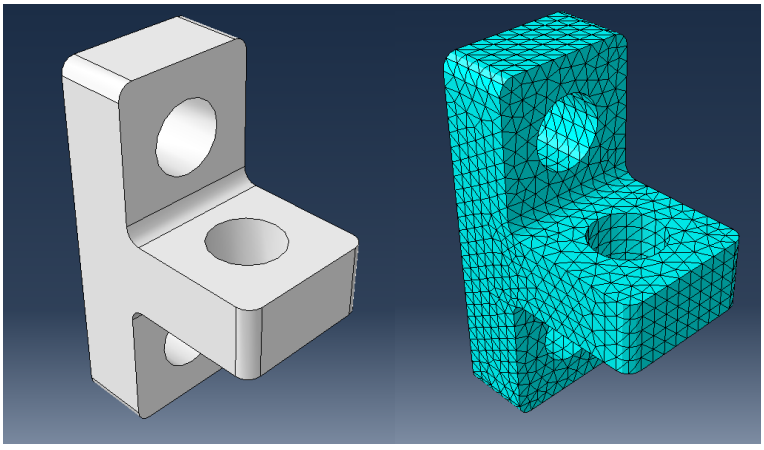
University of Nottingham
Department of Mechanical, Materials and Manufacturing Engineering

Computer Modelling Techniques

FE-WE1

Worked Example 1 using the ABAQUS FE Software

Introduction, Geometry and Initial Mesh

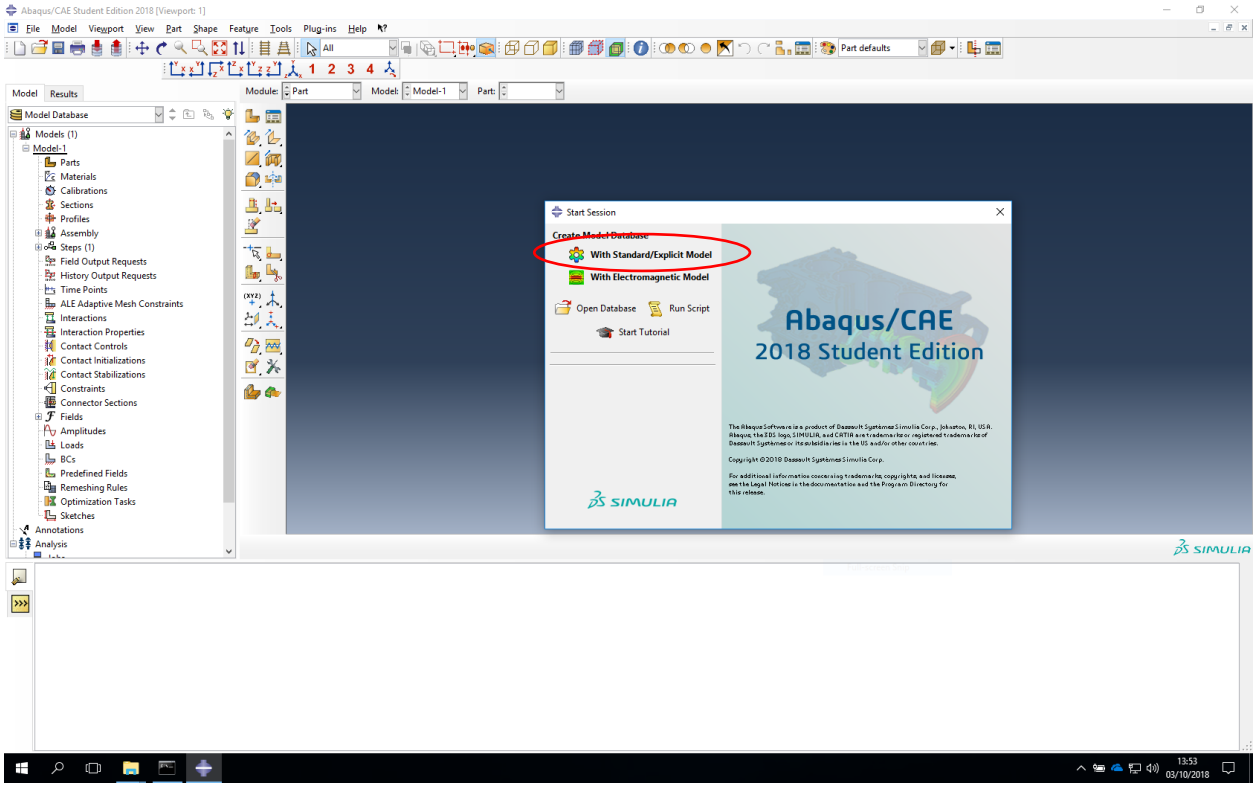


Options for accessing Abaqus:

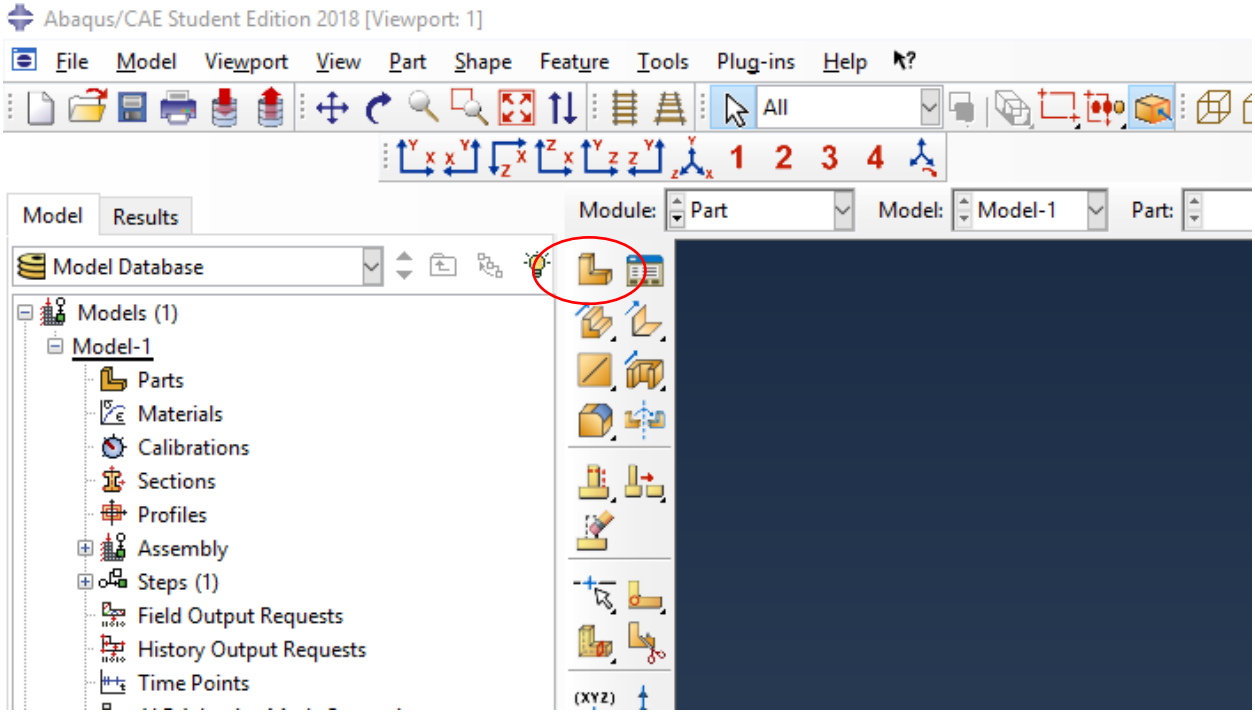
Abaqus student edition is available from: https://academy.3ds.com/en/software/abaqus-student-edition (you will need to create an account – analysis is limited to structural models of up to 1000 nodes) or you can access in the (engineering) computer rooms.

Steps:

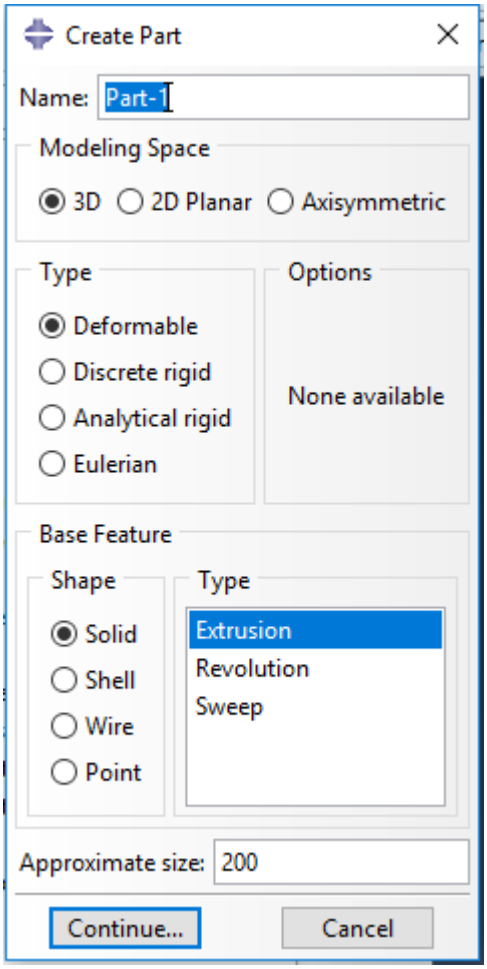
1. Open Abaqus/CAE and select *Create Model Database With Standard/Explicit Model*



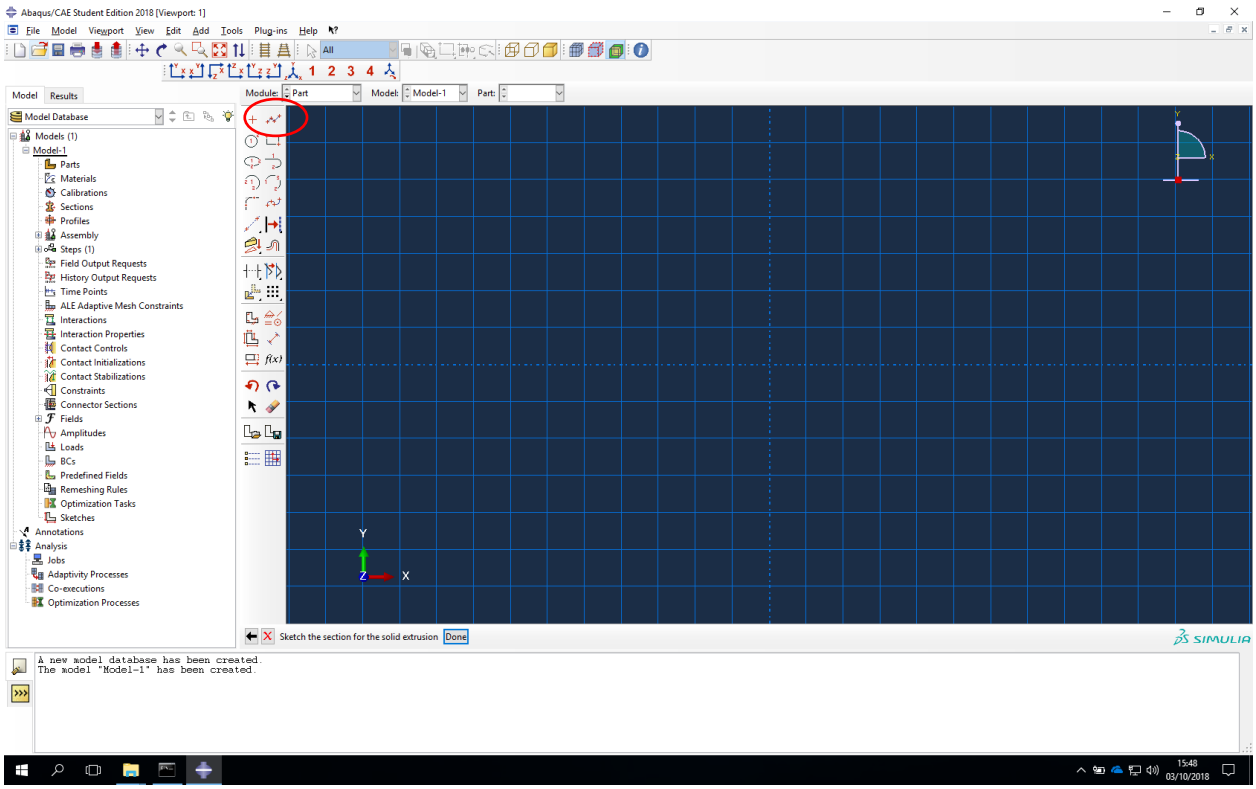
2. Select the *Create Part Icon*



3. Select the default settings (but read and take note – 3D, Deformable, Solid, Extrusion) and click *Continue…*



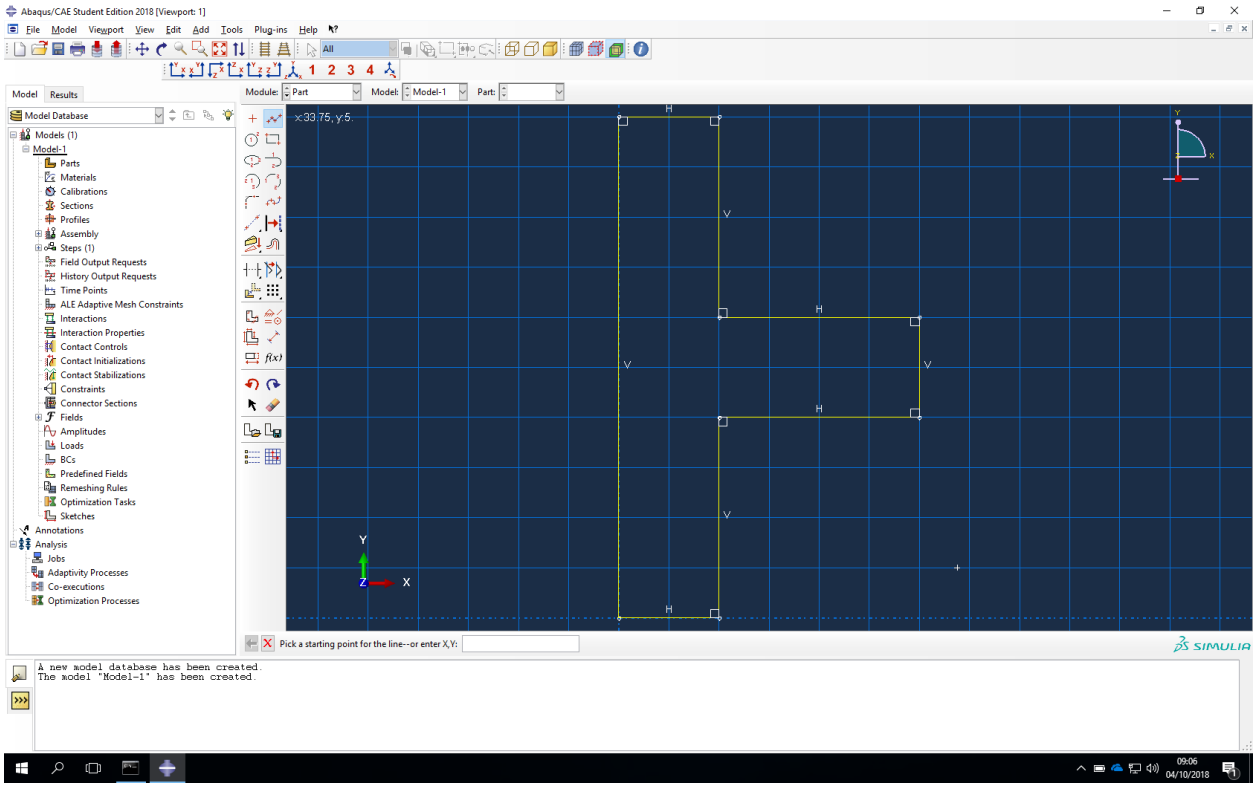
4. You will be taken to the screen to *Sketch the section for the solid extrusion* select the *Create Lines: Connected* tool



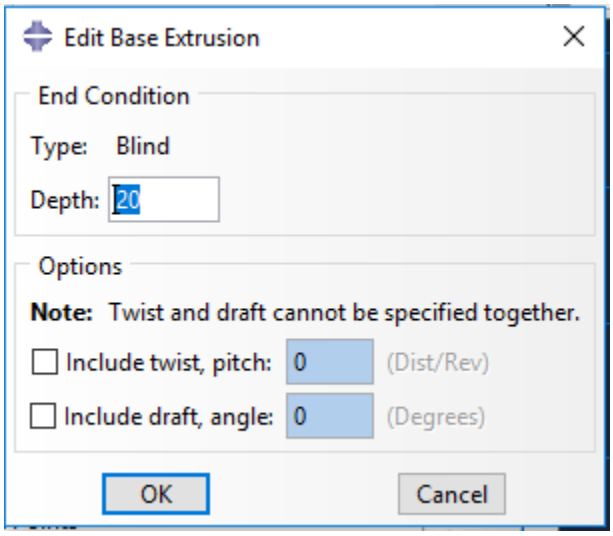
Enter the coordinates below:

x,y
0,0
10,0
10,20
30,20
30,30
10,30
10,50
0,50
0,0

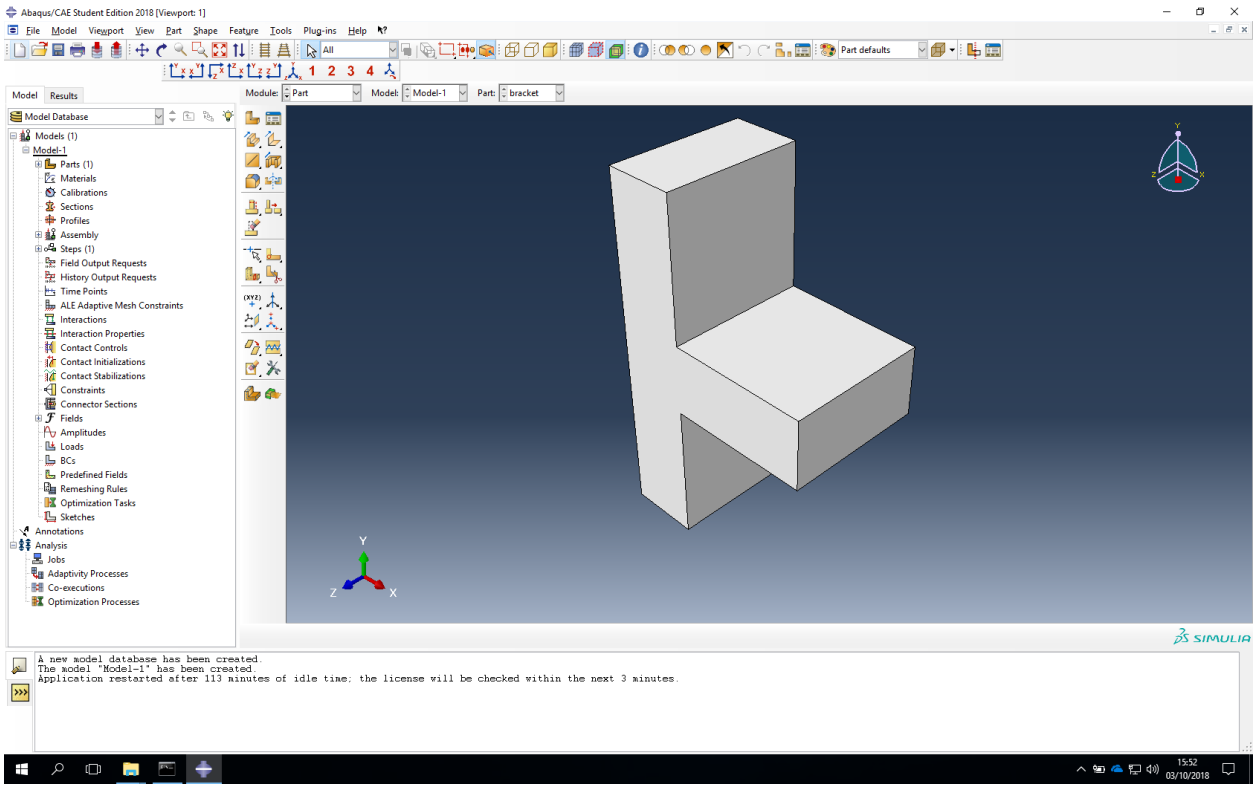
5. The resultant geometry should be as shown below



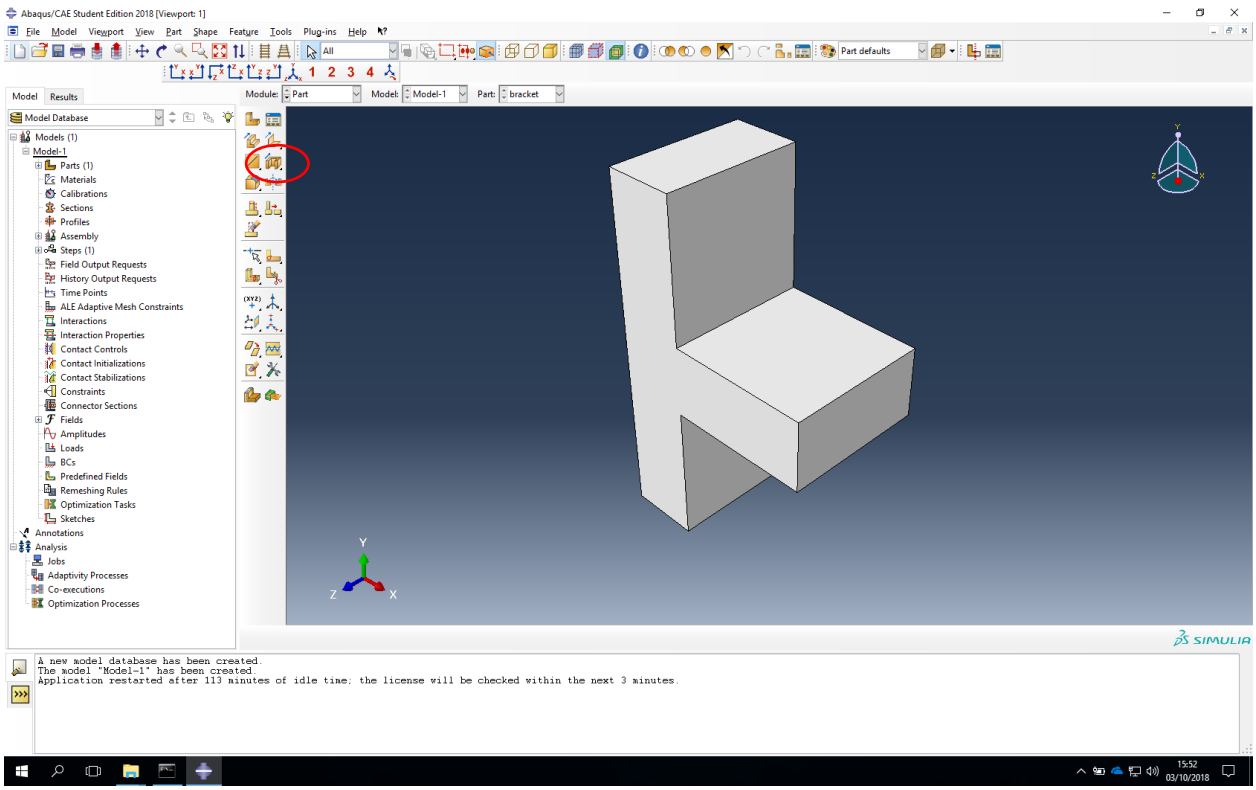
6. Click *Done* and you should be presented with the window below, select the OK button to keep the default settings (*Depth: 20*)



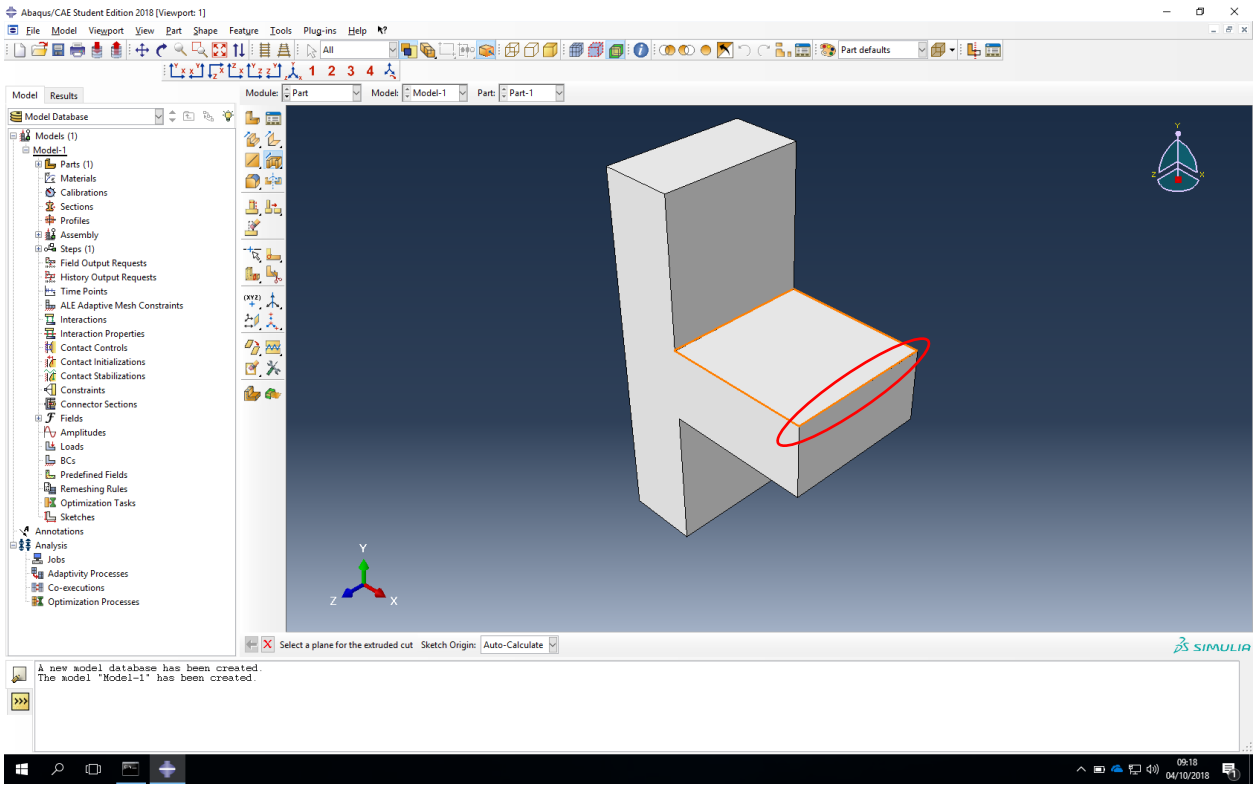
7. This completes the main body of the bracket shape, as shown below



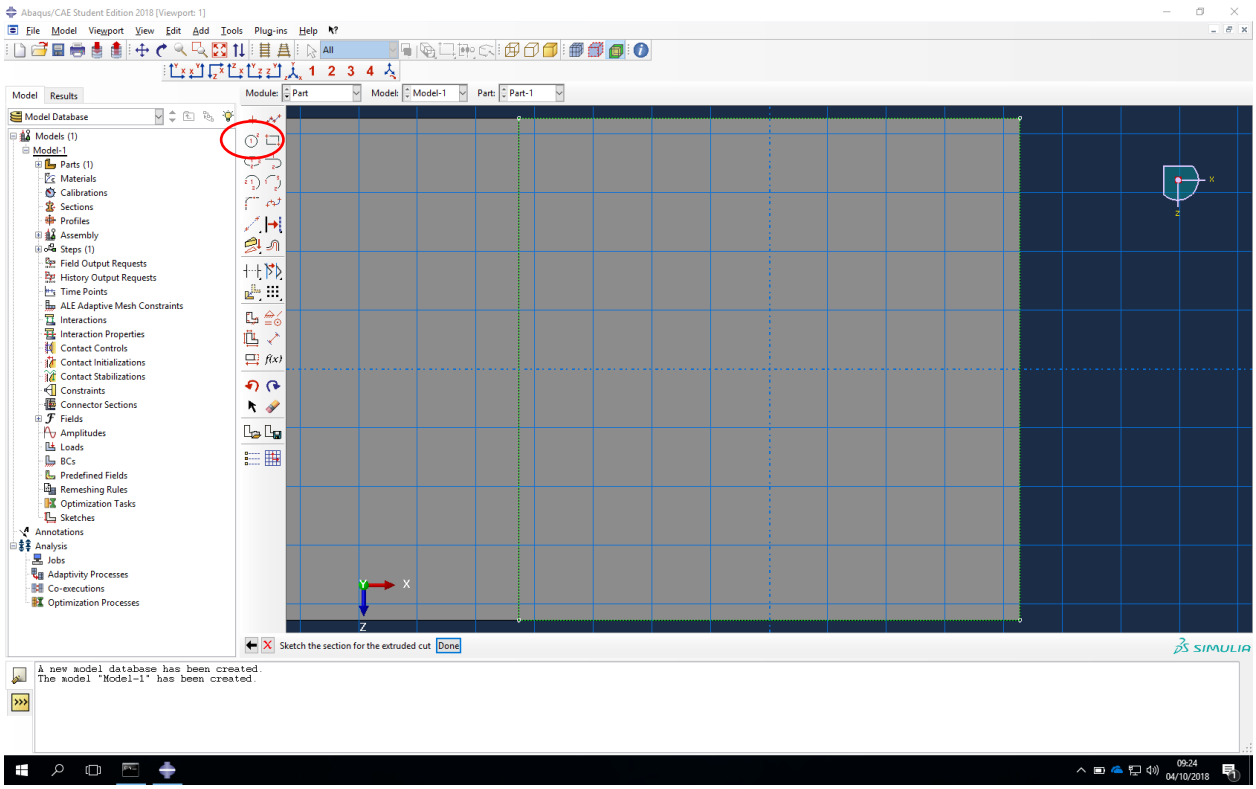
8. To add the holes, select the *Extrude Cut* icon



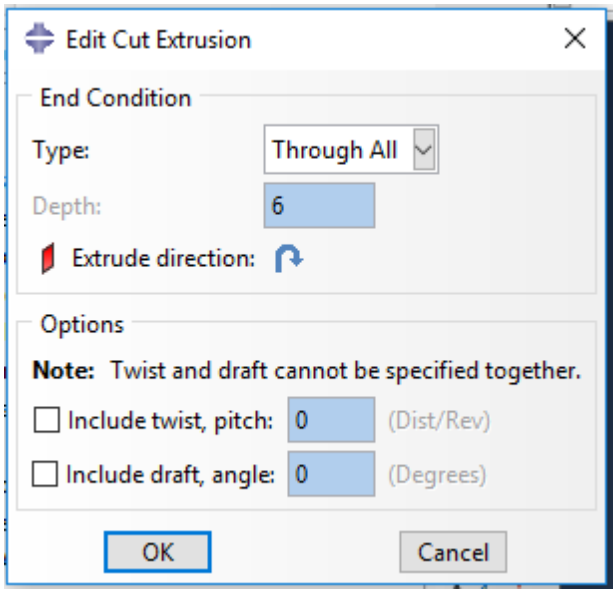
9. You will be asked to *Select a plane for the extruded cut*, select the top plane of the protrusion, you will then immediately be asked to *Select an edge or axis that will appear vertical and on the right*, select the front edge of the plane as indicated


Abaqus/CAE Student Edition 2018 [Viewport: 1]
File Model Viewport View Part Shape Feature Tools Plug-ins Help
Model Results
Model Database
Models (1)
Model-1
Parts (1)
Materials
Calibrations
Sections
Profiles
Assembly
Steps (1)
Field Output Requests
History Output Requests
Time Points
ALE Adaptive Mesh Constraints
Interactions
Interaction Properties
Contact Controls
Contact Initializations
Contact Stabilizations
Constraints
Connector Sections
Fields
Amplitudes
Loads
BCs
Predefined Fields
Remeshing Rules
Optimization Tasks
Sketches
Annotations
Analysis
Jobs
Adaptivity Processes
Co-executions
Optimization Processes
Module: Part
Model: Model-1
Part: Part-1
Select a plane for the extruded cut
Sketch Origin: Auto-Calculate
SIMULIA
A new model database has been created.
The model "Model-1" has been created.
09:18
04/10/2018

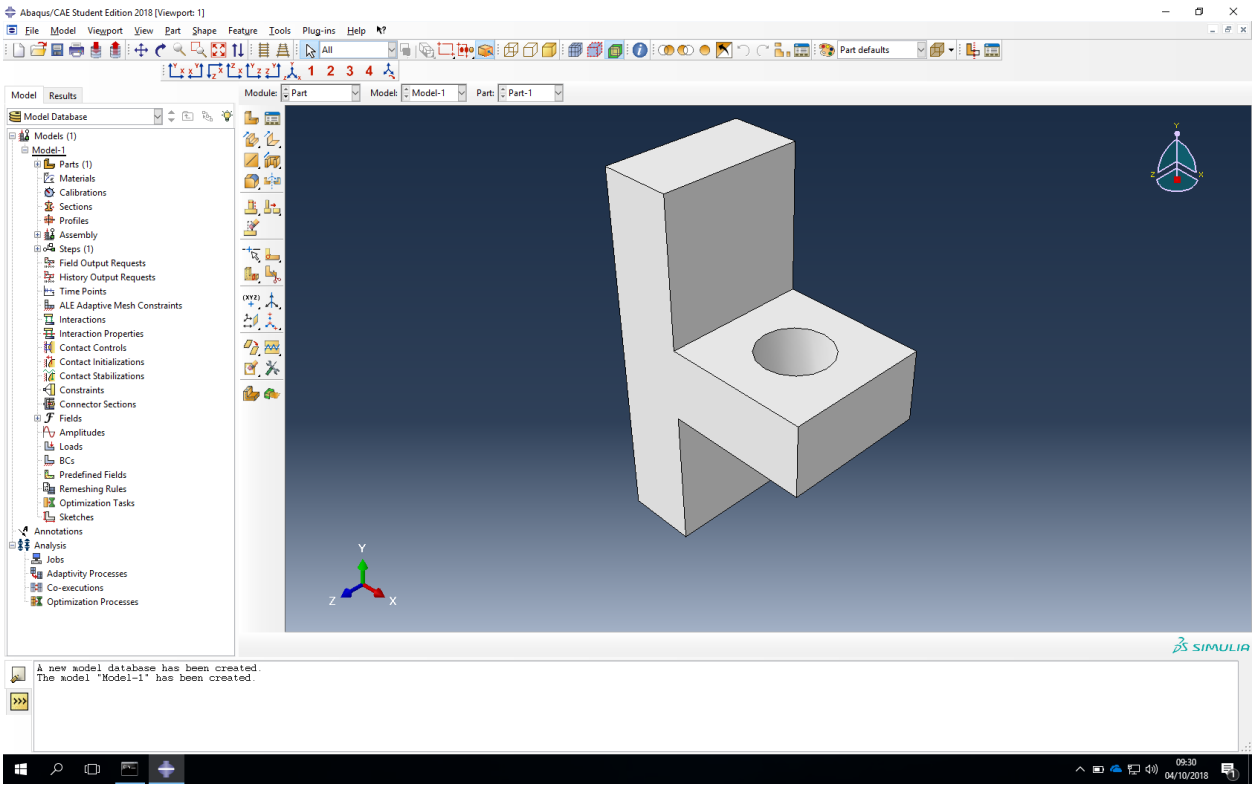
10. You will be presented with the screen below, select the *Create Circle: Center and Perimeter* icon



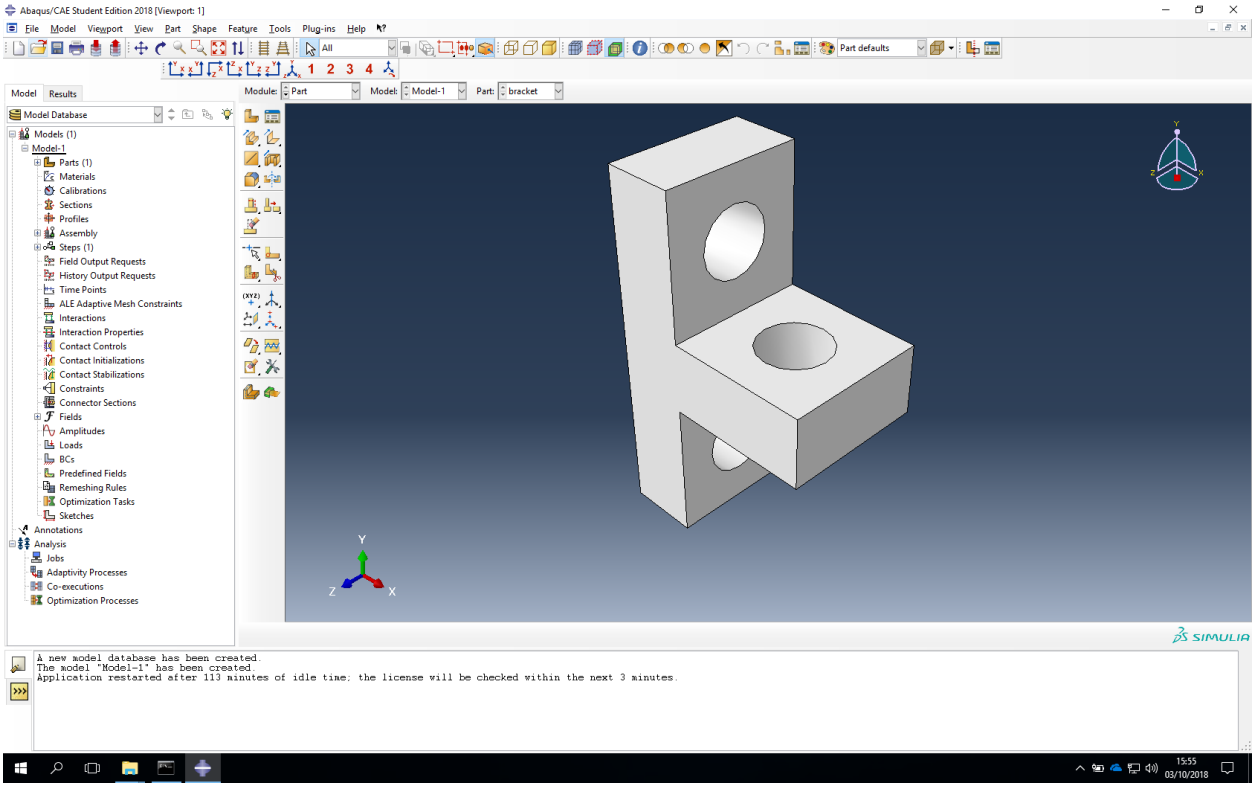
11. Create a circle with the origin at *(0,0)* of *radius 5mm* click *Done* to bring up the *Edit Cut Extrusion* window and select the default settings (Through All, check the extrude direction arrow on the viewport) by clicking *OK*.



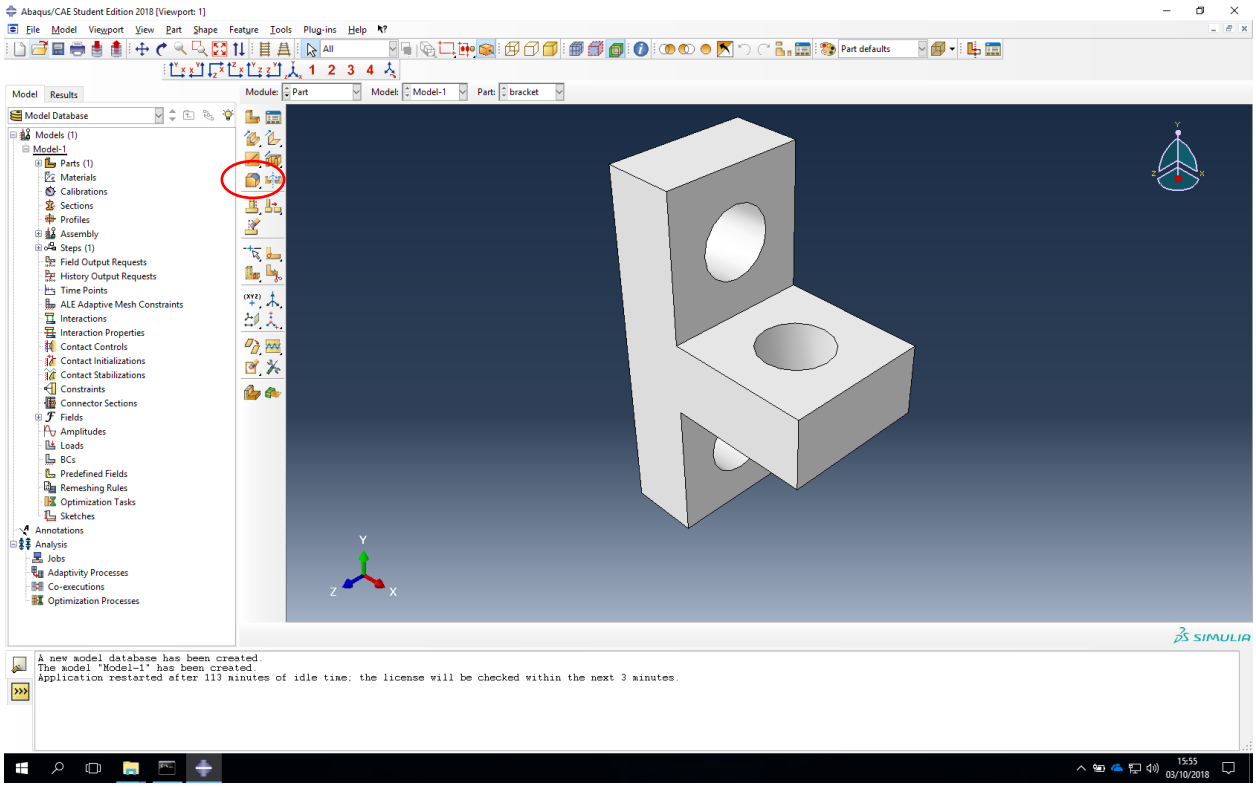
12. This should result in the geometry below.



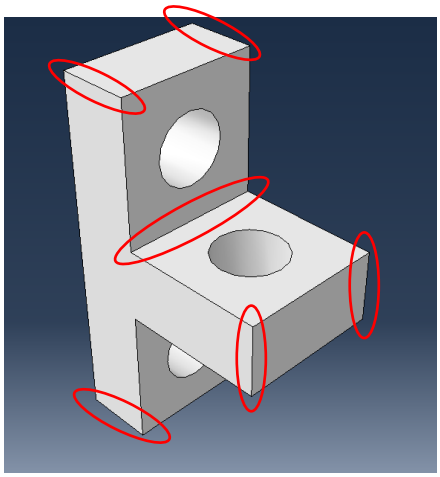
13. Add the holes in the top and bottom sections using the same approach



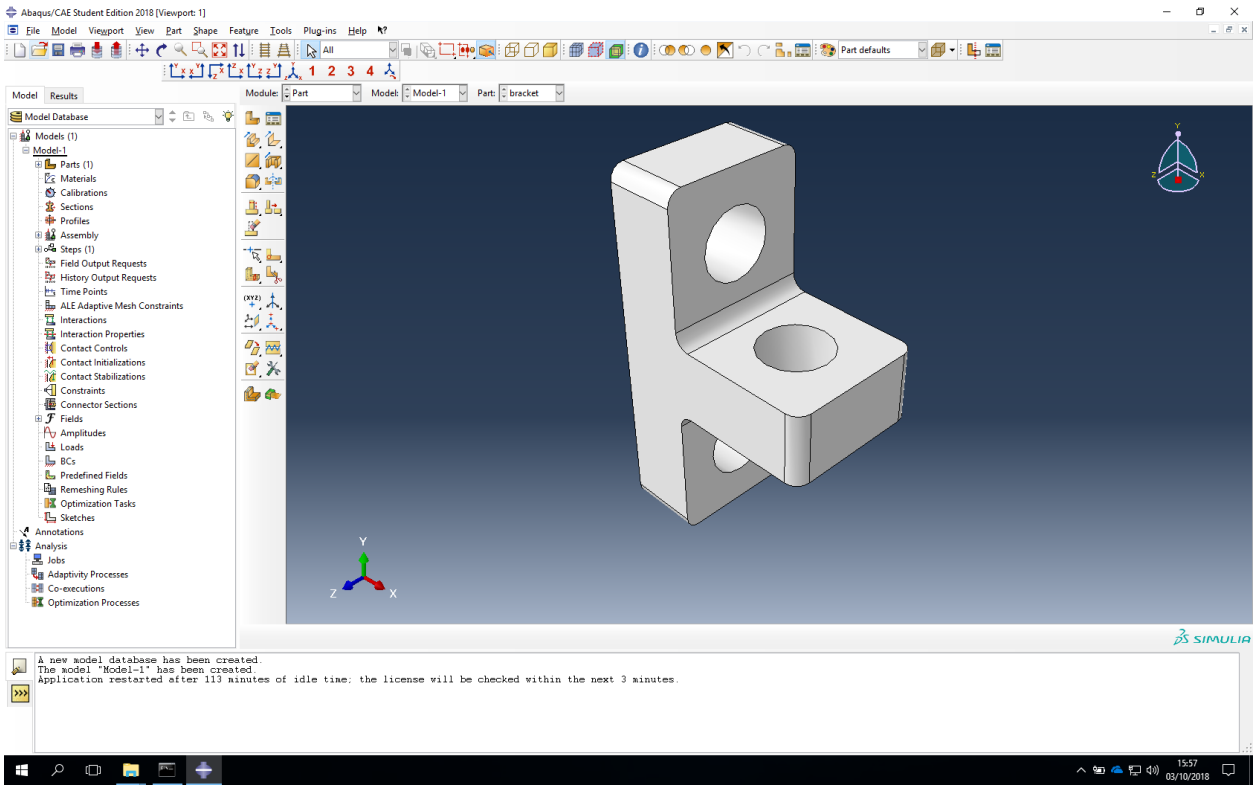
14. Select the *Create Round or Fillet* icon



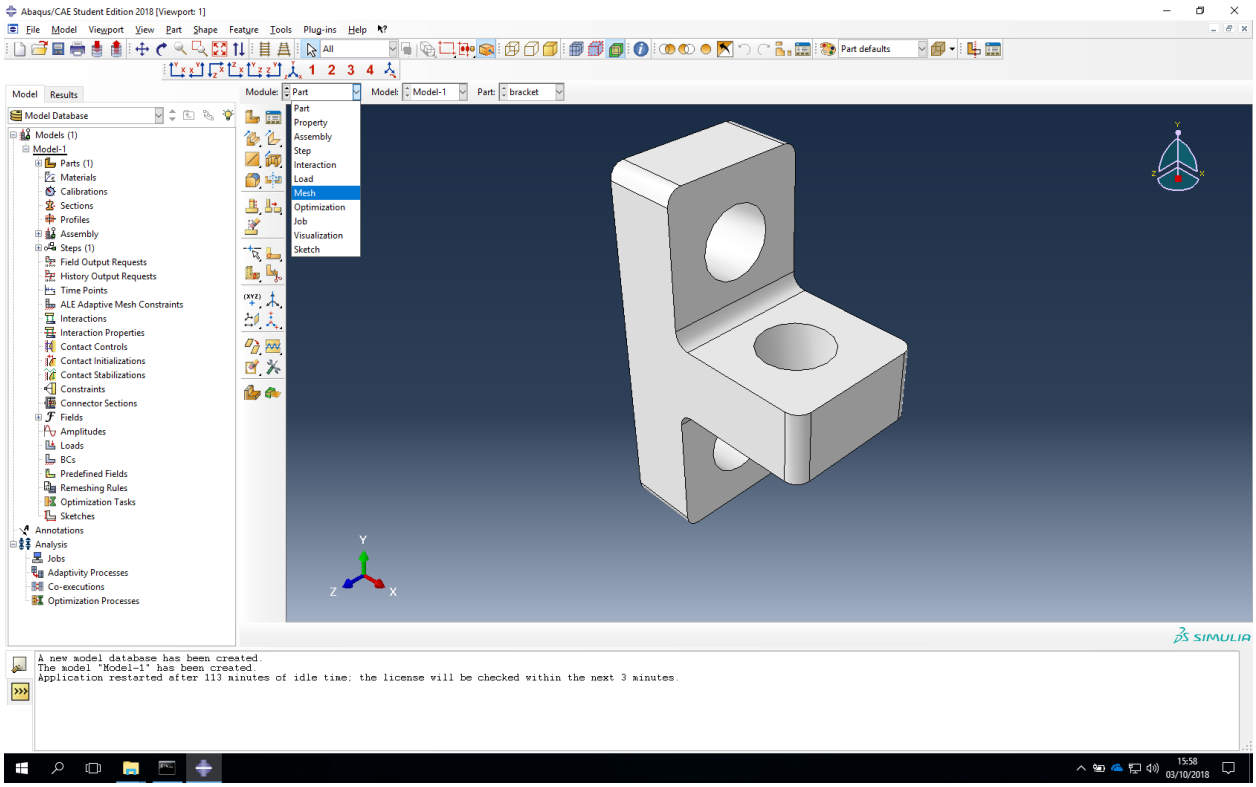
15. Select the edges indicated below (plus the two corresponding edges hidden in this view and not indicated, use shift for multiple select)



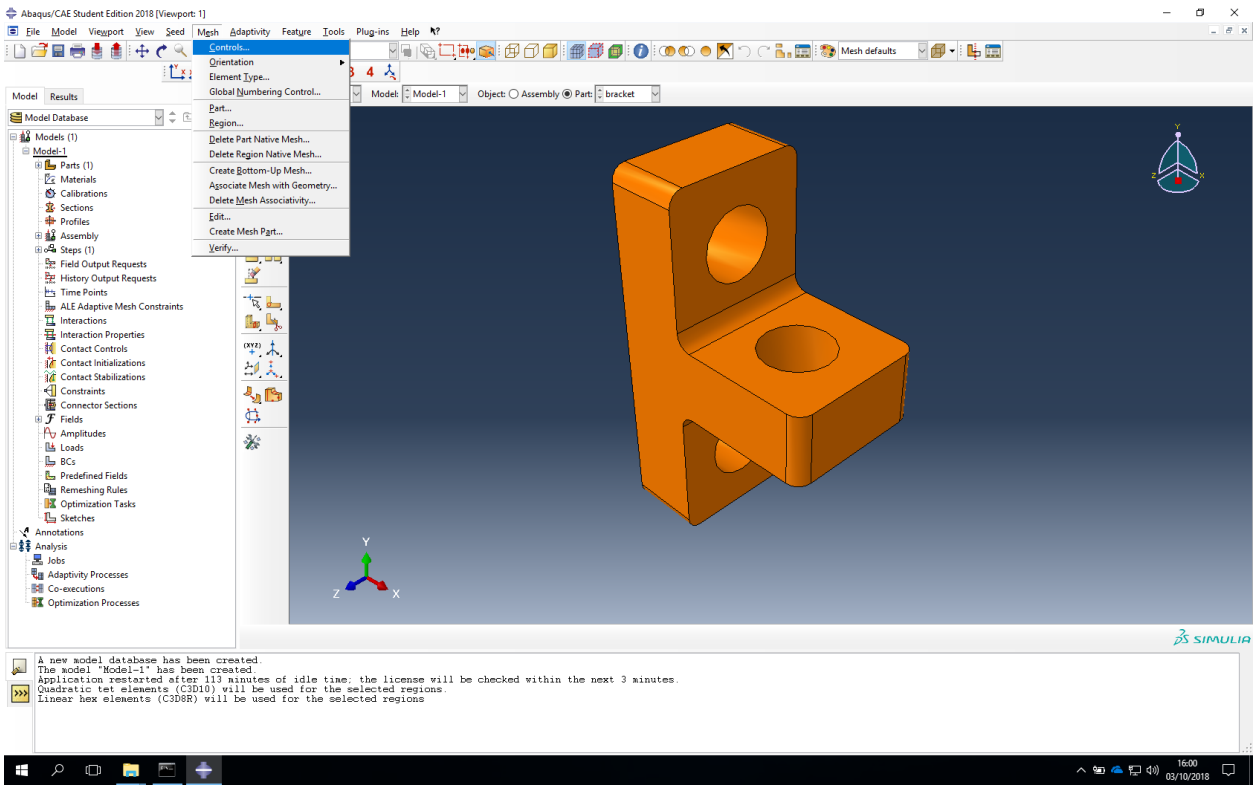
16. Click *Done* when all are selected and enter the *Radius as 2mm*. This will lead to the geometry below.



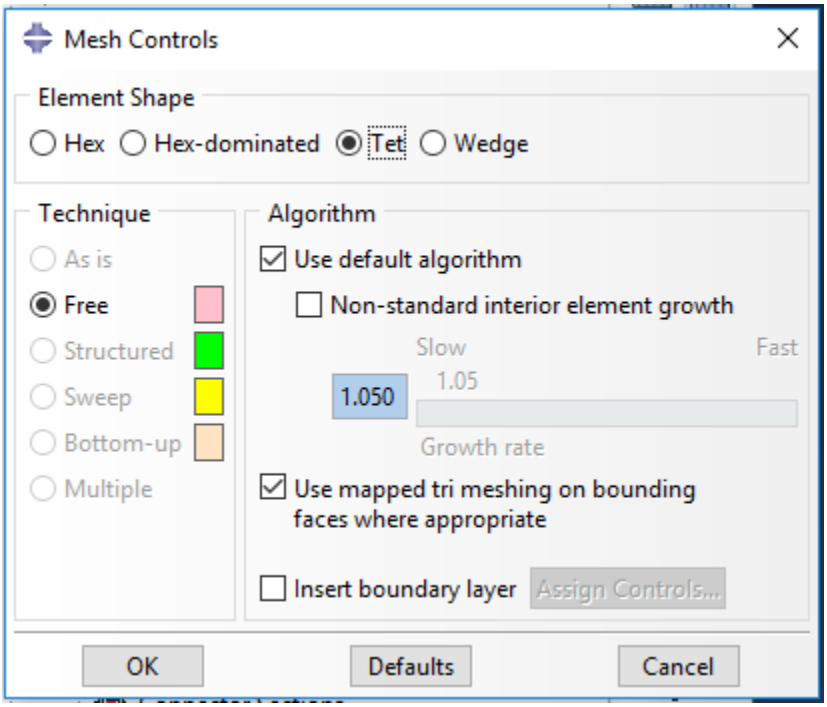
17. Select the *Mesh Module*



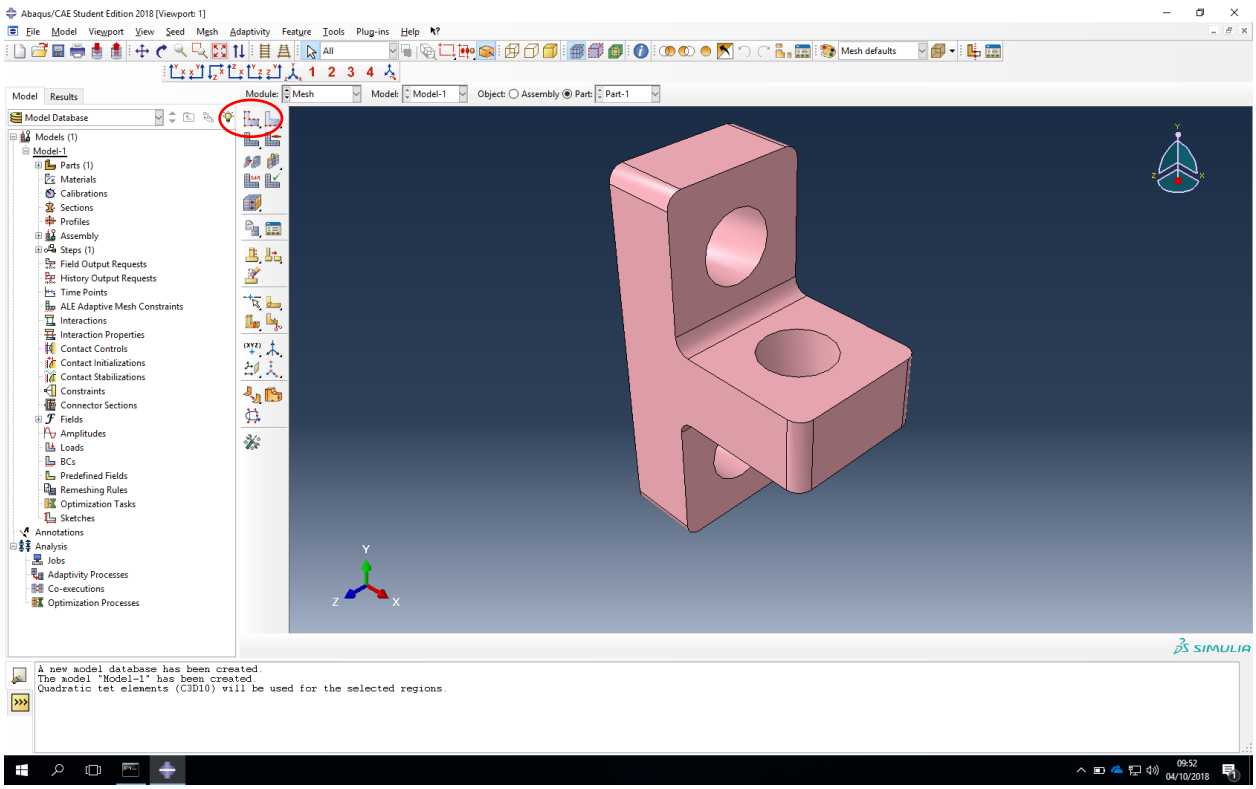
18. Select *Controls…* under the *Mesh menu*



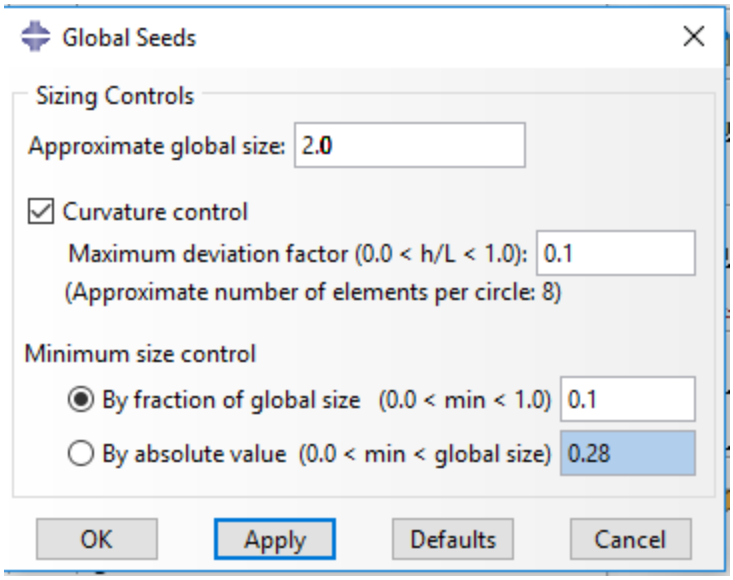
19. In the *Mesh Controls* window, change the selections to *Tet*, *Free*, select *OK* to choose the default options. The bracket will turn pink to indicate the applied Mesh Controls.



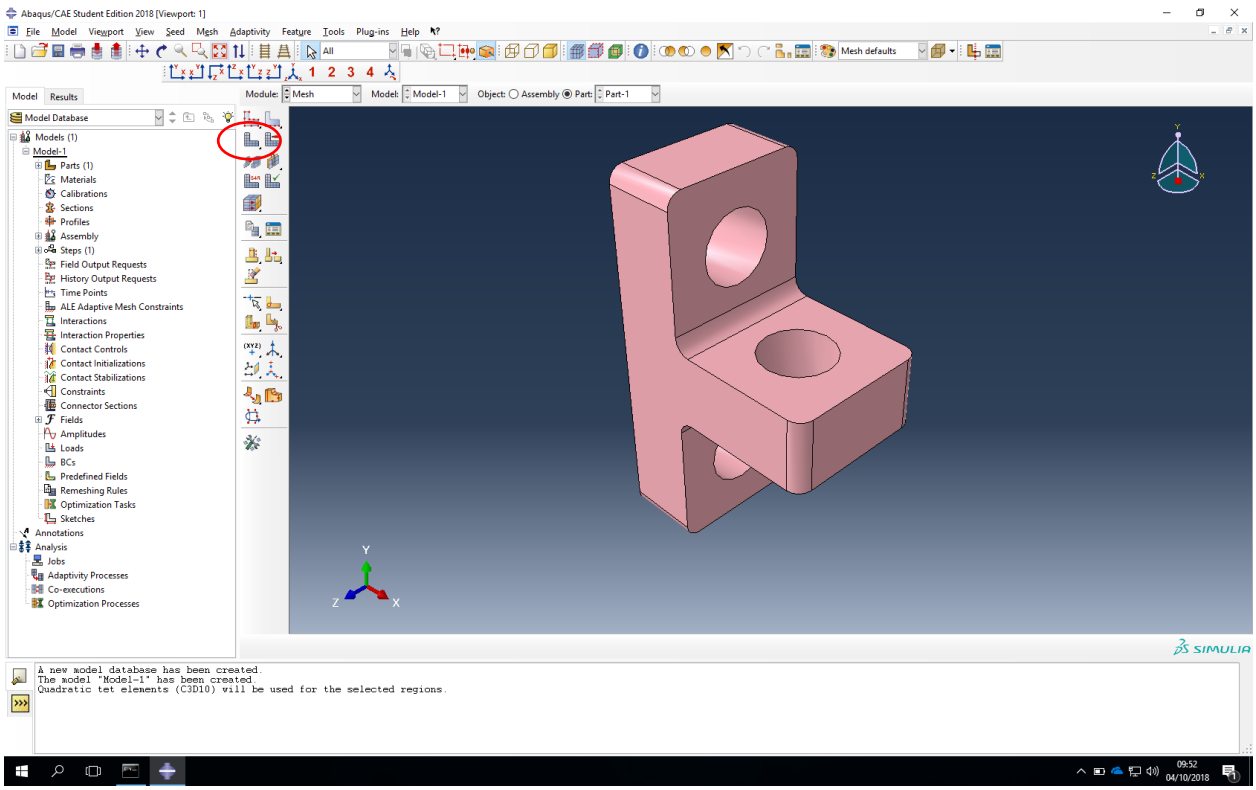
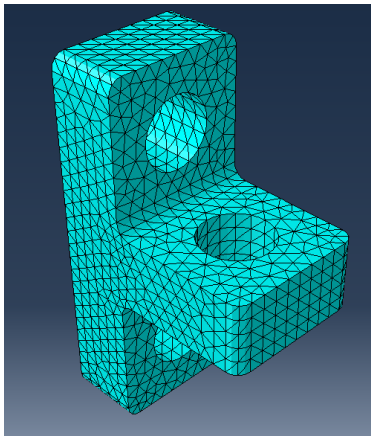
20. Select the *Seed Part* icon



21. In the *Global Seeds* window, change the *Approximate global size* to 2.0 and click *OK*, the seed points will be shown in the viewport, click *Done*.



22. Select the *Mesh Part* icon and click *Yes* to mesh the part



23. The resultant mesh should be similar to that shown below. Save your model as *bracket.cae*, this will save the geometry, mesh and settings applied.