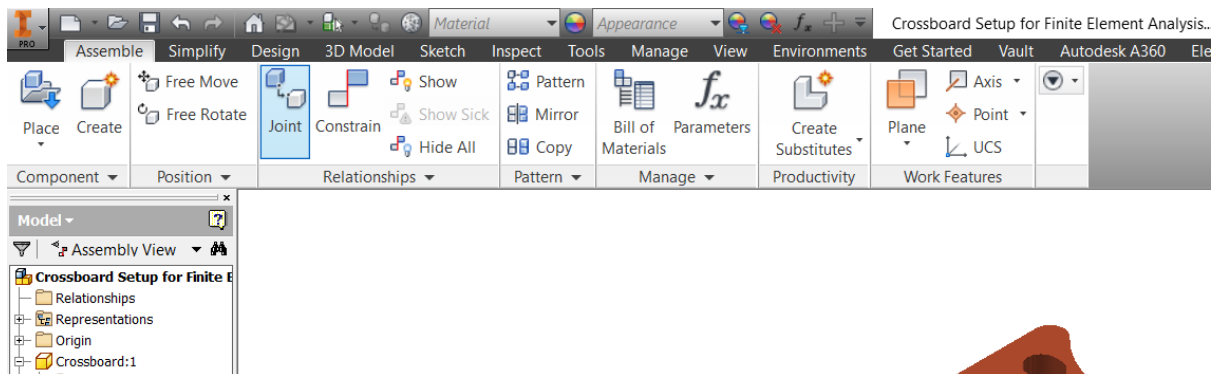
1. Create a new Assembly (.iam) file in Inventor 2016

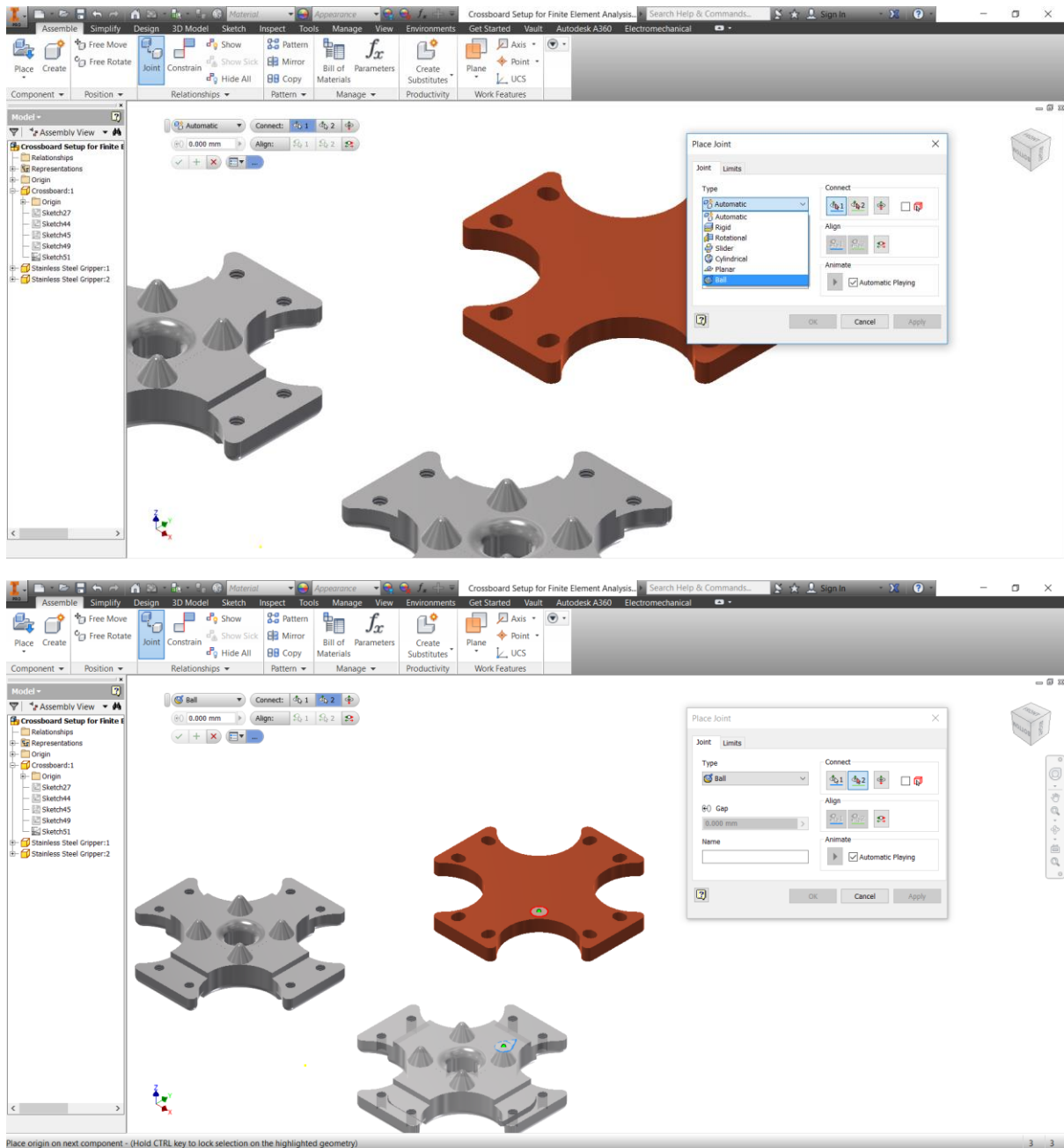


2. Insert the Crossboard and two copies of the stainless-steel grippers:

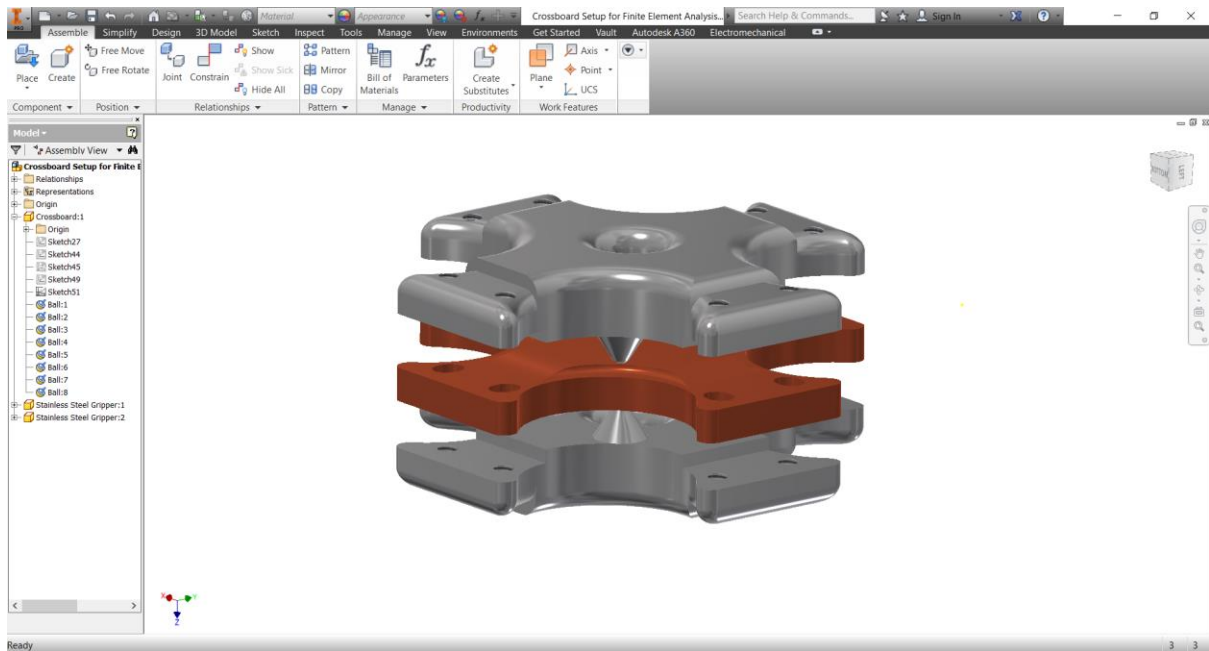


3. When the cursor hovers over the crossboard, 4 small red circles will become visible on each side. These are the locations where the teeth on the grippers are attached. Attach the teeth using a 'Ball Joint':

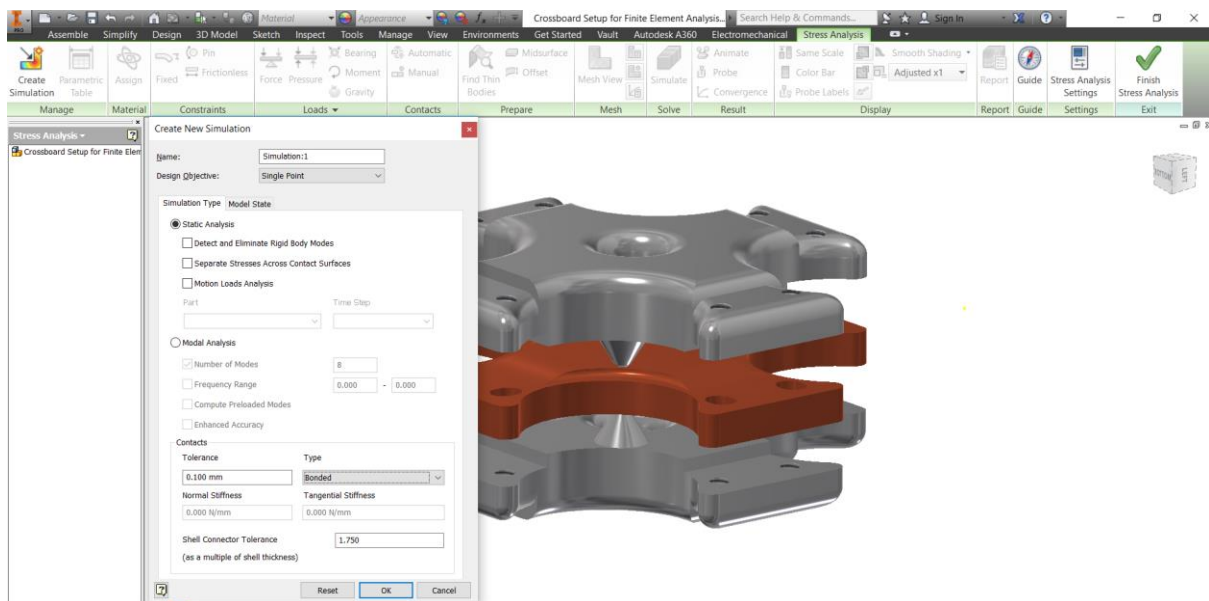




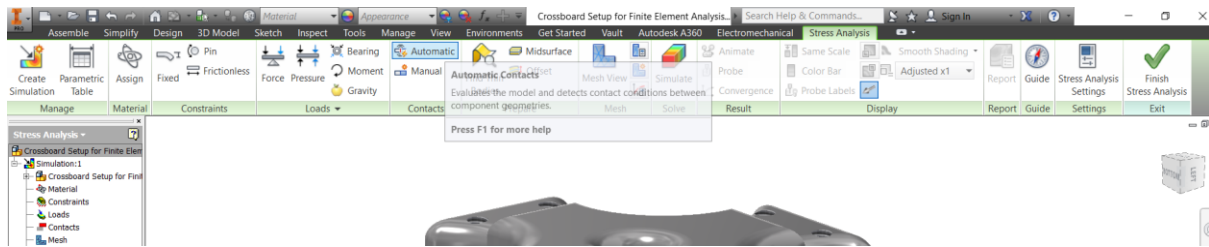
Ball joints ensure that the crossboard can bend around the teeth correctly. Change the alignment of the joints until the grippers are aligned with the crossboard correctly. The finished assembly should look like that shown in Fig. 1 in the paper:



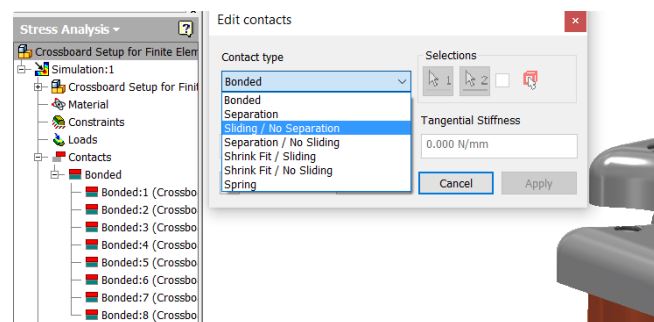
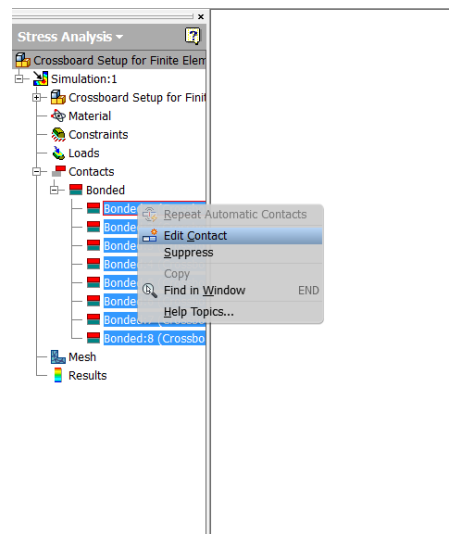
4. Go to the 'Environments' tab and click 'Create New Simulation'. Leave the simulation settings as default (picture below) and click OK:



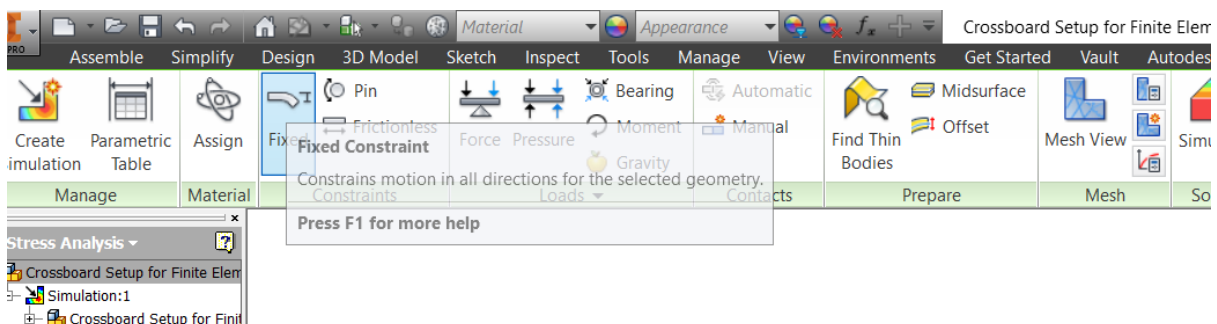
5. We now need to add 'contacts' to the assembly. The 'contacts' inform the software of the connections between the teeth and the crossboard. Click 'Automatic' under the 'Contacts' sub-tab.



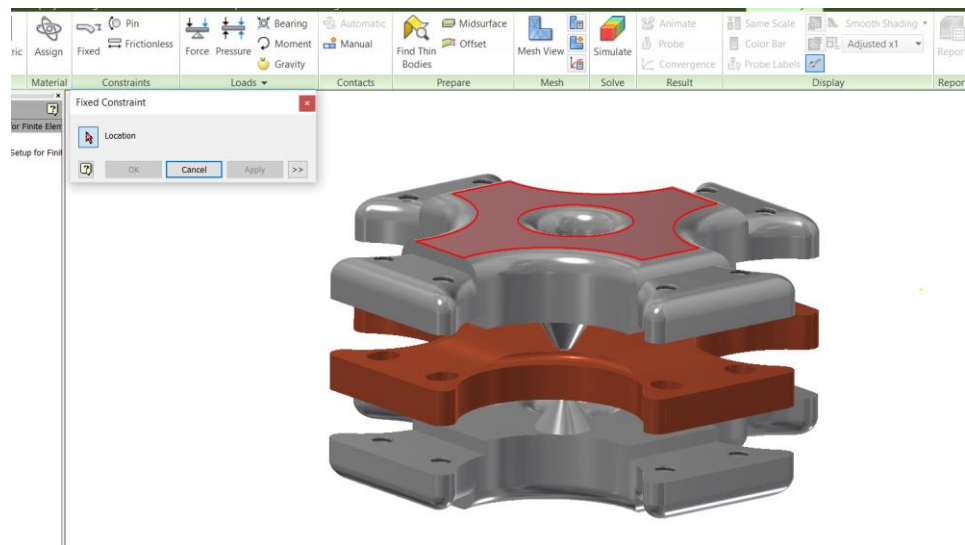
- The contacts section of the 'Stress Analysis' list on the left of the screen should now contain 8 'Bonded' contacts. These contacts should be changed to the 'Sliding / No Separation' setting, which ensures the crossboard can bend around the teeth without the crossboard becoming separated from the teeth.



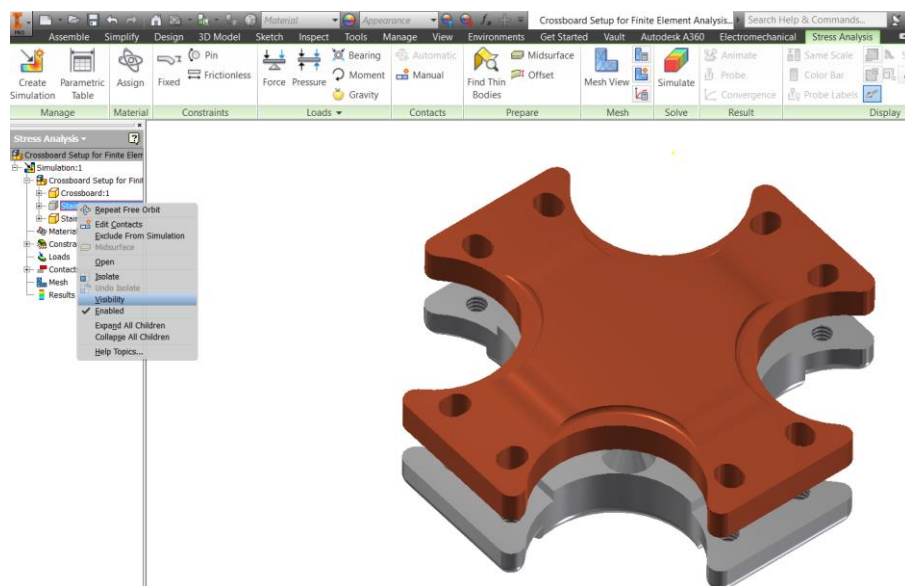
- We now need to add  $z$  constraints to the stainless-steel grippers. The constraints prevent the assembly from moving in the  $z$  direction when loads are applied to the crossboard. To do this, click the 'Fixed' button on the 'Constraints' sub-tab.



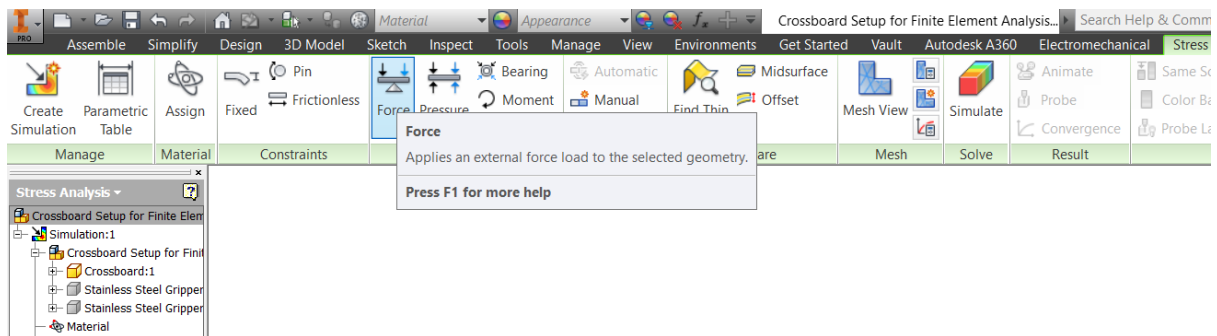
Apply the fixed constraints to the outer face of each gripper (red shaded area).



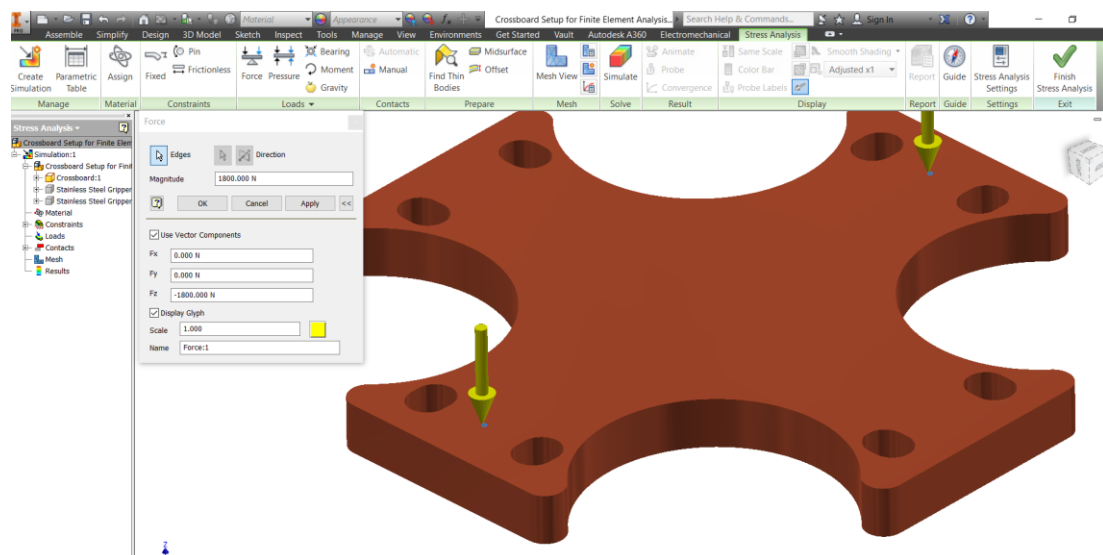
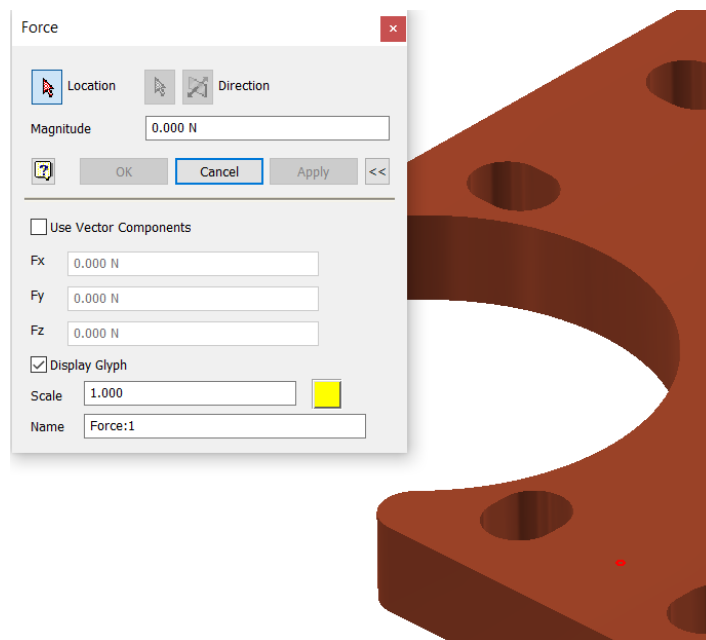
8. Now we must add the loads. First, turn off the visibility of the grippers:



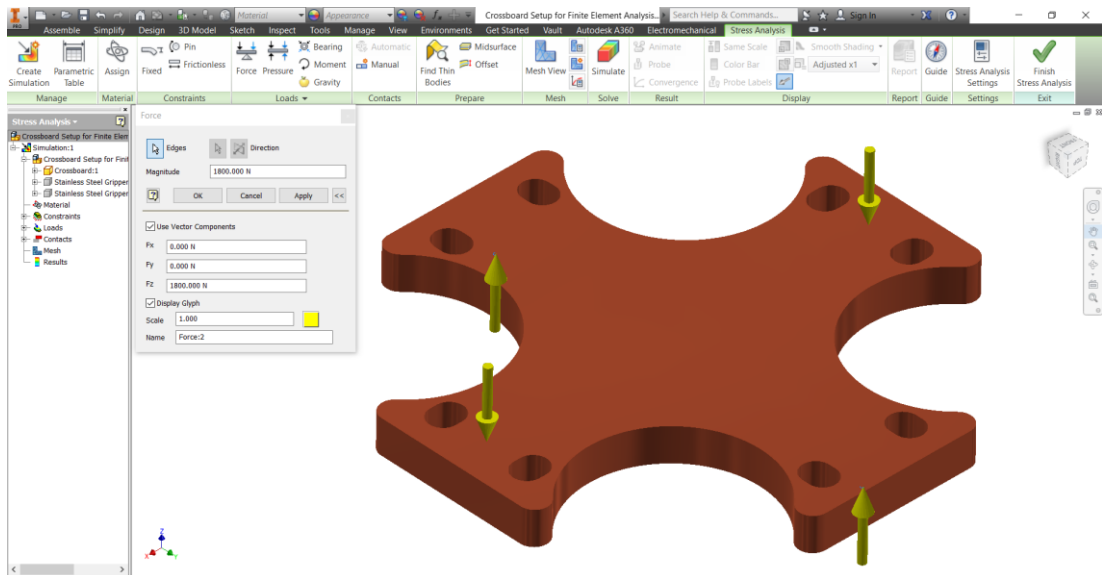
To add the loads, click 'Force' under the 'Loads' sub-tab.



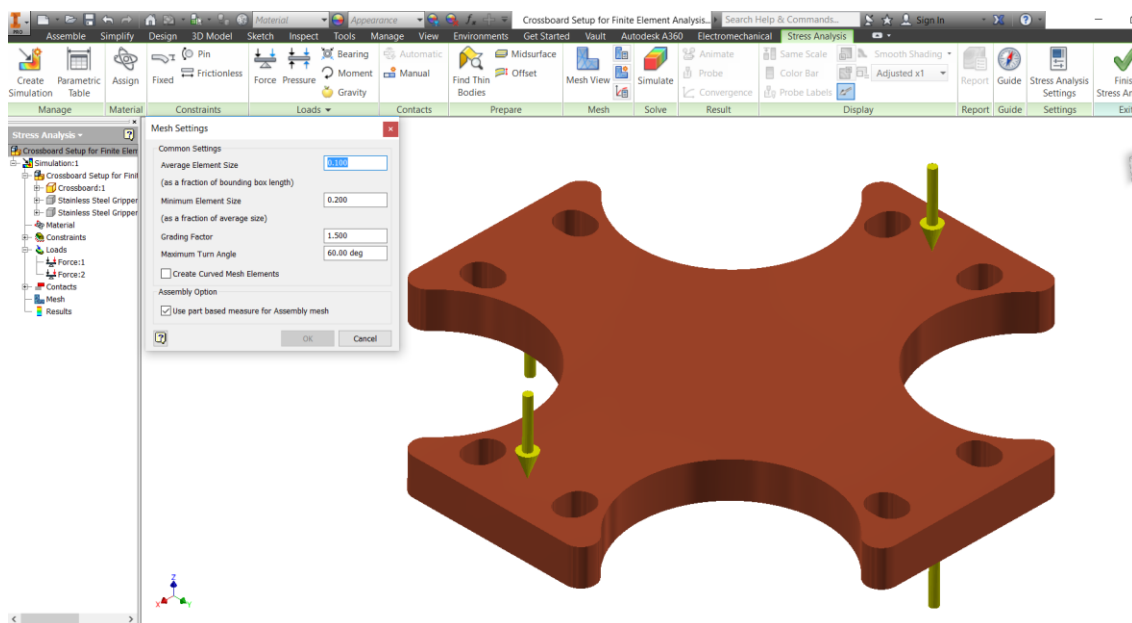
9. Apply the forces to the desired points (red circles). Apply vector forces of 1800 N at the four locations depicted in Fig. 2 in the paper.



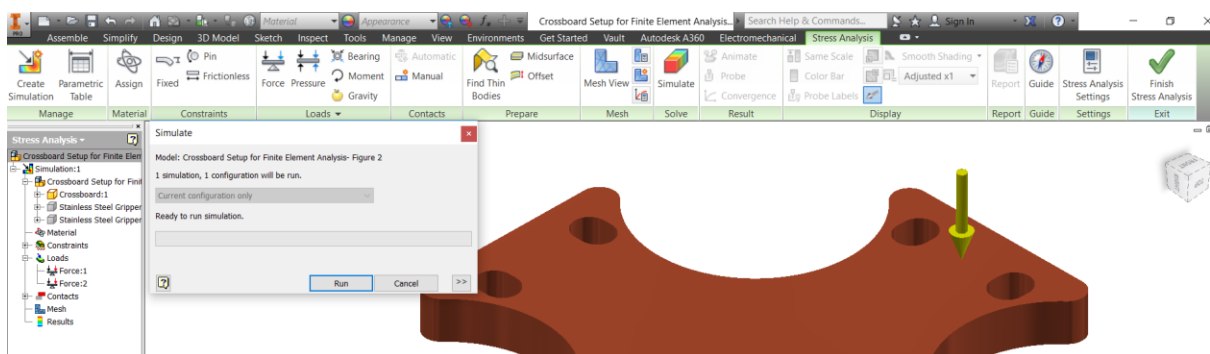




10. The final step before running the simulation is to configure the mesh settings. Change the settings so they match those in the picture below:

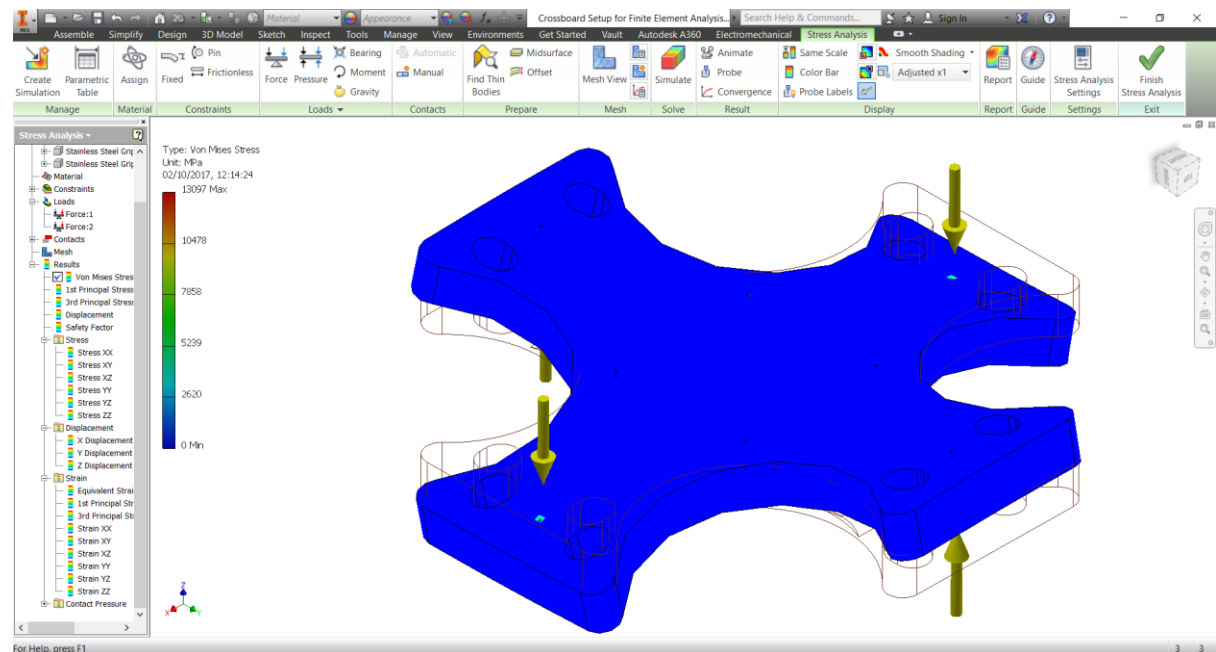


11. Run the simulation by clicking the 'Simulate' button in the 'Solve' sub-tab.

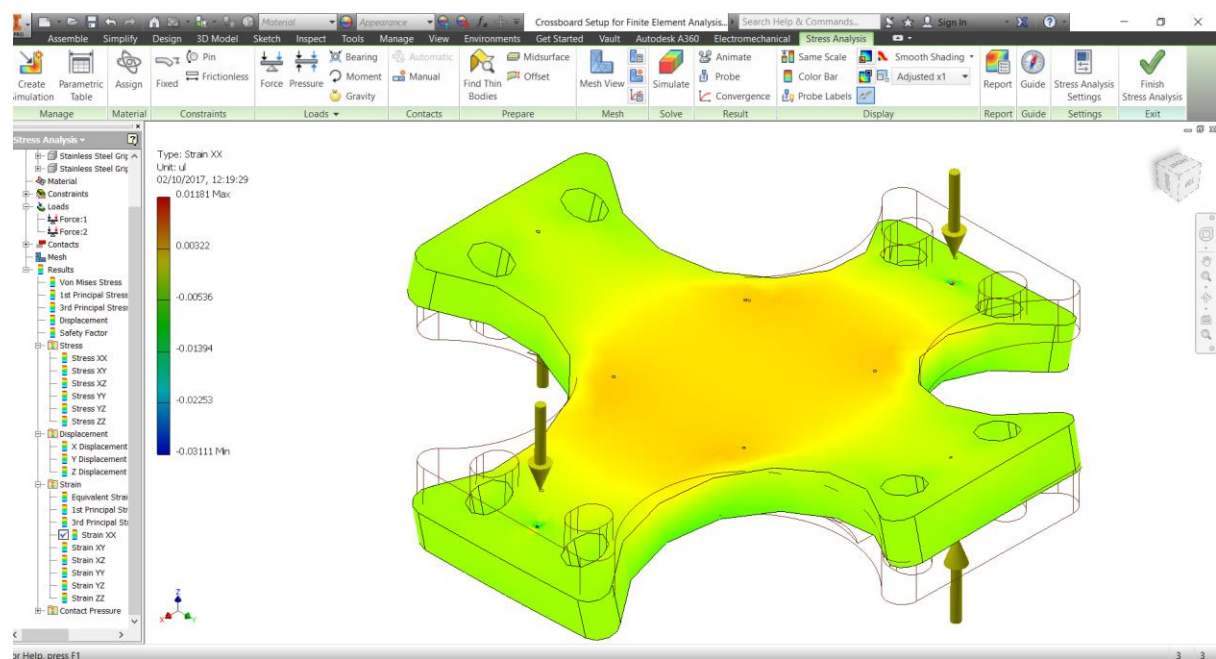




12. The results of the simulation should now be displayed.



Different stress and strain distributions can be viewed by selecting them in the menu on the left of the screen. We are interested in the strain distributions ('Strain XX' and 'Strain YY').



13. The colour bar settings can be changed from the 'Display' sub-tab. To view the results at a specific point on the crossboard, click 'Probe' under the 'Result' sub-tab.

