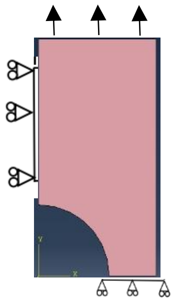
Coursework Submission Template
(Fill and submit as PDF file)

Student ID:	**20157875**
Student Name:	**Deen Sooky**
Module:	Computer Modelling Techniques
Coursework:	FEA

Task A

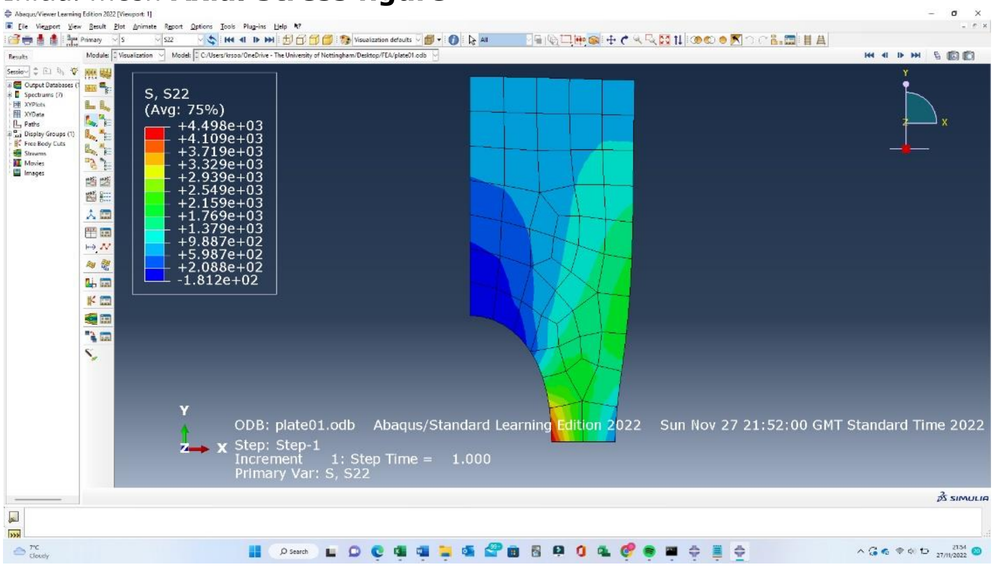
State *r* value used [mm]:	**15**
State *σ* value used [MPa]:	**875**

INFORMATION	**DETAIL**	**REASONING**
GEOMETRY	2D plane stress geometry Continuum elements Length of edge undoing stress = 50mm Arc radius = 15mm Applied stress = 875MPa 	Due to the geometry being symmetrical and the stress acting uniformly only a quarter of the geometry is ne needed for analysis.
MATERIAL PROPERTIES	E = $69000\ N/mm^2$ V = 0.33	Value of the young's modulus and poison's ratio of aluminium
ANALYSIS TYPE	Elastic Stress analysis	Needed for a linear elastic FE model
DISPLACEMENT BOUNDARY CONDITIONS	Two displacement conditions: - Zero y-displacement on the bottom edge - Zero x-displacement on the left edge	Avoids rigid body movement
APPLIED LOADS	Uniform tensile stress on top edge Applied stress is a negative value	Geometry is being stretched where upwards is indicated by negative values and acts on the top surface

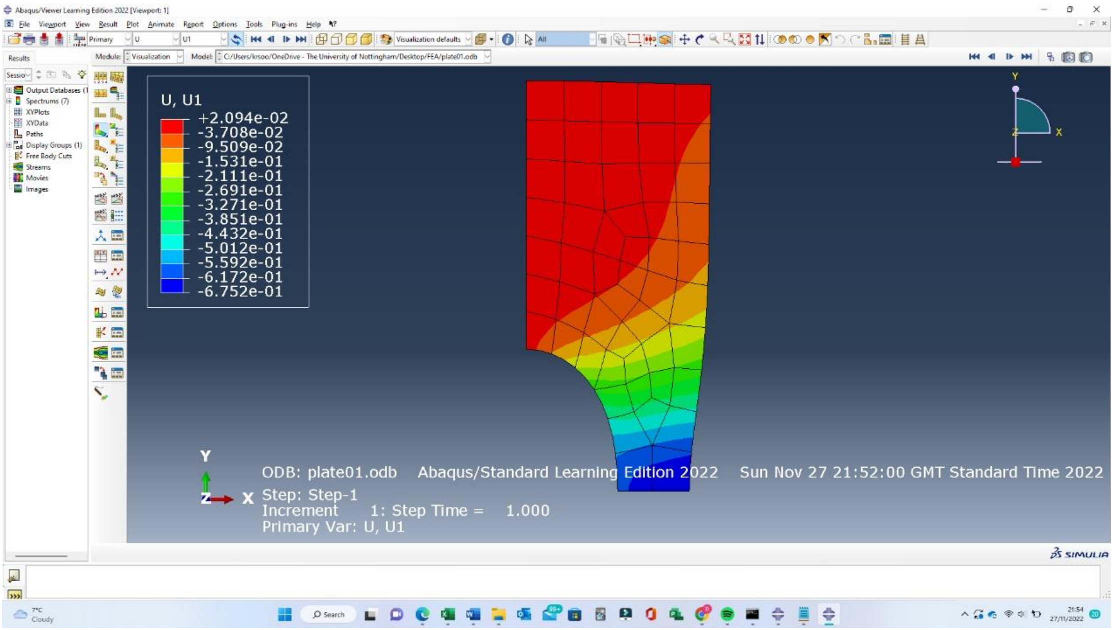
ELEMENT TYPE	8-node isoperimetric quadratic element 2x2 integration points Element code in ABAQUS – CPS8R	Geometry has unusual boundary shapes hence Elements have a quadratic element formed of 8 nodes
ELEMENT SIZE	5mm	Size of each element is 5mm
OTHER INFORMATION	Aim is to determine the stress distribution in the component	See how stress changes within a component

Task A2

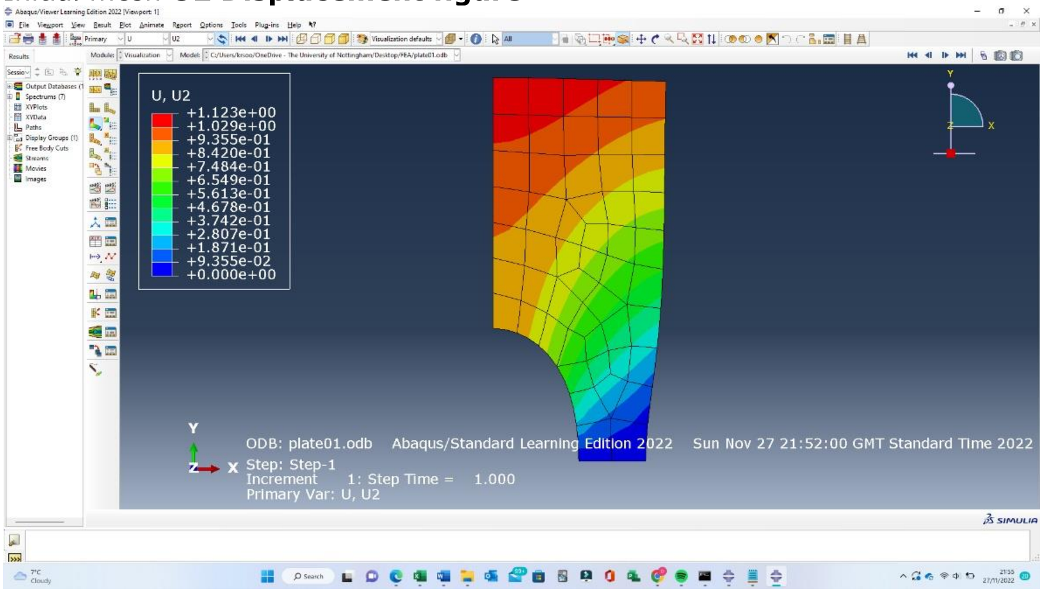
Initial mesh **Axial Stress figure**



Initial mesh **U1 Displacement figure**



Initial mesh **U2 Displacement figure**

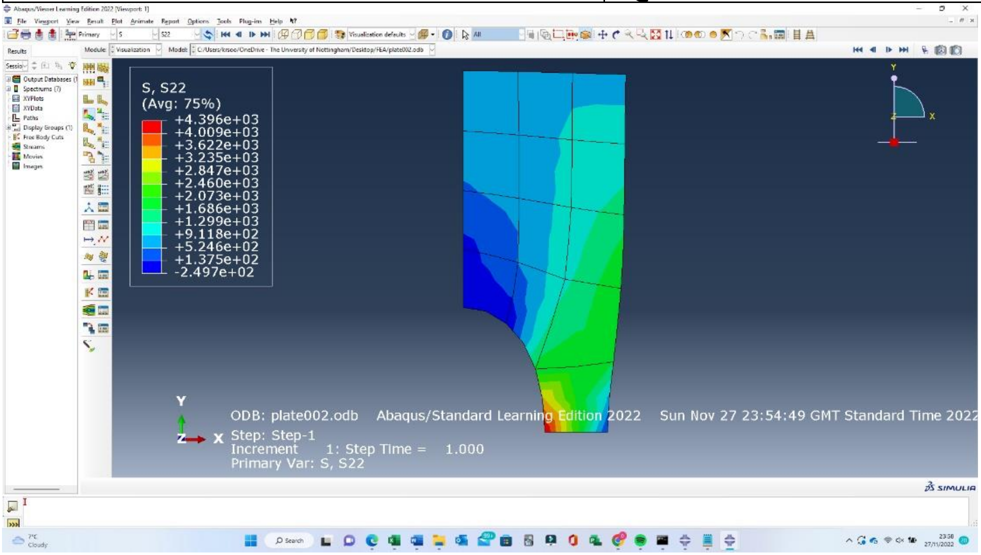


Task B

Task B1

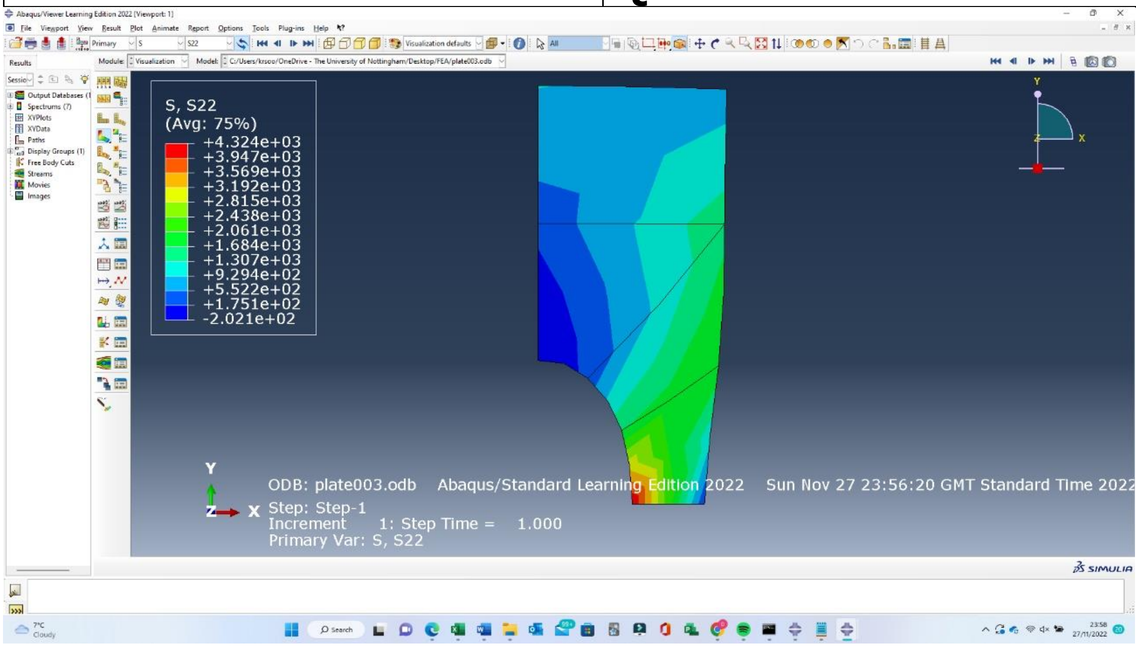
Mesh 1: Axial Stress figure

State element size [mm]:	**10**
State element type:	**Quad**



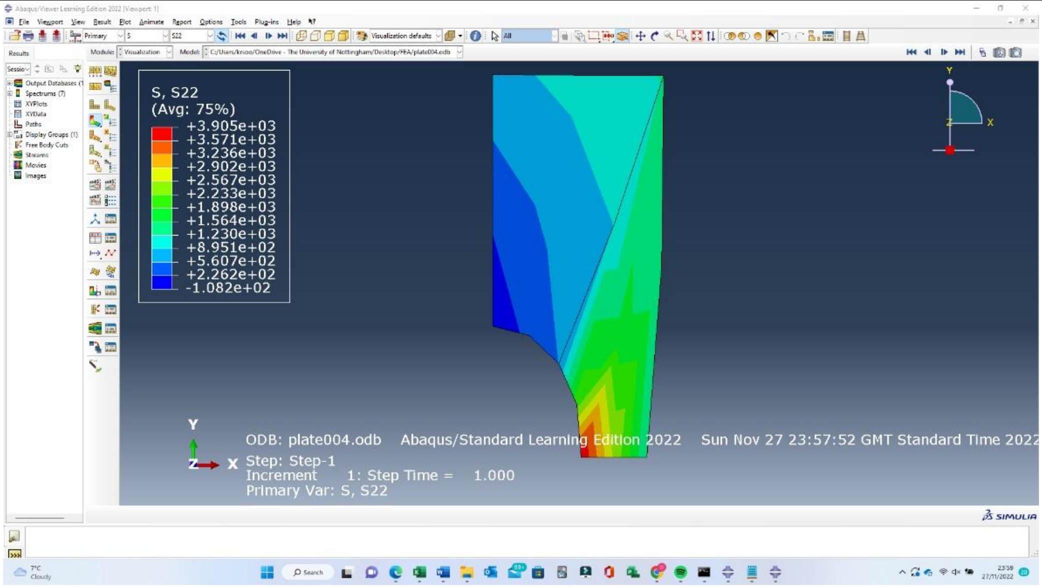
Mesh 2: Axial Stress figure

State element size [mm]:	**20**
State element type:	**Quad**

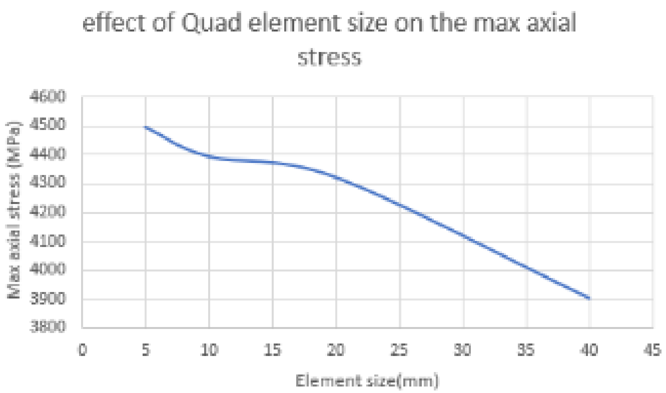


Mesh 3: Axal Stress figure

State element size [mm]:	**40**
State element type:	**Quad**



Task B2: Graph of the effect of element size and type on the solution



Explanation of chosen variable

Insert answer below:
I decided to use max axial stress as my chosen variable because the max axial stress will depend on the size of the elements only and therefore won't be affected by other variables. As a result, you can clearly see increasing the element size will decrease the max axial stress.

Task B3

Insert answer below:
From the graph you can see as the element size increases the max axial stress in the y direction decreases, this is because the effect of increasing the element size would be fewer number of elements being present. The analysis of the convergence of the last displacement value depends on the number of elements, hence fewer elements mean a rougher mesh because ABAQUS may not detect or miscalculate stress concentrations as a result of the distance between the loading point and the nodes of each element being too great. From the equation $\sigma_{yy} = 0.5 \times \sigma_0(2 + \left(\frac{R}{x}\right)^2 + 3\left(\frac{R}{x}\right)^4)$ you can calculate the Y axial stress

acting the y direction. R is equal to the radius of the arc and x is the distance from the centre of the arc thus to calculate the max σ_{yy} we will consider calculating the stress at the corner of the arc which corresponds to the max axial stress in the graph by setting R=15mm and x=15mm. Hence $\sigma_{yy} = 2625Mpa$,from the graph you can clearly see the extrapolated element size of a max axial stress of 2625MPa will be over 40mm but as the geometry is not infinite the max axial stresses from the graph will be greater than the value calculated.

Task C

For Task C, use the table below for your answers.

Task	Question	Your answer *(Only show your chosen answer- Delete other options)*	Your Explanation *(A few lines of text are sufficient - there is no need to use equations or perform any calculations)*
C1	If the FE mesh is refined further in Task A, say 50 times more elements are used, the computed displacements are expected to:	(b) slightly increase	**A greater moment is applied due to the smaller area as a result the bending stiffness decreases towards True stiffness of each element, Therefore the size of the displacements increases slightly towards the true FE solution**
C2	If the dimensions in Task A are given in inches instead of mm, i.e. the length is 100 inches and the radius is 10 inches, the maximum von Mises stress value will:	(a) remain the same	**ABAQUS does not use units for variables**
C3	If Young's modulus is doubled in Task A, the computed **displacements** will:	(c) decrease	**Young's modulus is stress over strain therefore doubling the young's modulus will half the value of the strain hence displacements.**
C4	As in question C3, i.e. if Young's modulus is doubled in Task A, the computed **stresses** will:	(b) increase	**From the formula stated in C3 doubling young's modulus will mean the stress will have to double given the strain remains constant**
C5	Can beam elements be used to model the 2D problem below? Assume that the thickness in the z-direction is very small.	(a) Yes, beam elements can be used	**Firstly, the load experienced by the beam is uniaxial, the geometry of the problem is non-symmetrical and finally the area is smaller than the 2D plane.**