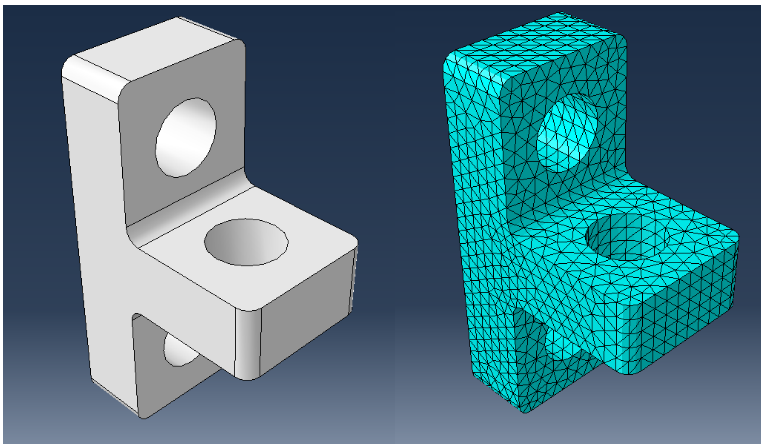
University of Nottingham
Department of Mechanical, Materials and Manufacturing Engineering

Computer Modelling Techniques

FE-WE2

Worked Example 2 using the ABAQUS FE Software

Material Properties, Boundary Conditions, Loads, Solving, Viewing Results

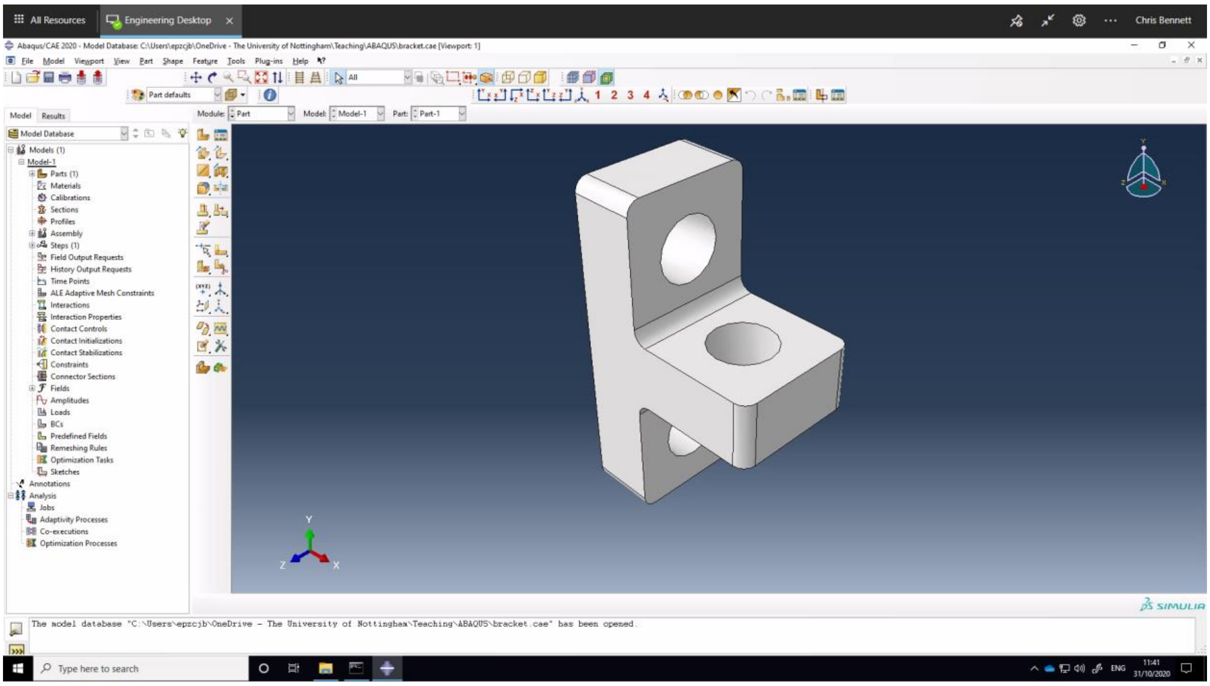


Options for accessing Abaqus:

Abaqus student edition is available from: https://academy.3ds.com/en/software/abaqus-student-edition (you will need to create an account – analysis is limited to structural models of up to 1000 nodes) or you can access via the Remote Desktop Web Client.

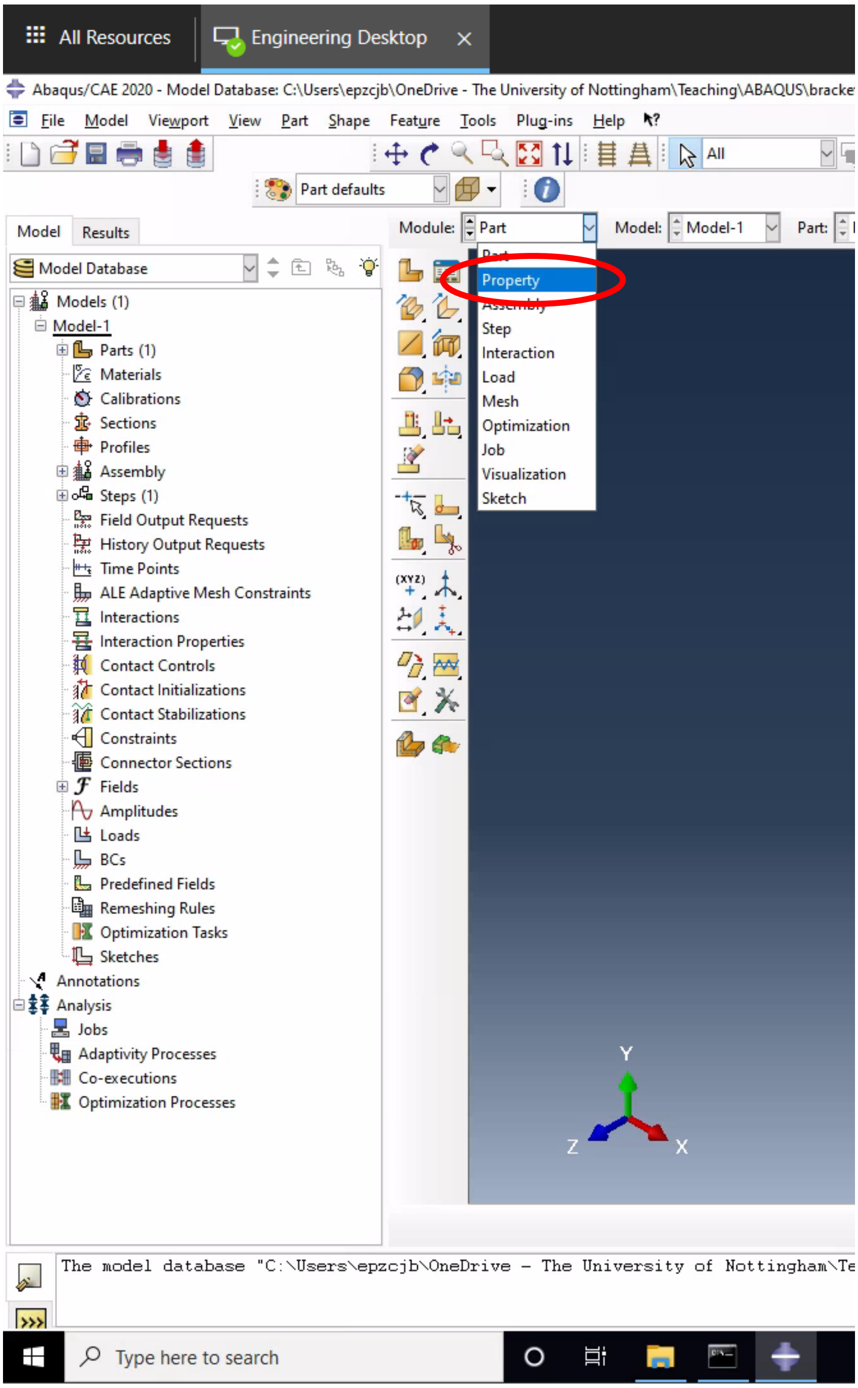
Steps:

1. Open the cae file from the previous exercise (*bracket.cae*)

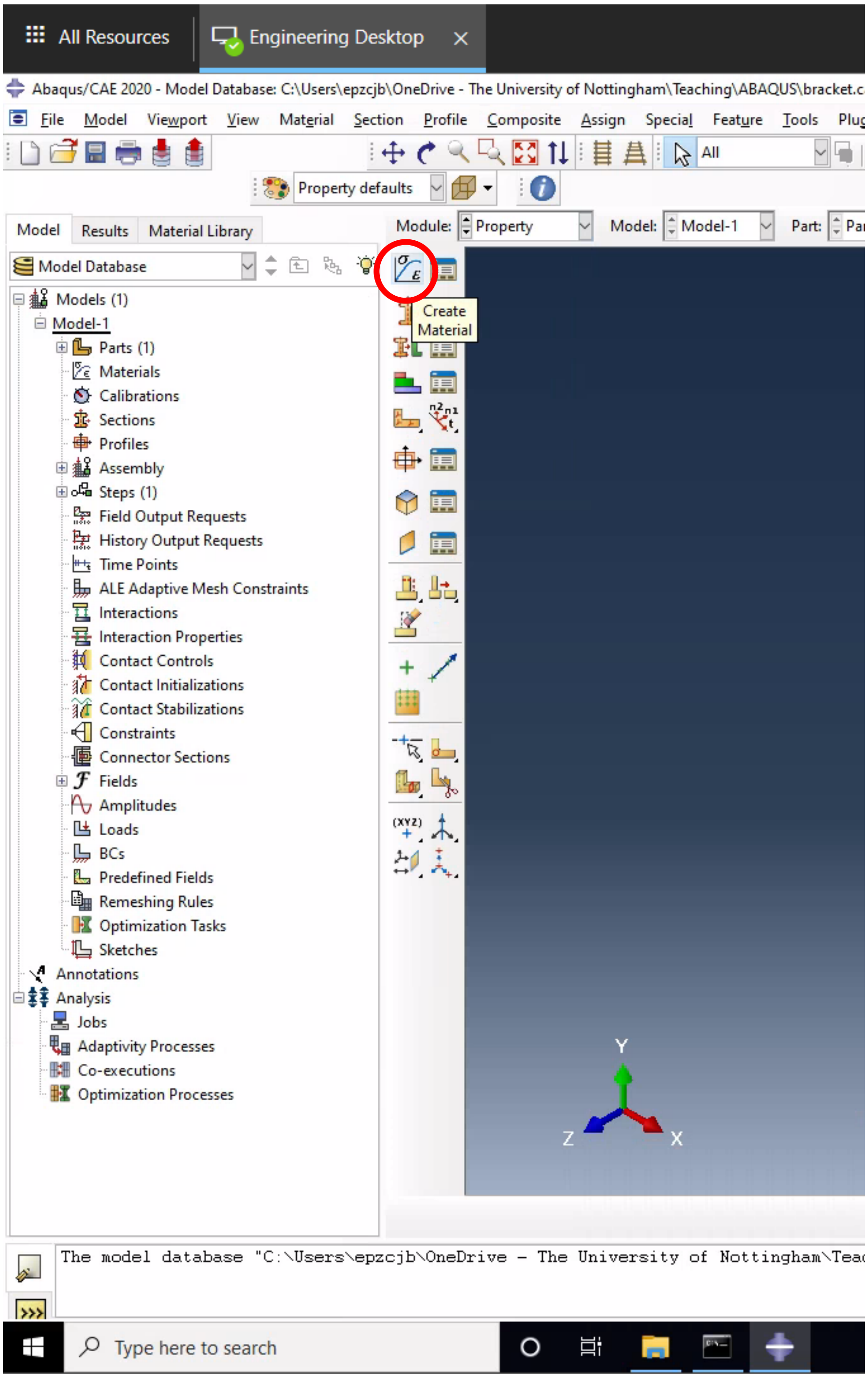


You can specify the properties of a part or part region by creating a section and assigning it to the part. When you create or edit a material, profile, or section, you must enter data in the appropriate editor. For example, when you create a material, you must enter data in the material editor.

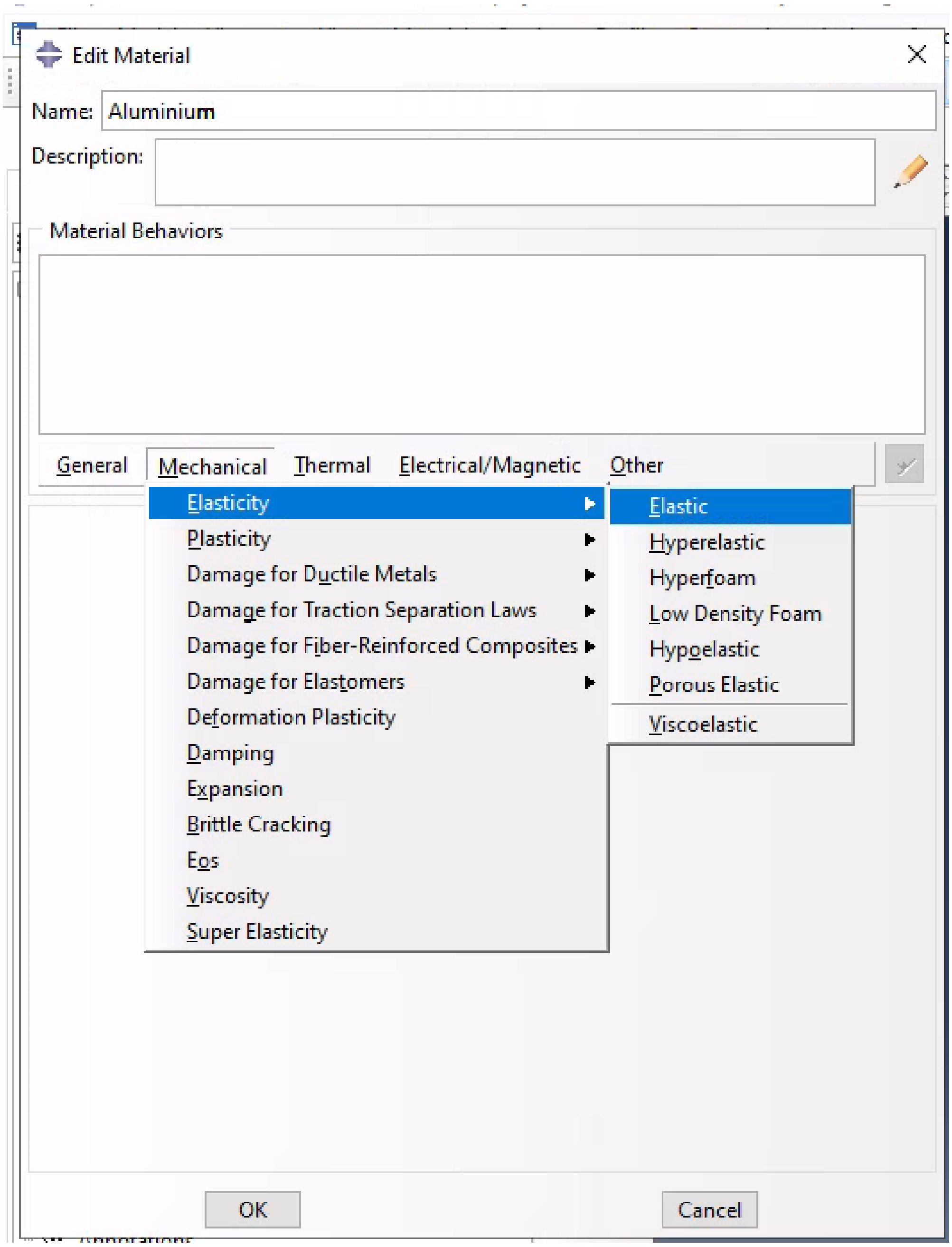
2. Select the Property module



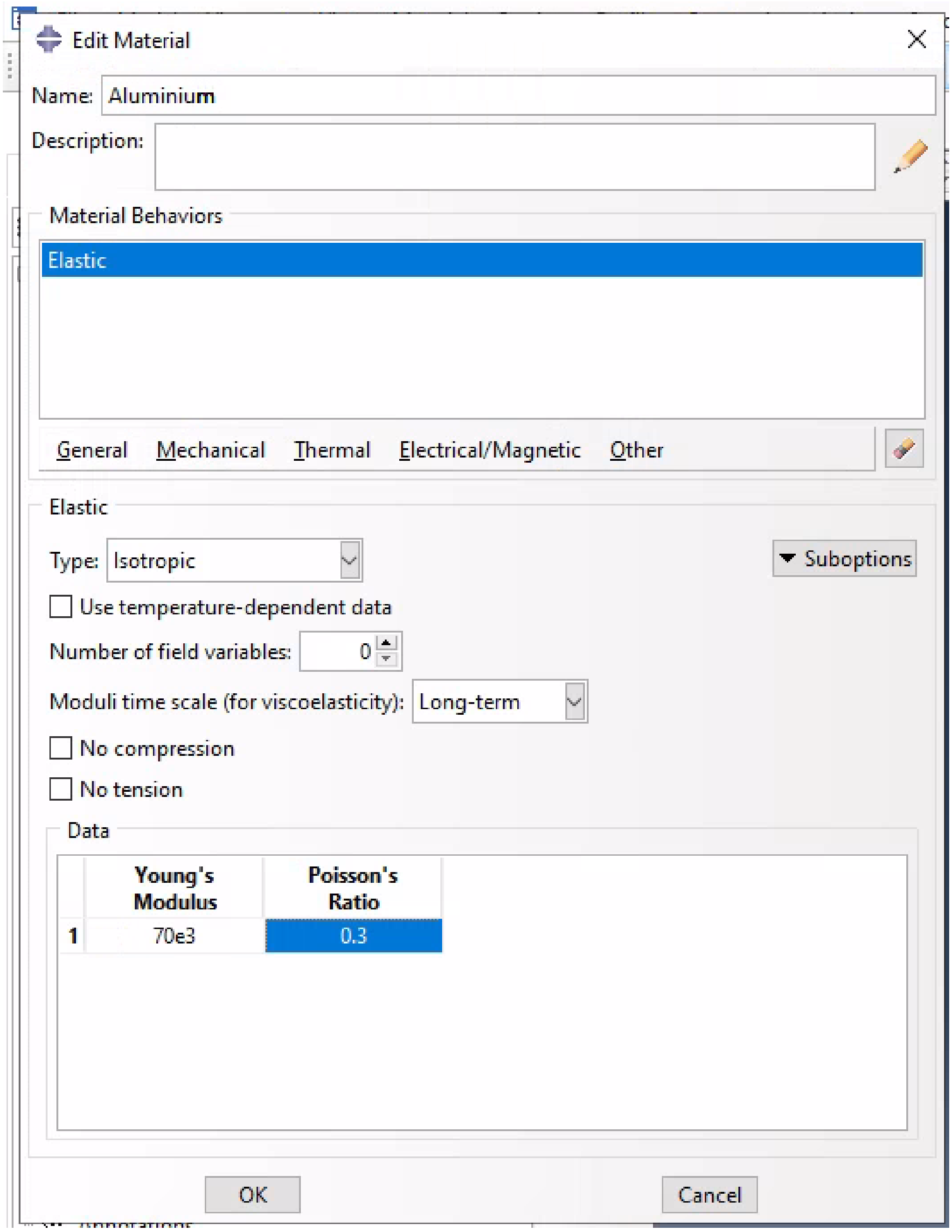
3. Select Create Material



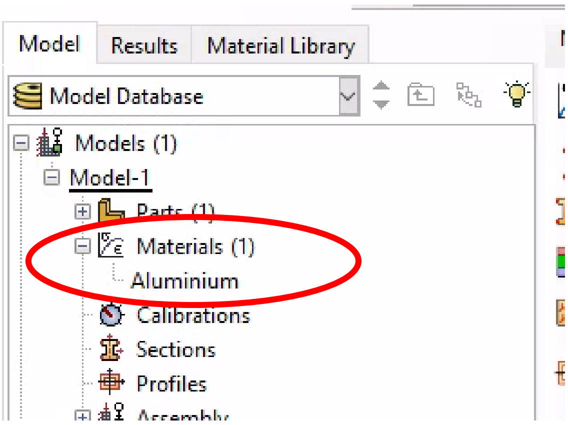
4. In the Edit Material window, call the material Aluminium and then select Mechanical -> Elasticity -> Elastic



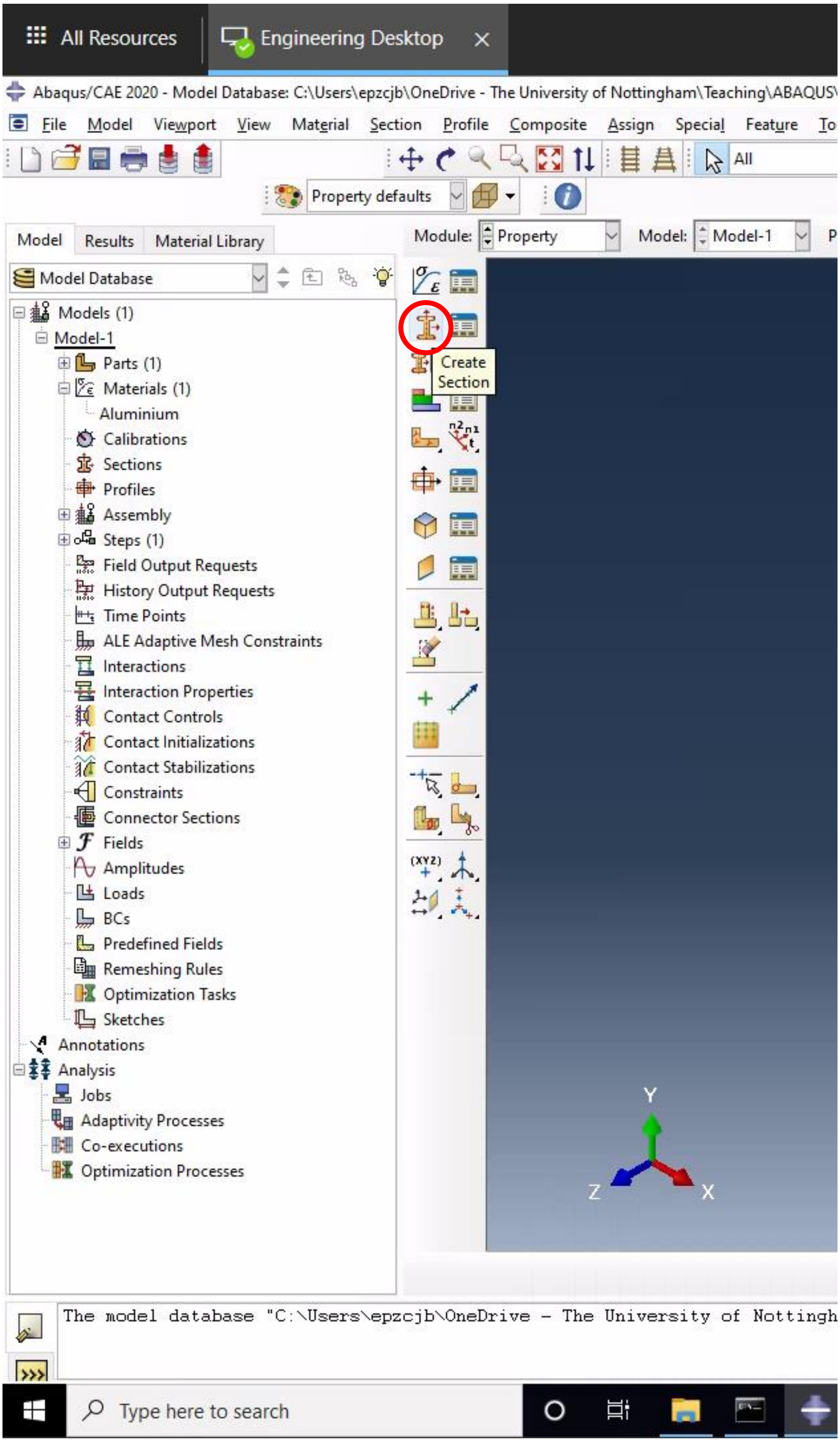
5. Enter the values shown below to define the Young’s Modulus and Poisson’s ratio, leave all other options as-is (Isotropic for e.g.) then click OK



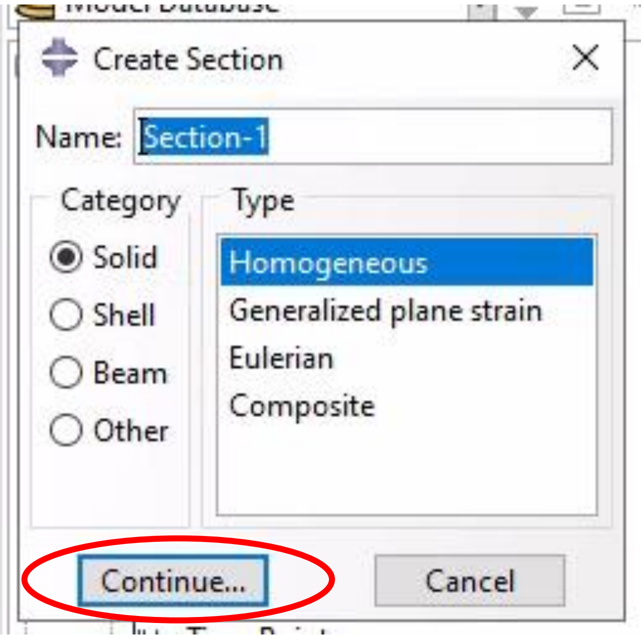
The material definition will appear under the Materials section of the Model tree:



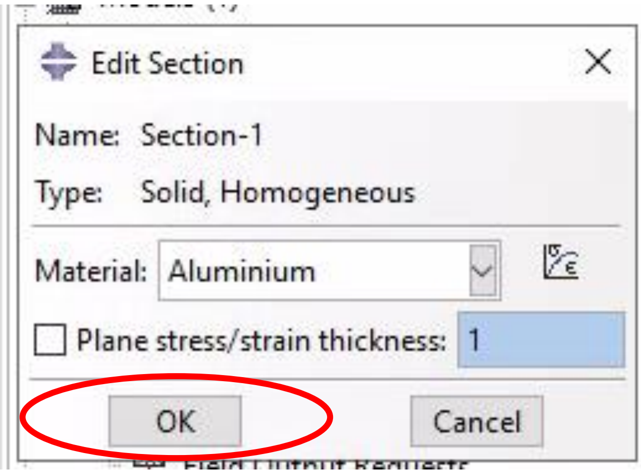
6. Next, we need to create a section and assign the material definition to it. Select Create Section



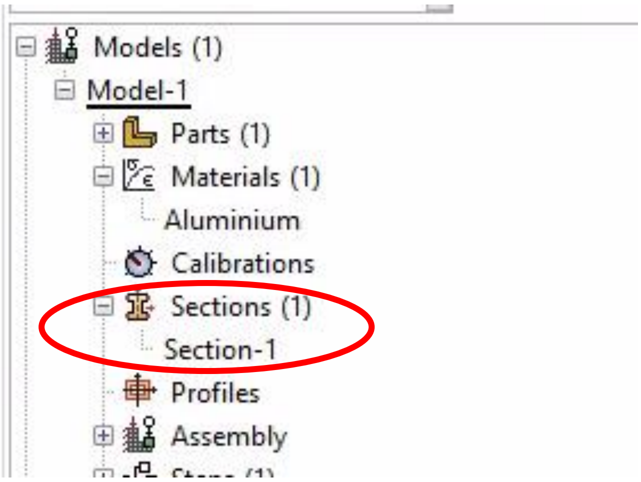
7. Leave the default options in the Create Section window (Name, Solid, Homogeneous) and click Continue…



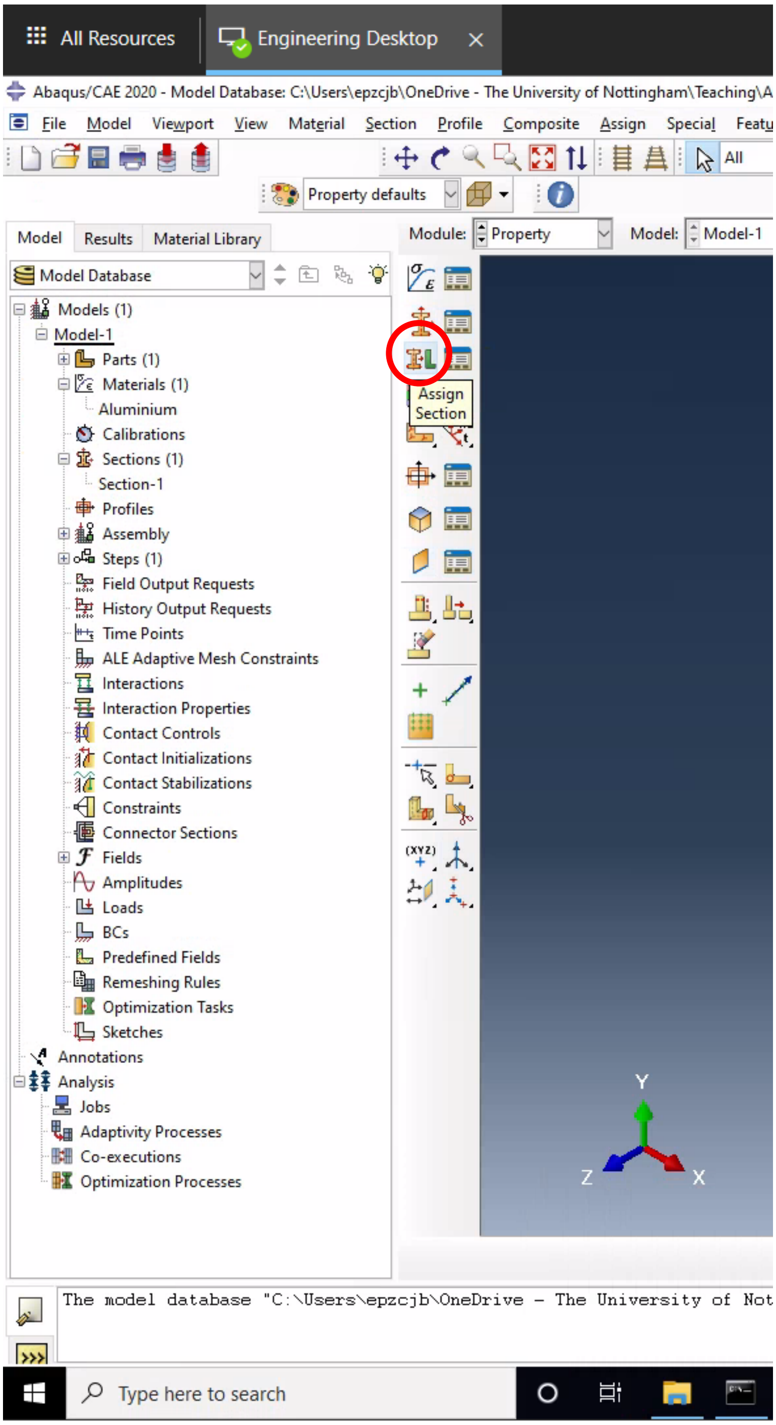
8. Next, we assign the material to the section, by default, the Aluminium material definition is chosen, leave the defaults and click OK



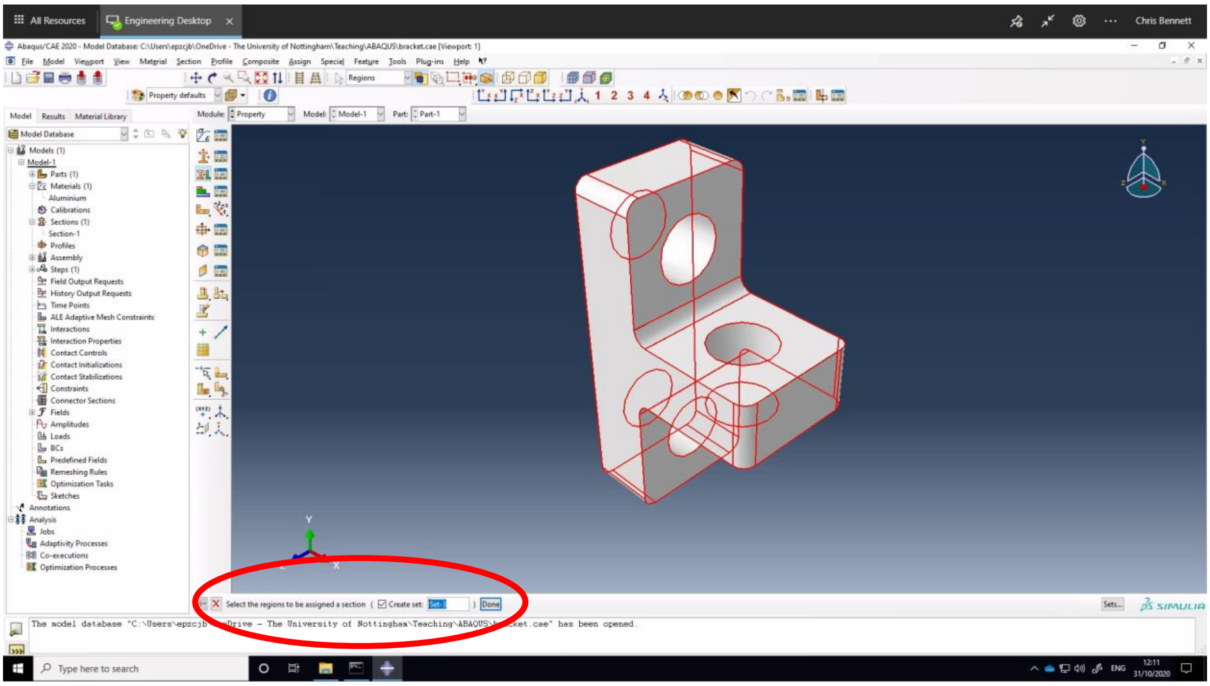
The section definition will appear under the Sections section of the Model tree:



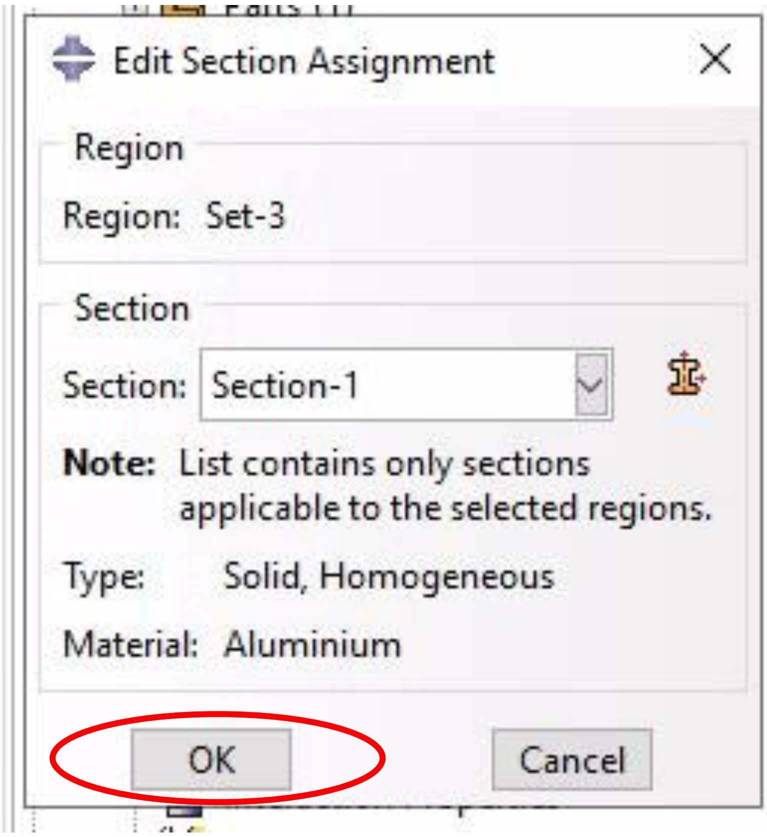
9. To assign the section to the part, click Assign Section



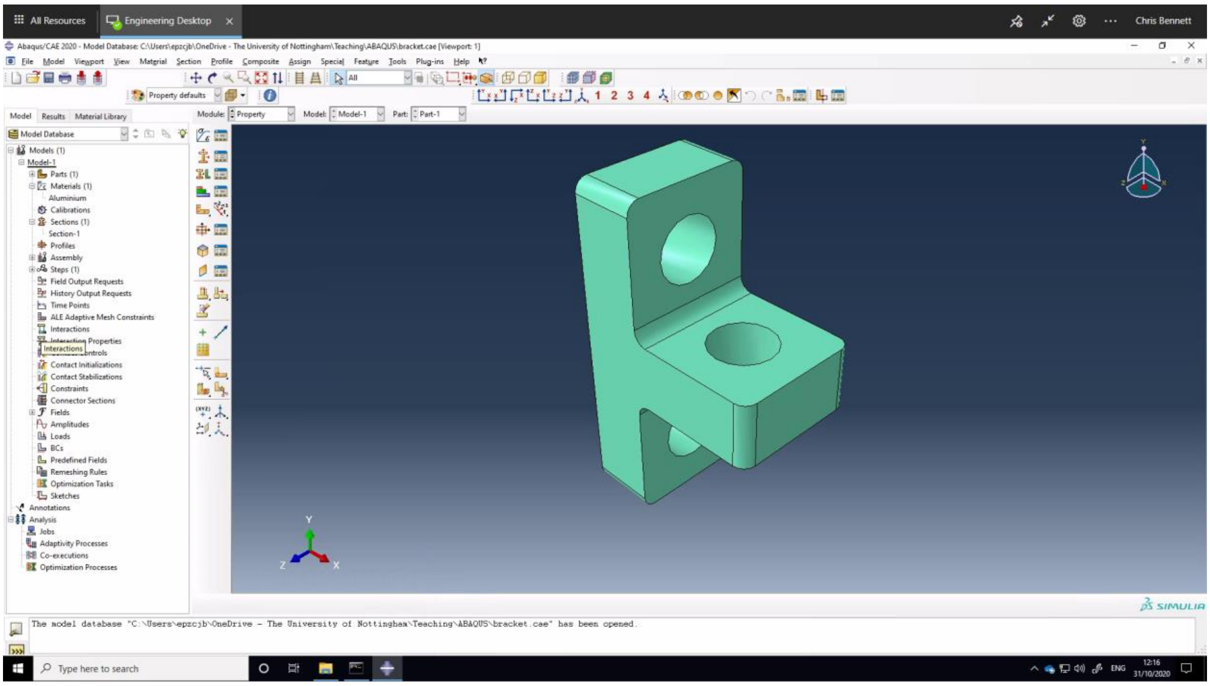
10. When prompted to select the regions to be assigned a section, click on the bracket part in the viewport. This will also create a set containing the geometry, leave the default name (Set-1). Click Done.



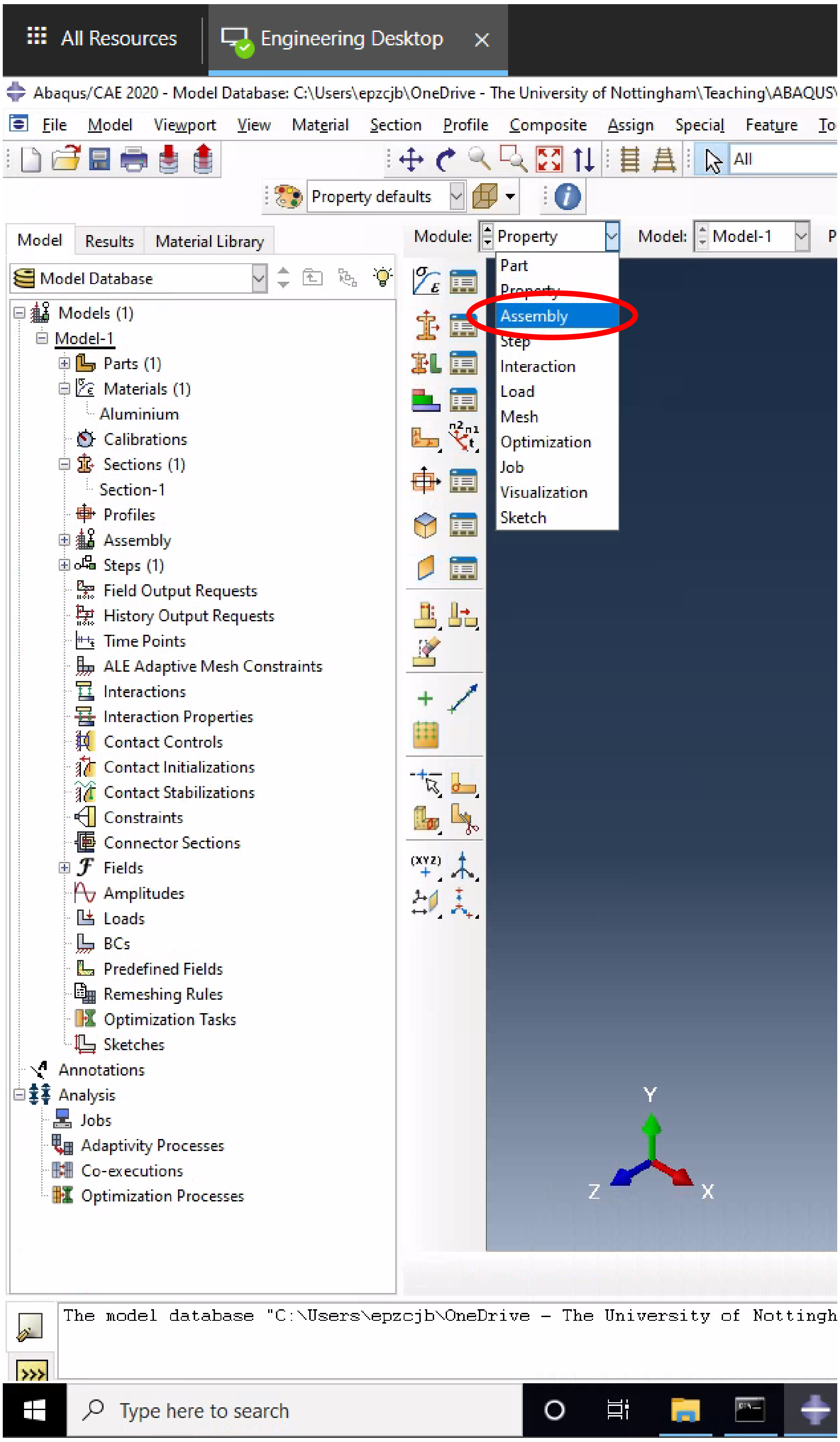
11. The Edit Section Assignment window will appear, click OK to apply the Section-1 definition previously created with the Aluminium material definition



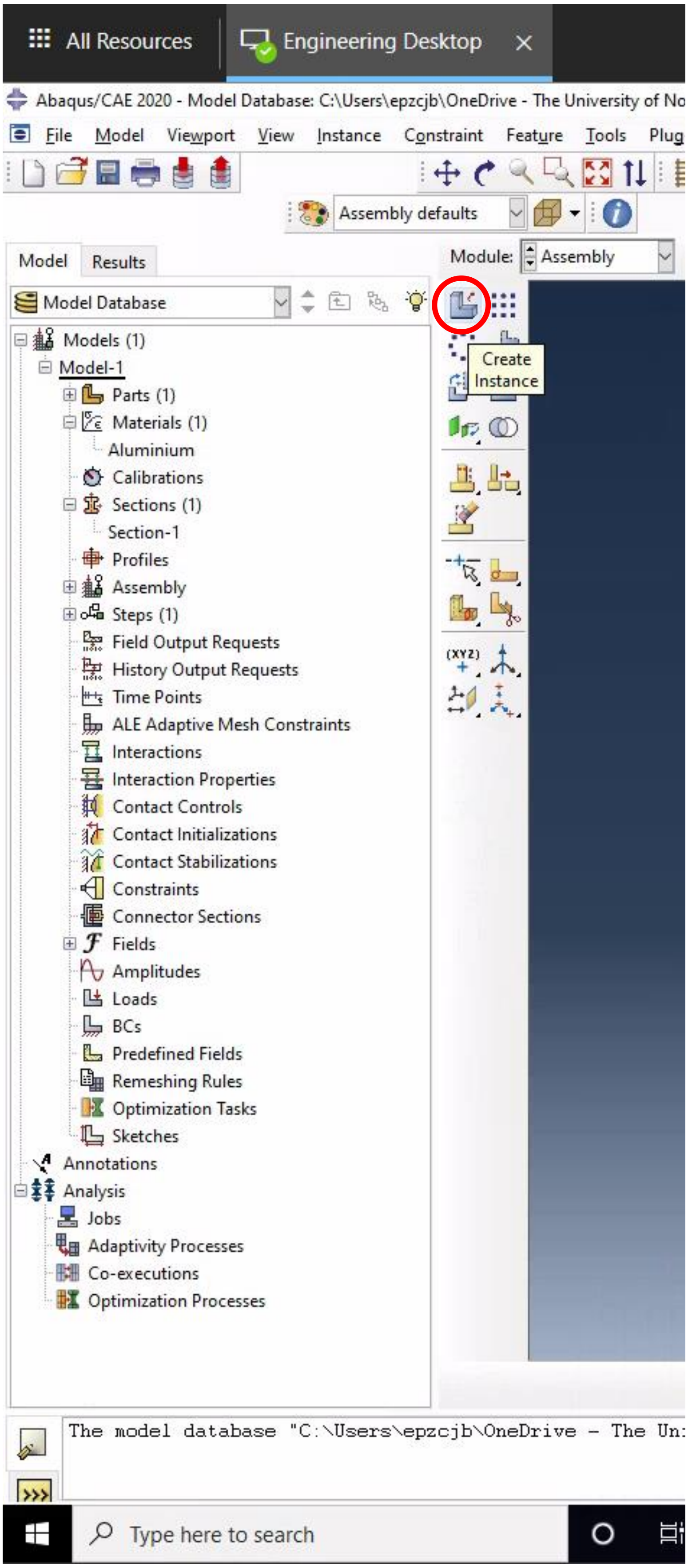
The part will turn green to show it has a section definition applied



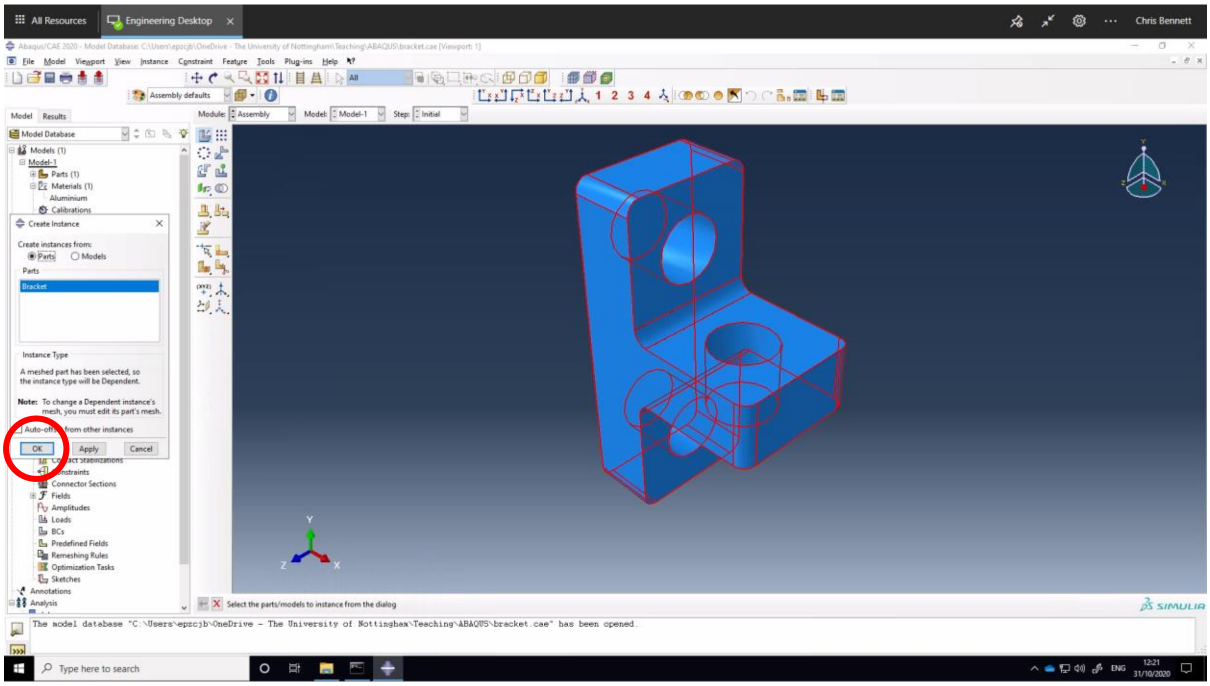
12. Select the Assembly module



13. Next, add an instance of the bracket to the assembly, select Create Instance



14. The Create Instance window will open and by default the Bracket part will be selected. Click OK to add the instance and close the window.



An ABAQUS/CAE model uses the following two types of steps:

The initial step

ABAQUS/CAE creates a special initial step at the beginning of the model's step sequence and names it Initial. ABAQUS/CAE creates only one initial step for your model, and it cannot be renamed, edited, replaced, copied, or deleted.

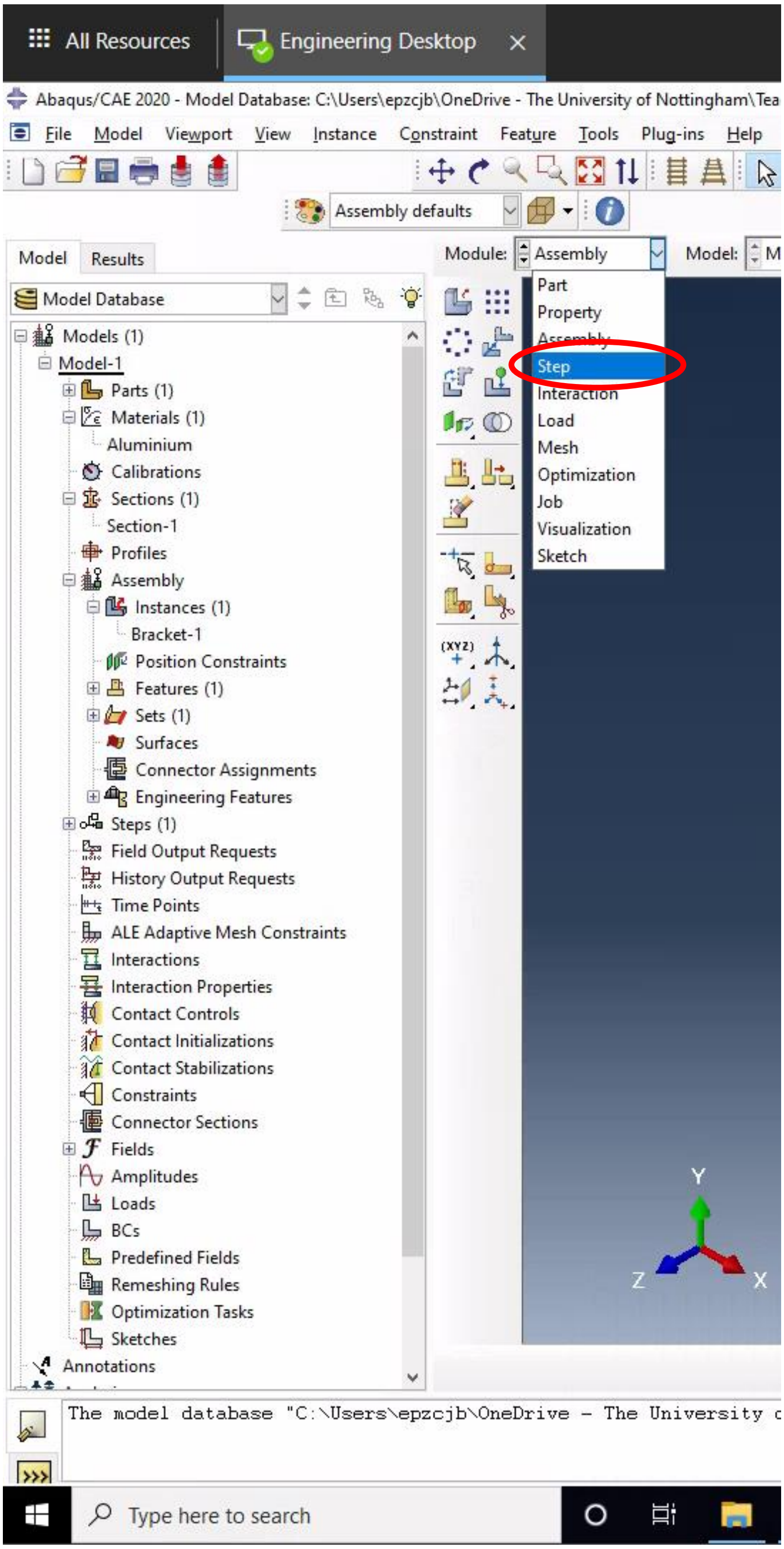
The initial step allows you to define boundary conditions, predefined fields, and interactions that are applicable at the very beginning of the analysis. For example, if a boundary condition or interaction is applied throughout the analysis, it is usually convenient to apply such conditions in the initial step. Likewise, when the first analysis step is a linear perturbation step, conditions applied in the initial step form part of the base state for the perturbation.

Analysis steps

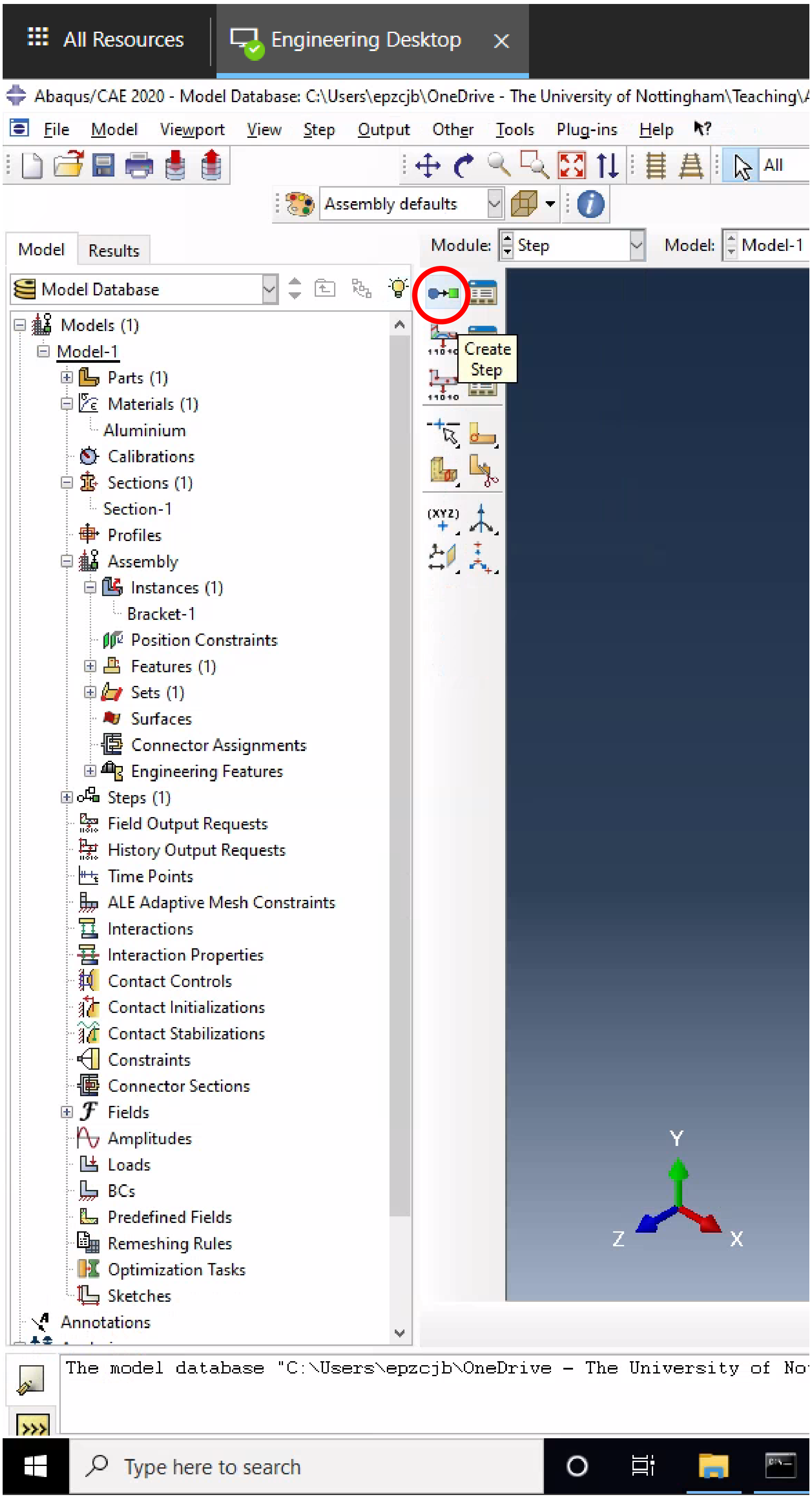
The initial step is followed by one or more analysis steps. Each analysis step is associated with a specific procedure that defines the type of analysis to be performed during the step, such as a static stress analysis or a transient heat transfer analysis. You can change the analysis procedure from step to step in any meaningful way, so you have great flexibility in performing analyses. Since the state of the model (stresses, strains, temperatures, etc.) is updated throughout all general analysis steps, the effects of previous history are always included in the response for each new analysis step.

There is no limit to the number of analysis steps you can define, but there are restrictions on the step sequence.

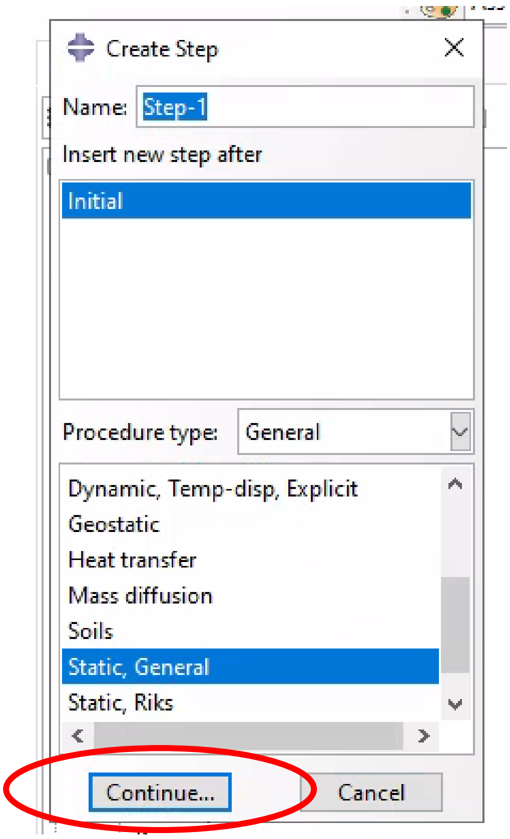
15. Select the Step module



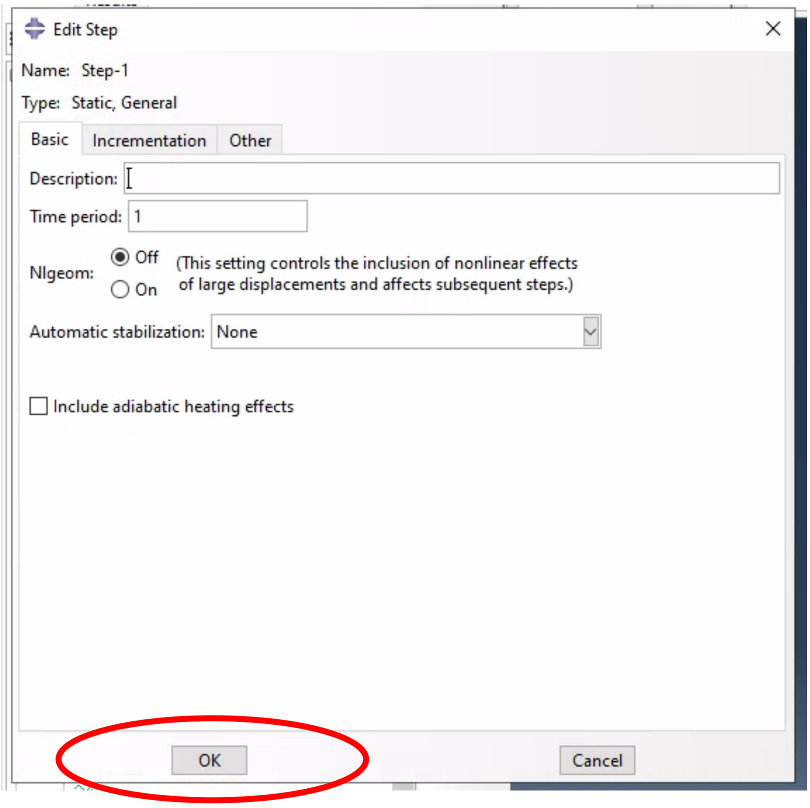
16. Click Create Step



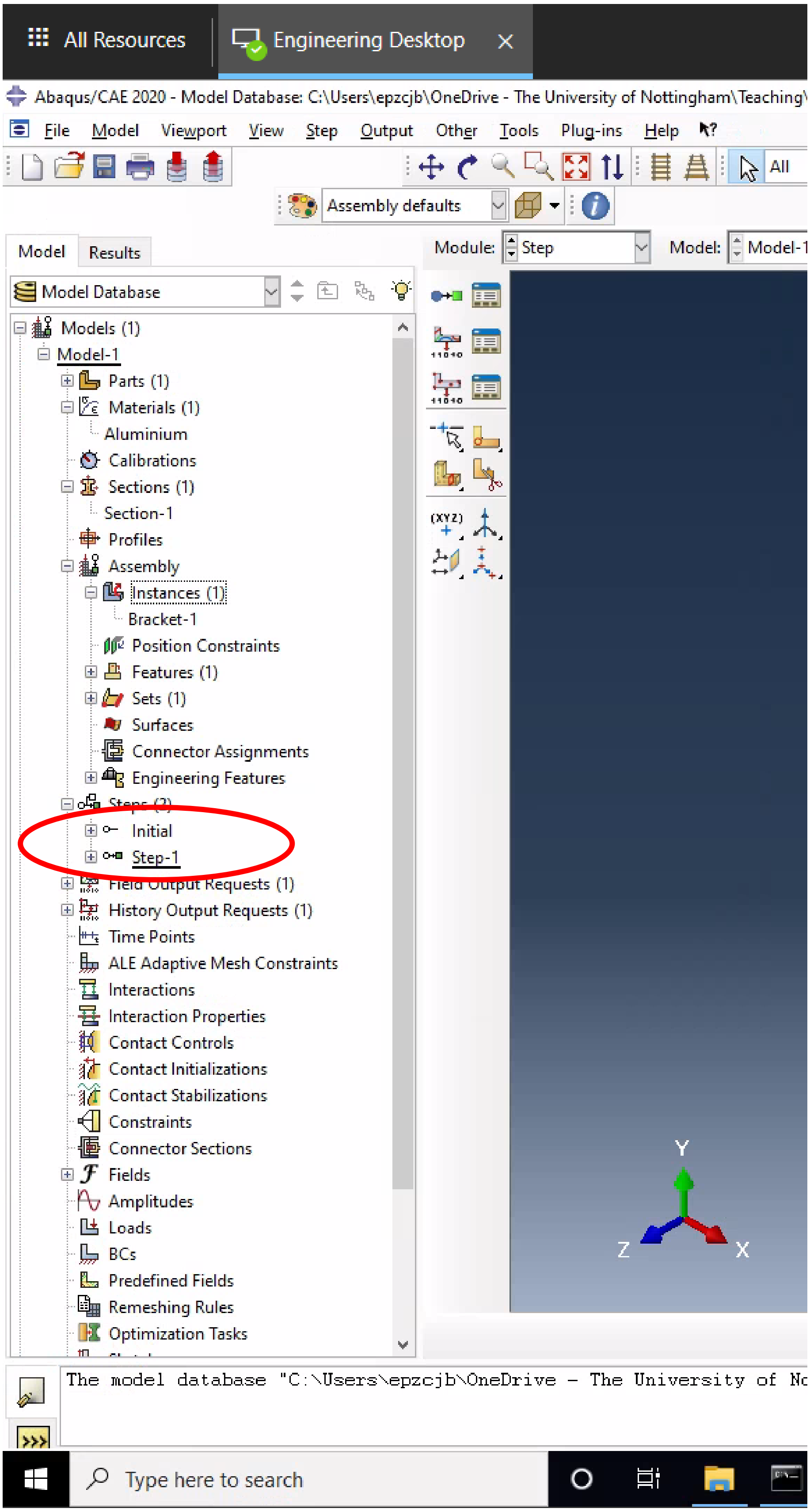
17. The Create Step window will appear where the Name of the step, position of the step and step type can be defined. Leave the defaults and create a Static, General step called Step-1, it will appear after the Initial step. Click Continue…



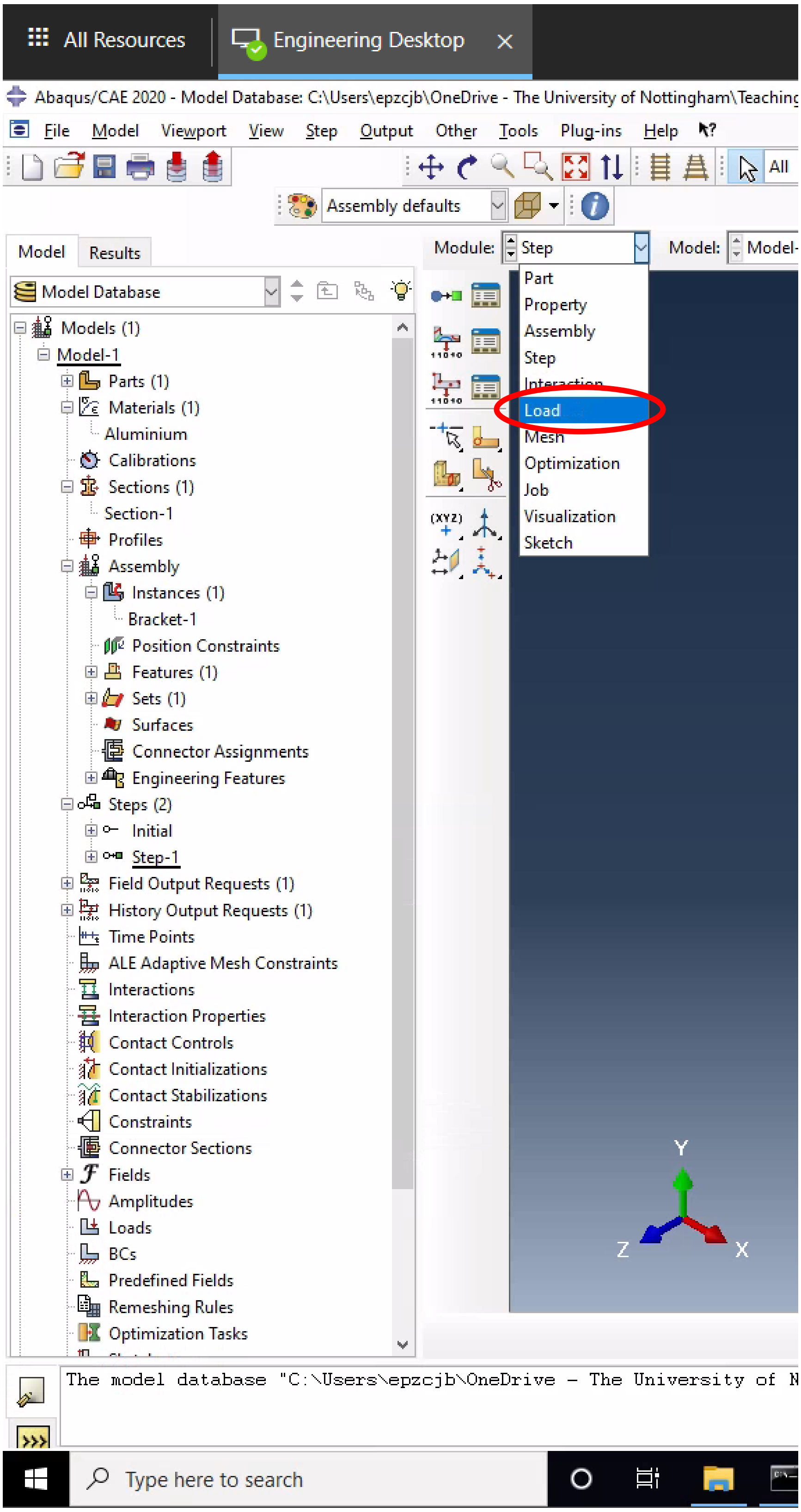
18. The Edit Step window will appear, where further options regarding the step can be defined, leave the default options and click OK



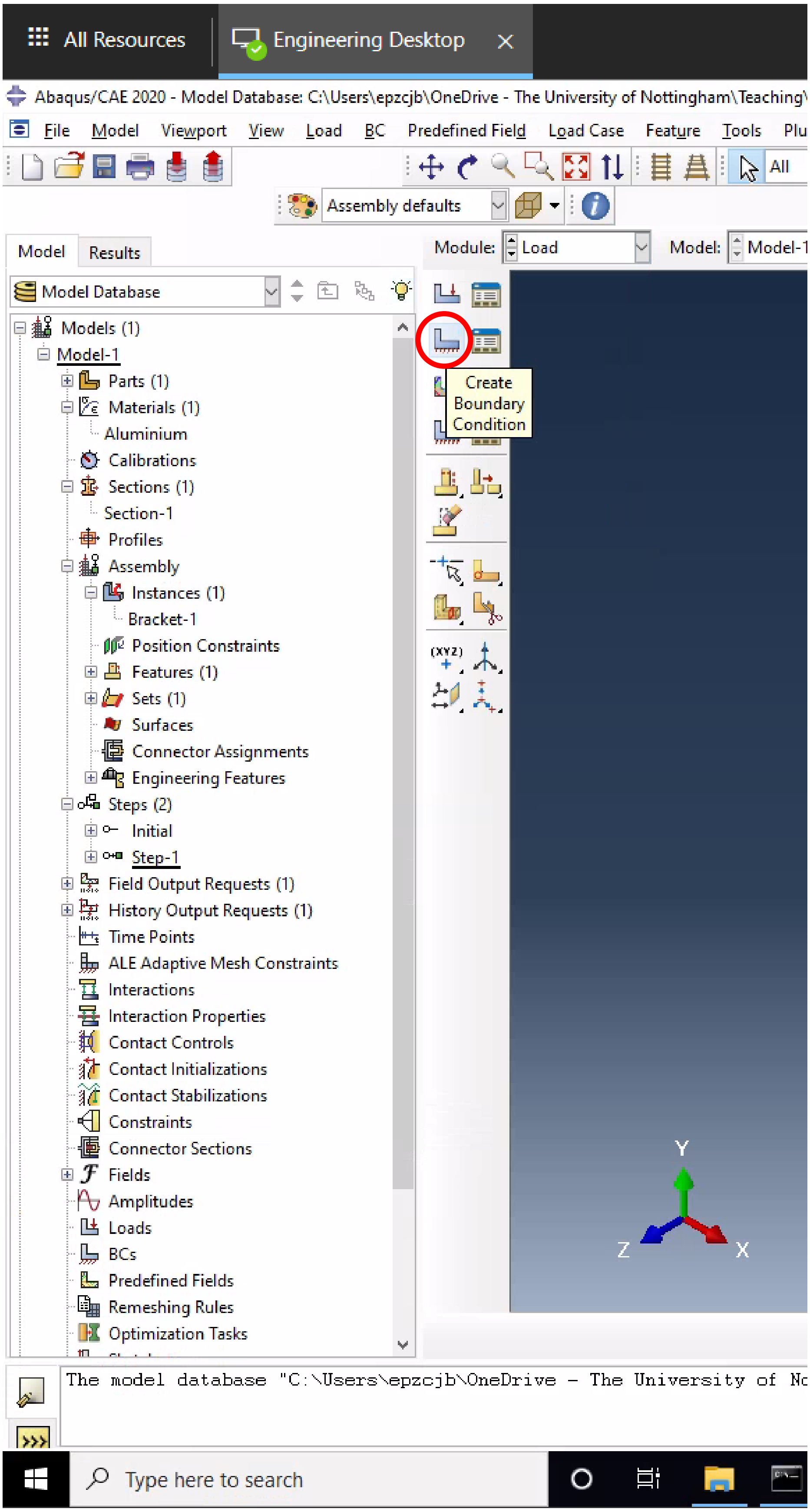
Step-1 will appear under the Steps section of the model tree



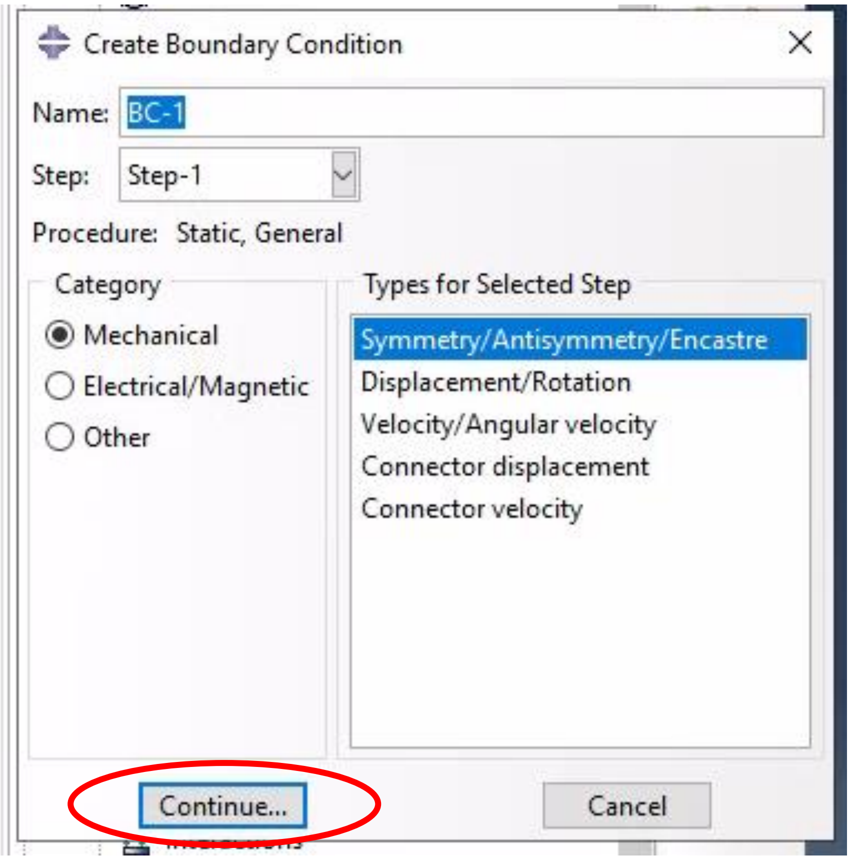
19. Select the Load module, this will be used to apply the boundary conditions and loading to the part.



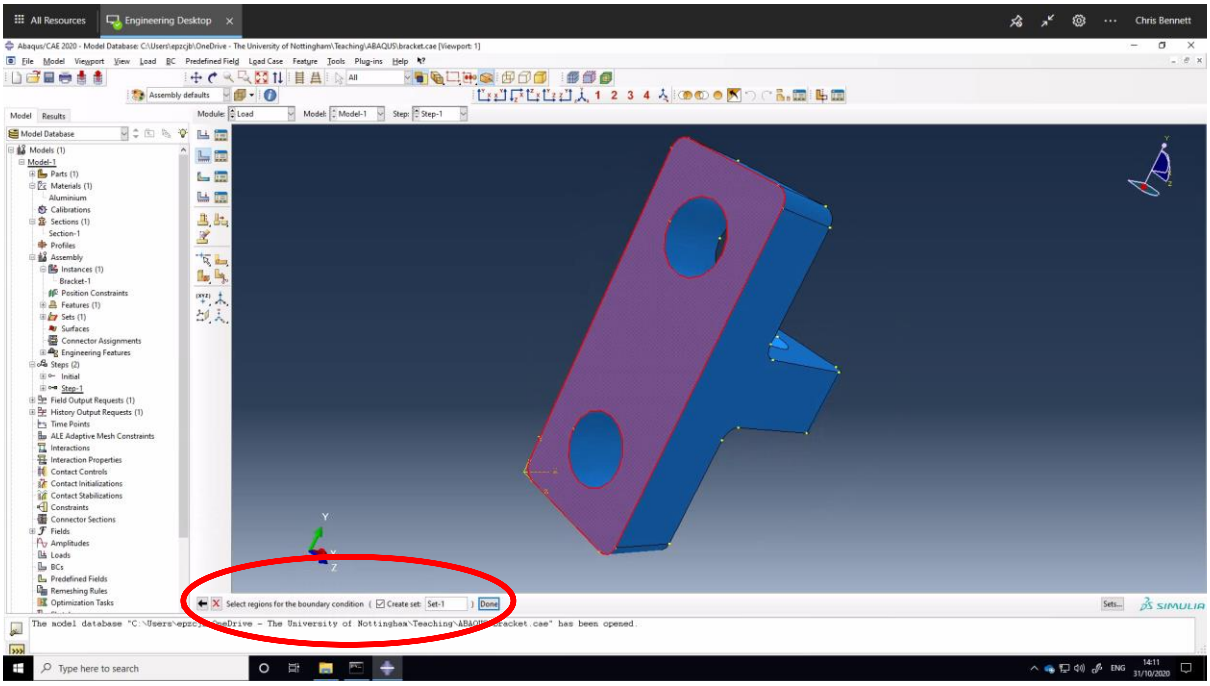
20. Click Create Boundary Condition



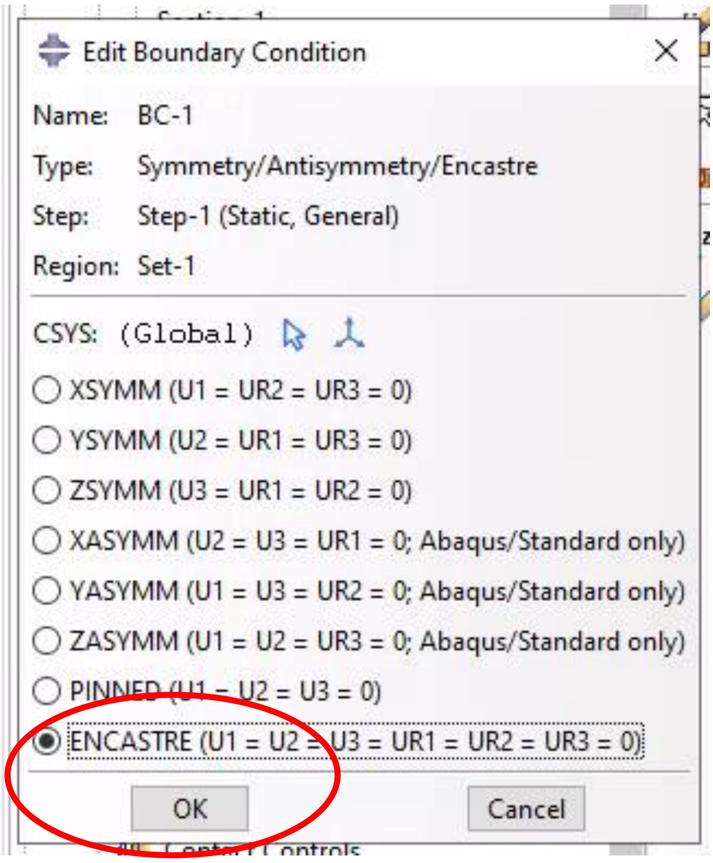
21. To create a fixed boundary condition (Encastre) on the rear face of the bracket, keep the default options in the Create Boundary Conditions window. Click Continue…



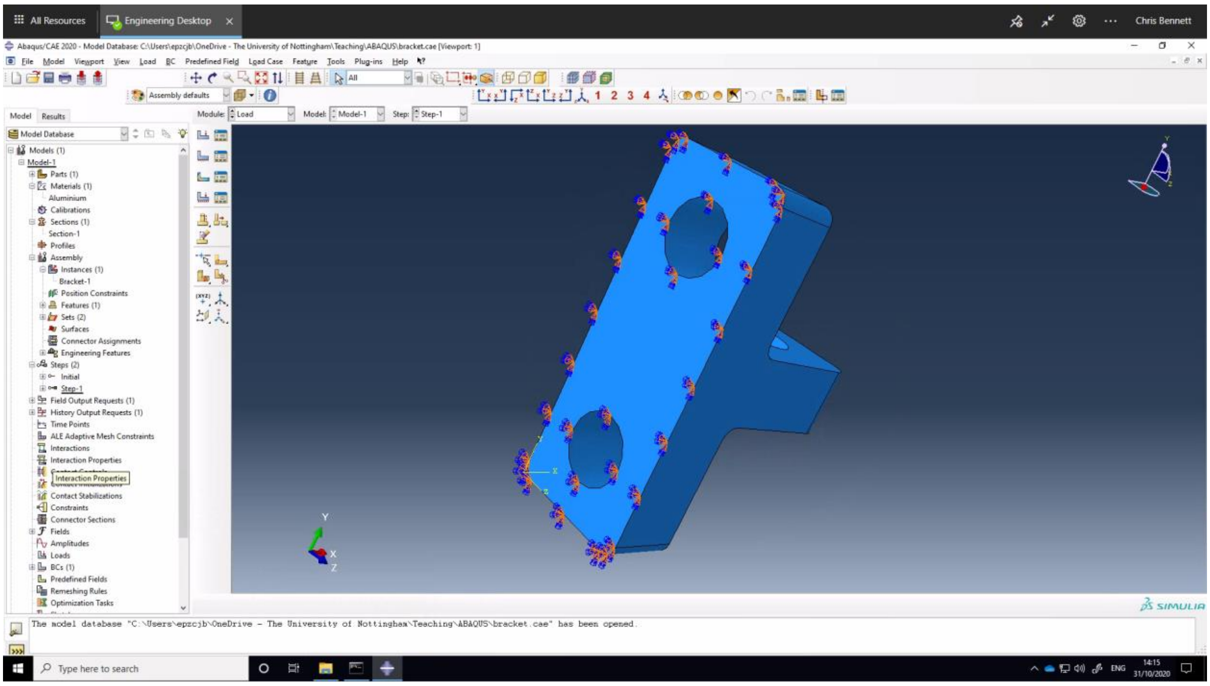
22. Rotate the bracket in the viewport and select the rear face, click Done



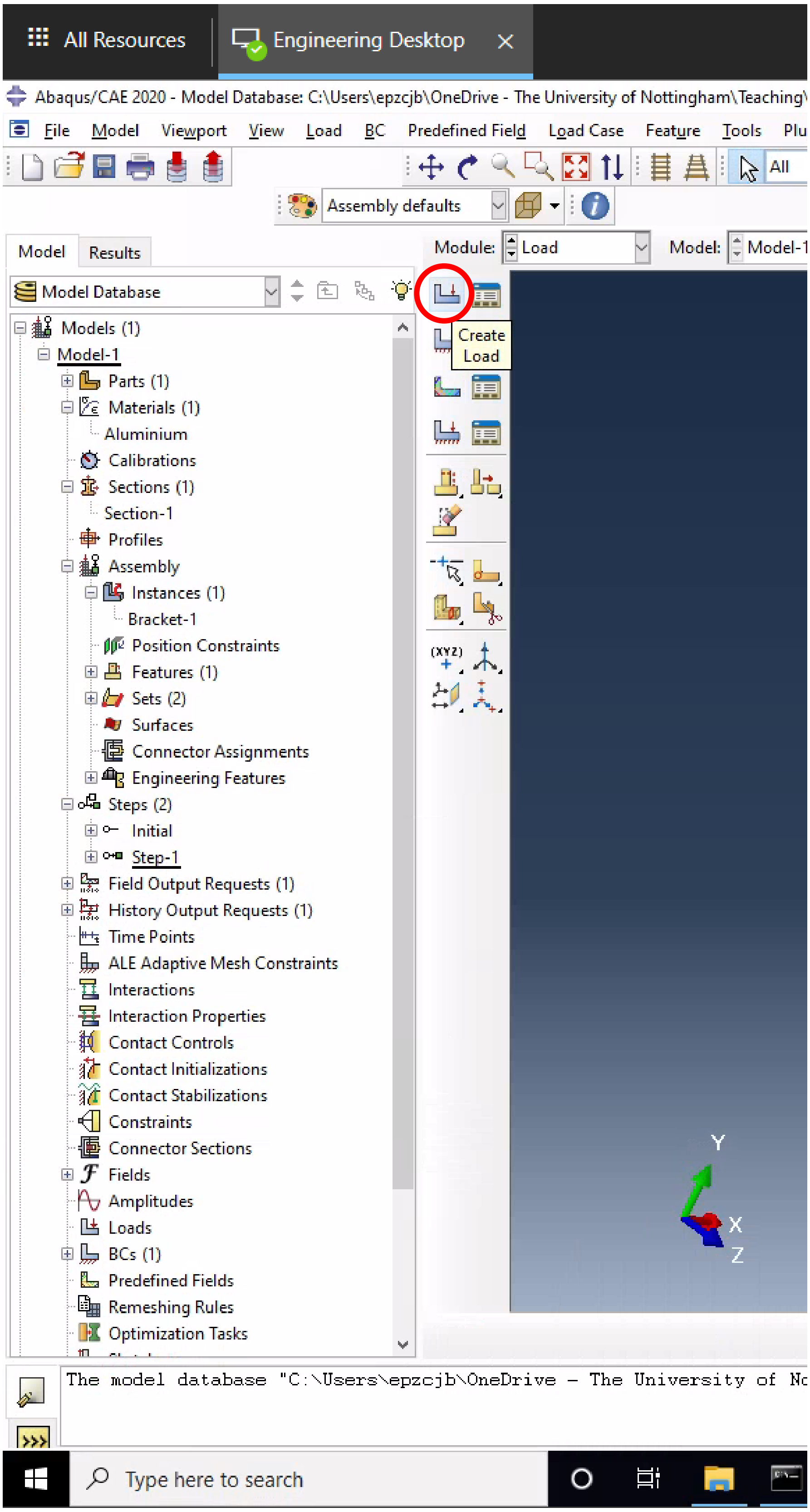
23. Select Encastre (this constrains all six degrees of freedom) in the Edit Boundary Condition window and click OK



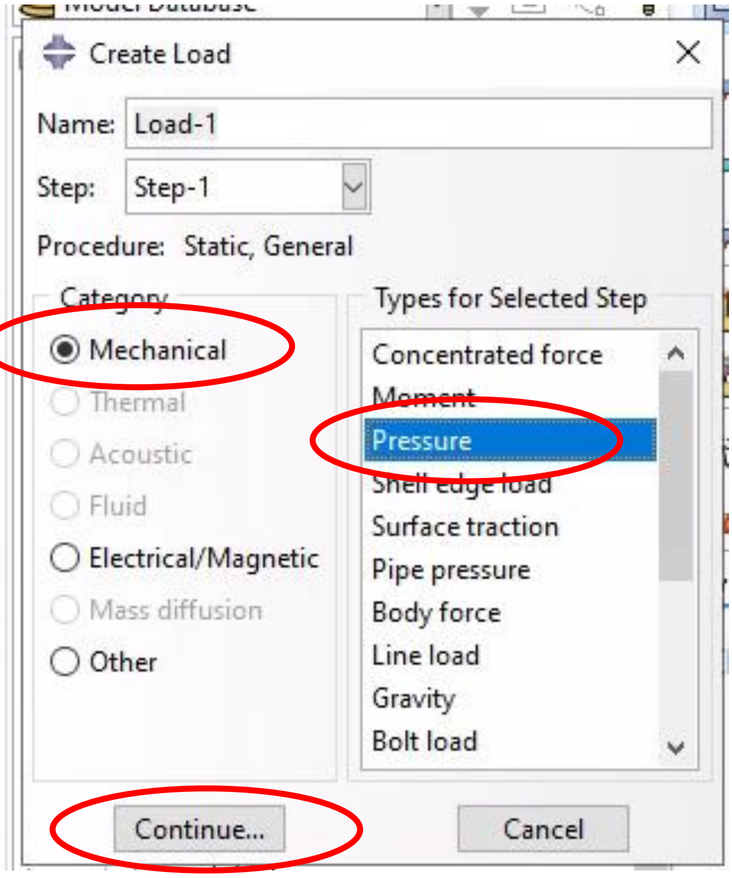
The constraints will be shown on the model



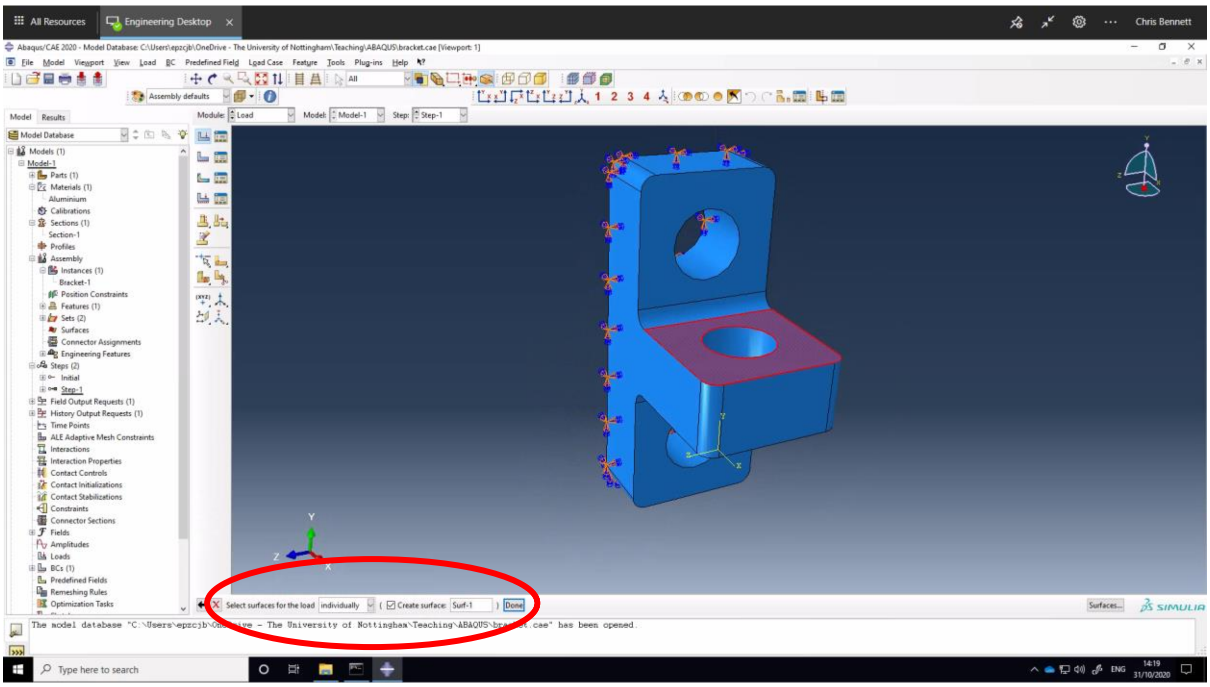
24. Click Create Load



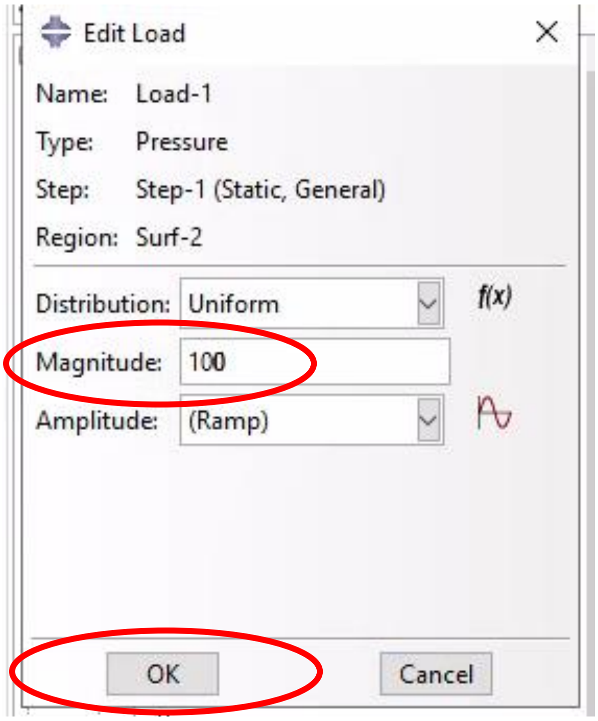
25. In the Create Load window, chose Mechanical, Pressure, click Continue…



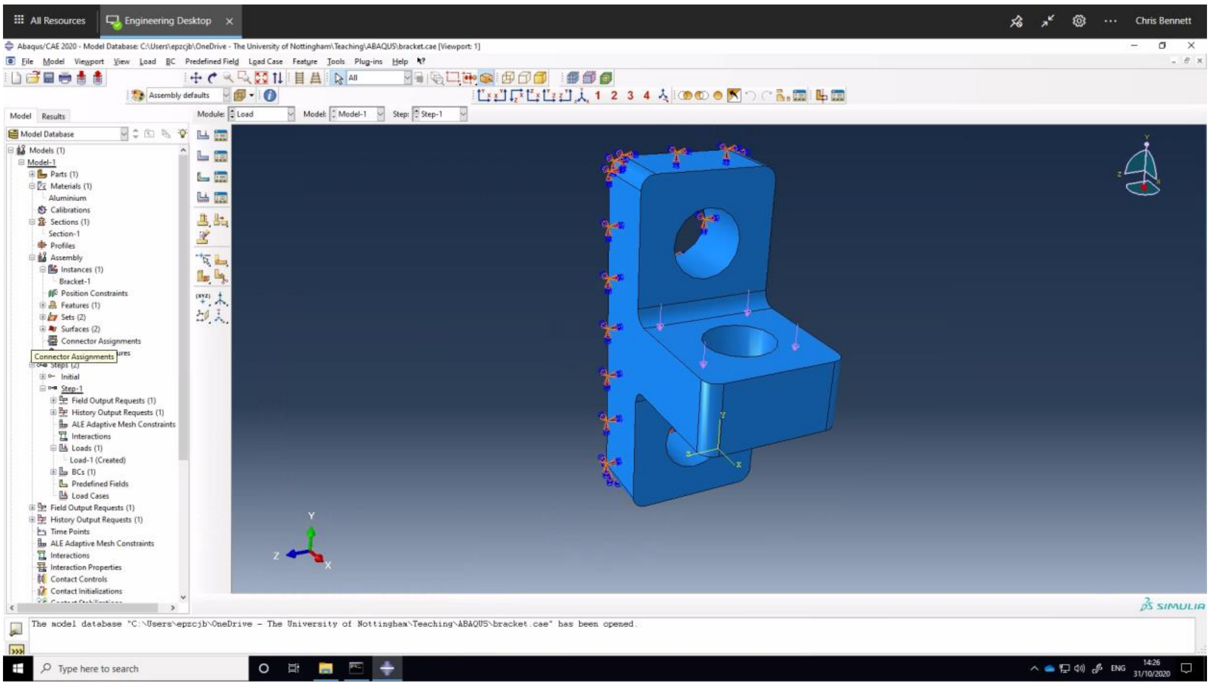
26. Rotate the view and select the top face of the protrusion, as shown, click Done



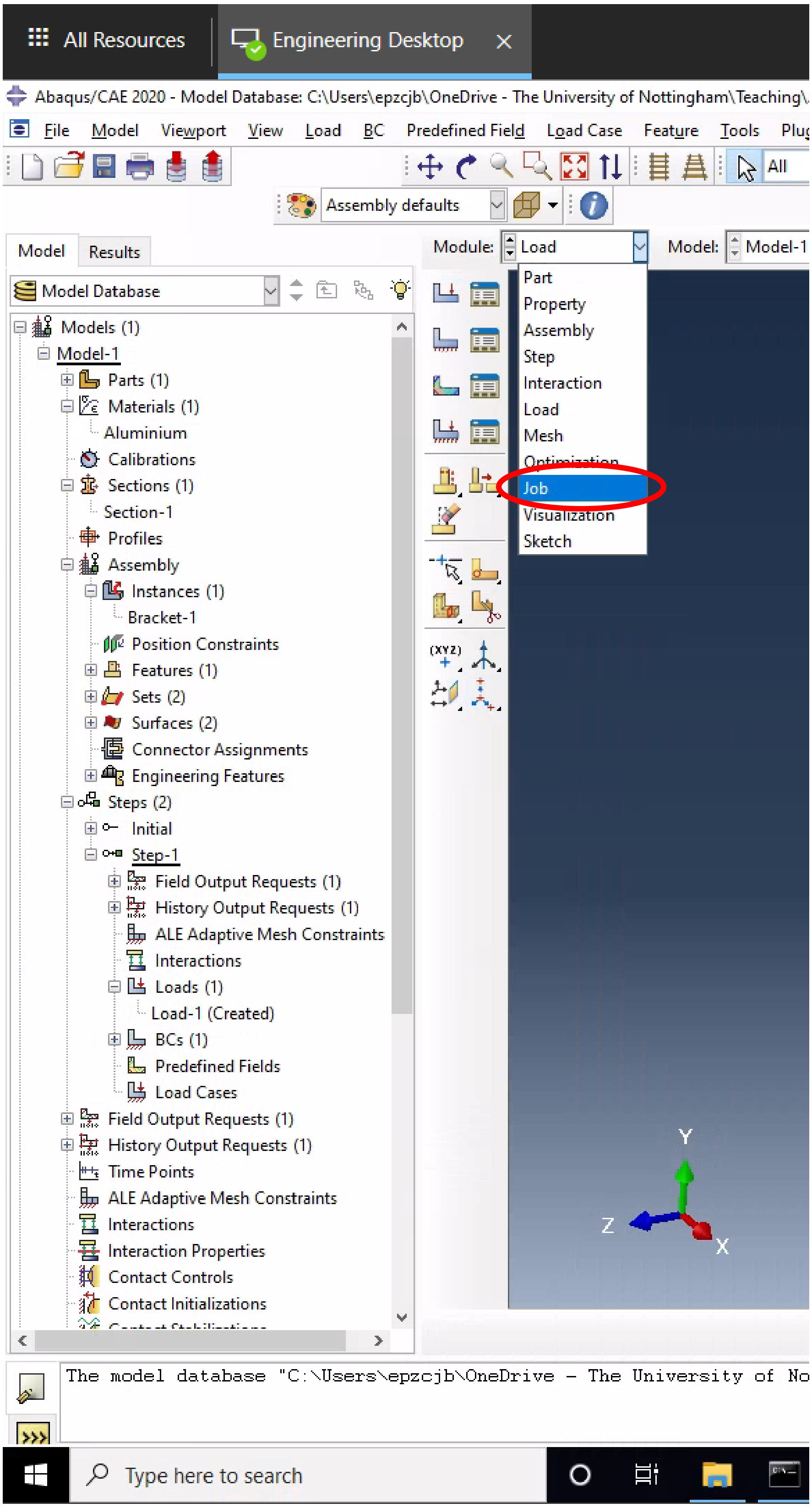
27. In the Edit Load window, specify the Magnitude as 100 (this value is MPa as the geometry is in mm and the Properties in MPa) and click OK



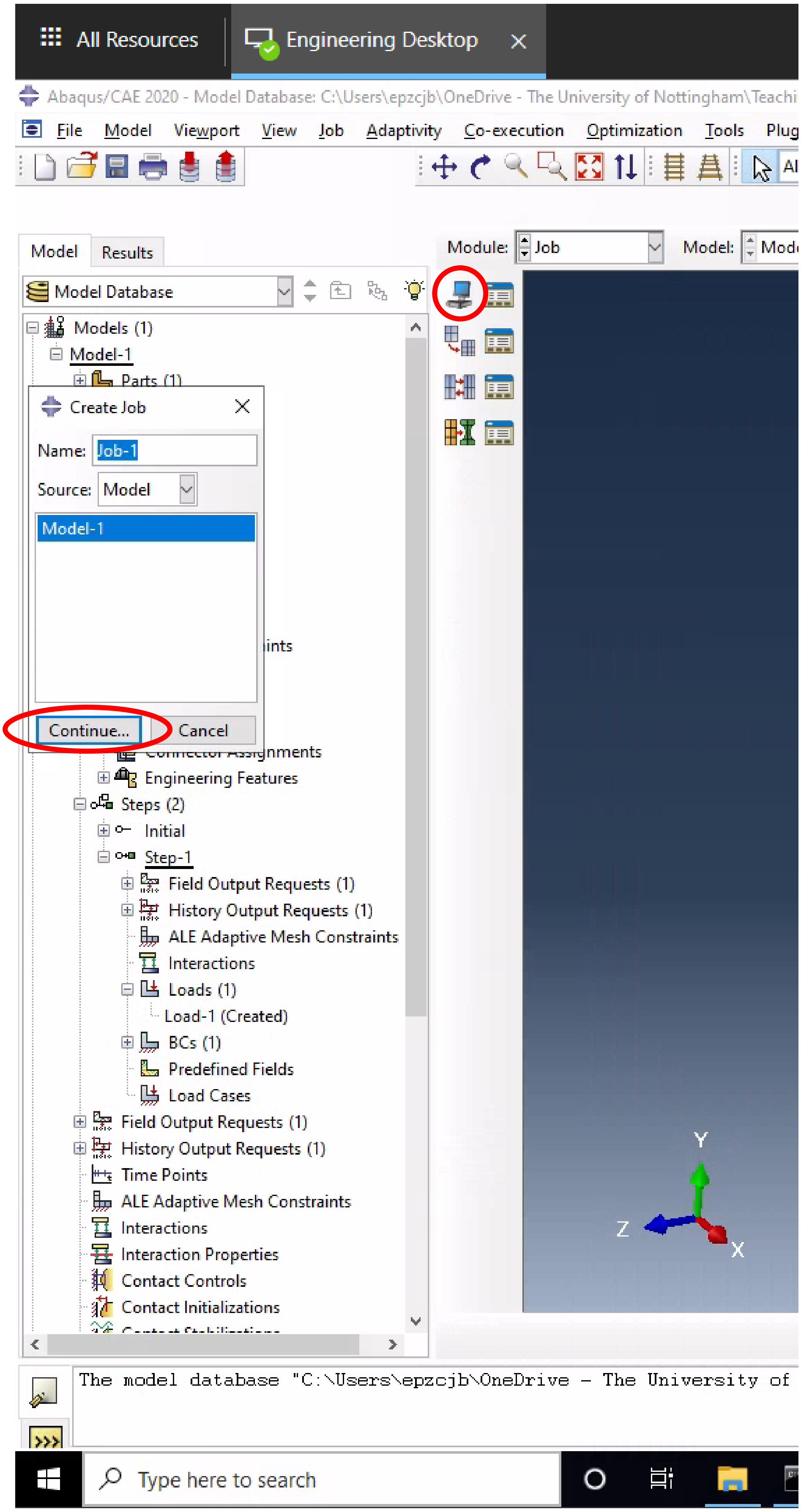
The details of the load will be added (as arrows) to the part in the viewport



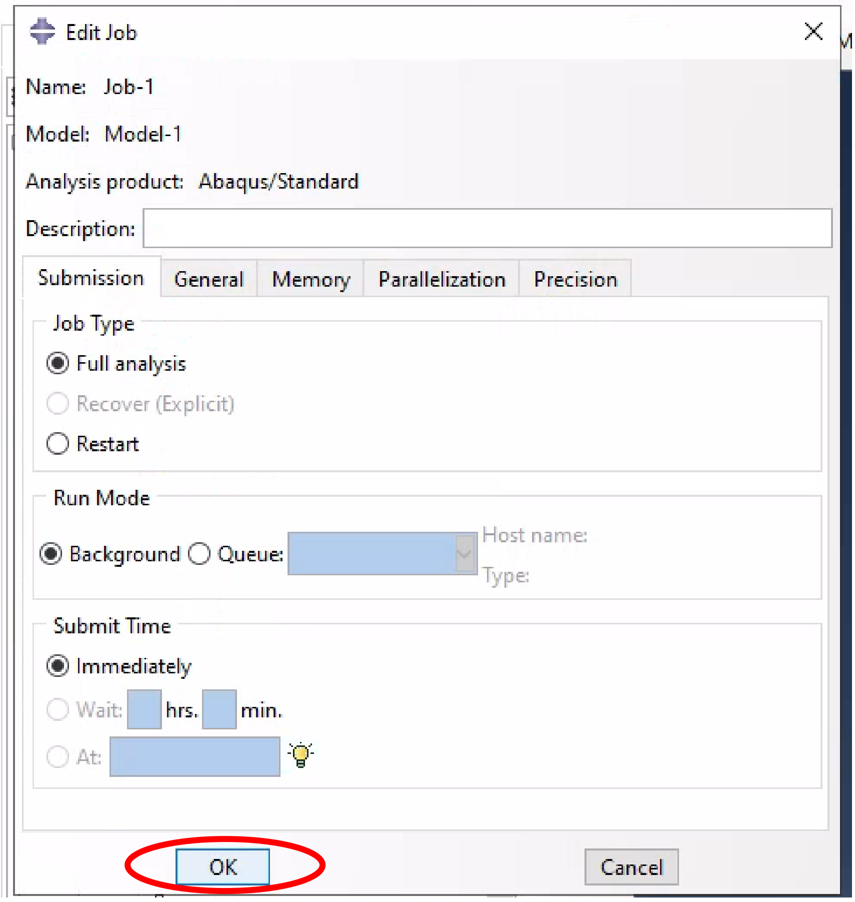
28. Select the Job module



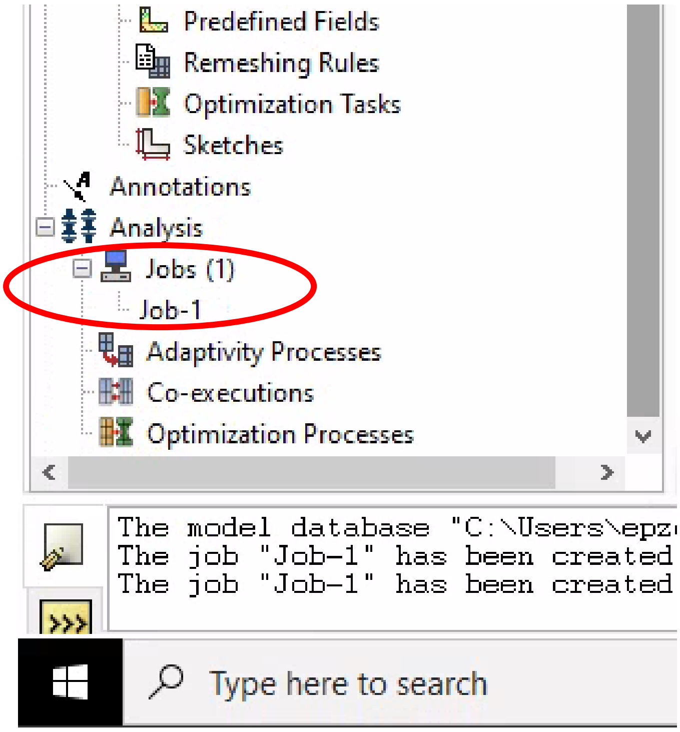
29. Click Create Job and click Continue… in the Create Job window



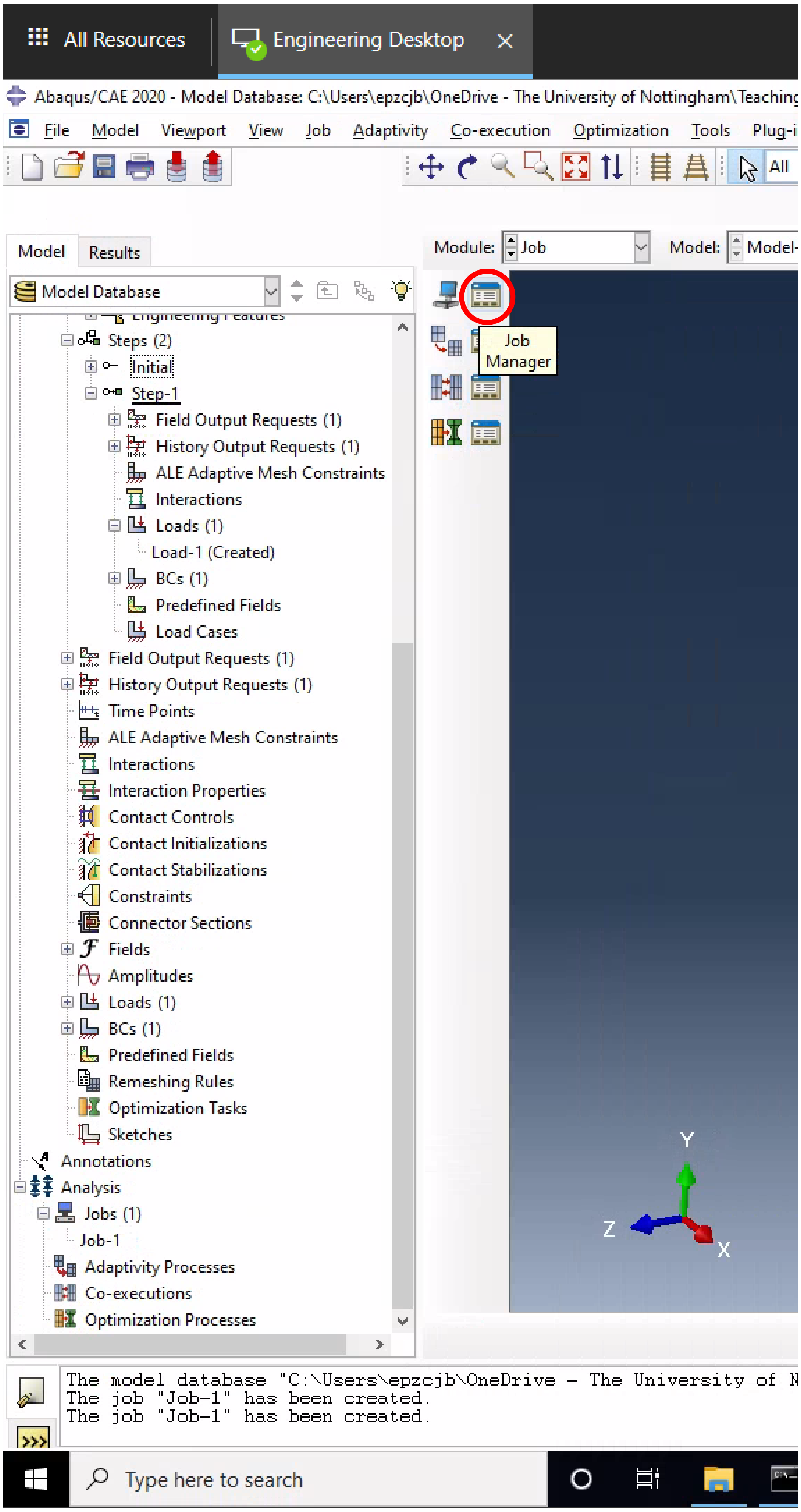
30. Click OK to select the default options in the Edit Job window



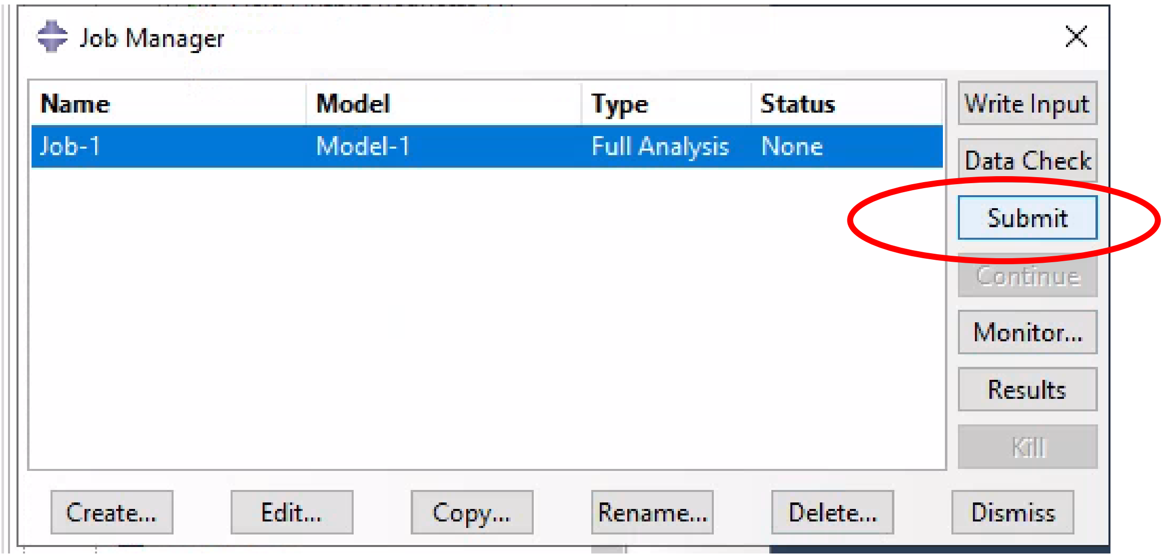
Job-1 will appear under the Jobs section of the Analysis Tree



31. Open the Job Manager



32. In the Job Manager window, select Job-1 and click Submit



You may receive a warning that Job files for Job-1 already exist, someone may have used the default name before, click OK to overwrite – this shows the importance of naming if you're running important analyses!

The following messages should appear in the message window:

The job input file "Job-1.inp" has been submitted for analysis.

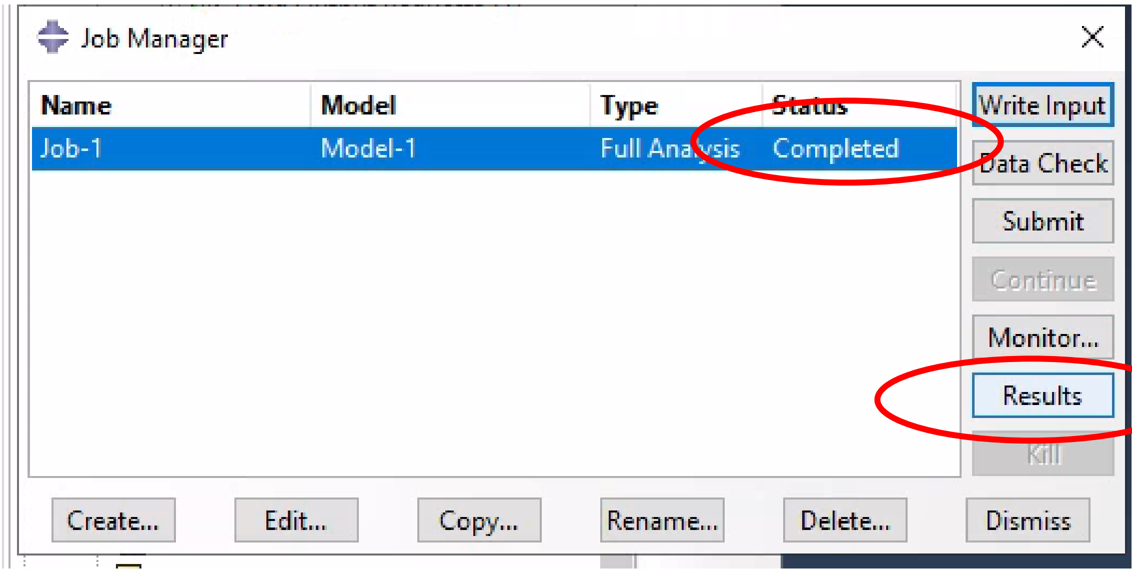
Job Job-1: Analysis Input File Processor completed successfully.

Job Job-1: Abaqus/Standard completed successfully.

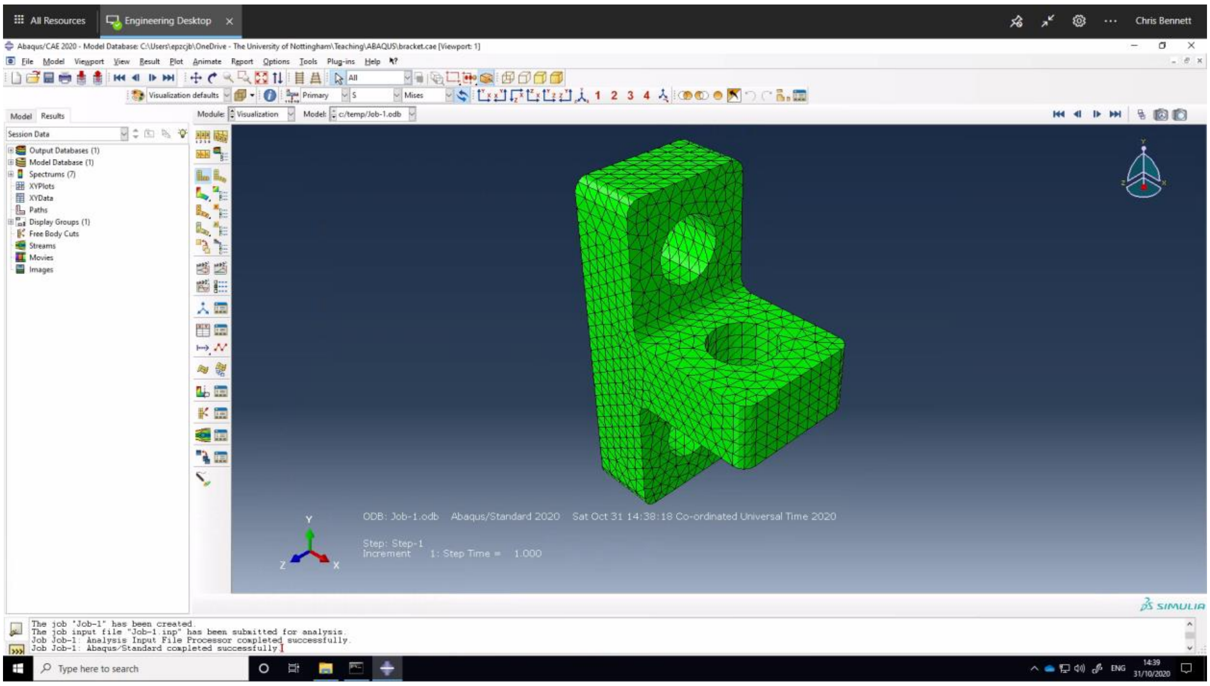
Job Job-1 completed successfully.

The Job should take around 30s to run through the Virtual Desktop

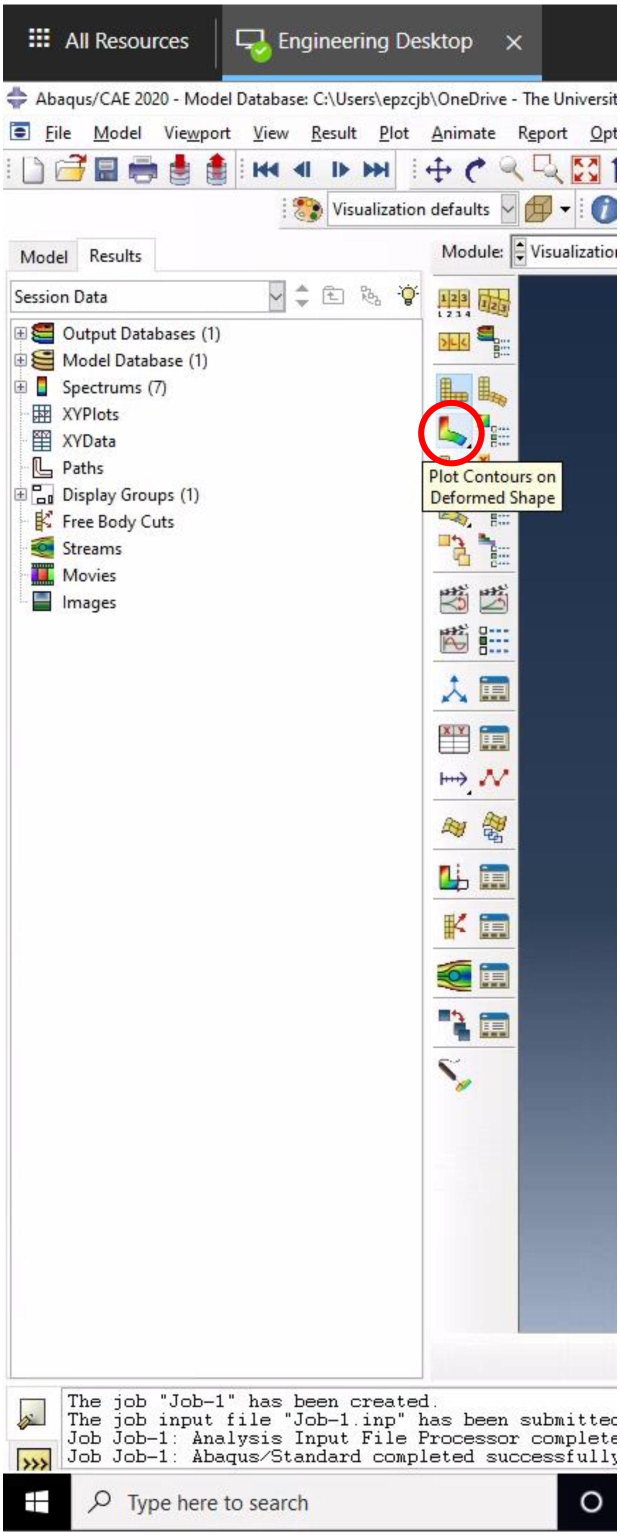
33. When the job has Completed, click Results in the Job Manager to start the Post-Processor



The window will change as below



34. Click Plot Contours on Deformed Shape to view the von Mises stresses on the exaggerated deformed geometry



You should see the stress contour results below, note the Deformation Scale Factor is 7.428 meaning the deformation results are exaggerated by more than 7 times.