

Assessment 2

Unit Number: ENGR6011 (Mechanics of Solids and Numerical Methods)

Name: S M Abul Fatha Rifat

Student ID: 21446012

Answer to the question no.1

In Western Australia, the span of minor bridges can range from 2m to almost 10m. Here, the bridge spans are represented by l_1 & l_2 . Here, $l_1 = (1 + 0.02 * ID)m$ & $l_2 = (5 + 0.05 * ID)m$. We can obtain the ID value from the chart table by utilizing our student ID. When constructing bridges, Jarrah Stringer, a type of Australian Hardwood, is commonly used. The question provides the elastic properties of this wood, including Young's Modulus and Poisson's Ratio. The bridge deck is 7m wide and constructed with a concrete material that has a thickness of 100 mm. A load of 5 kN/m² is applied to the deck. We need to create a model of a stringer using a 1D beam and 3D solid in Abaqus Software. Our goal is to measure the displacement at the midspan. Next, it is necessary to develop a theoretical solution that can be compared to the FEA solutions. We will explain the QS no 1 by dividing it into four parts, where each part will be represented by a different model. First, we will calculate the theoretical displacement values for l_1 & l_2 . Then, we will compare these theoretical values with the FEA solutions for each part.

Theoretical Solution

Maximum Deflection for l_1

$$\text{Here, } l_1 = (1 + 0.02 * ID)m = l_1 = (1 + 0.02 * 105)m = 3.1m$$

The ID value is obtained from the question paper table. My student ID, which is 21446012, corresponds to the value of ID, which is 105.

For Jarrah wood,

$$l_1 = 3.1m, \rho = 835 \text{ Kg/m}^3, r = 0.2m, g = 9.8 \text{ ms}^{-2}$$

We know,

$$V = \pi r^2 l_1 = \pi (0.2)^2 * 3.1 = 0.389 \text{ m}^3$$

Here,

$$\rho = \frac{m}{V}$$

$$\text{Hence, } m = \rho V = 835 * 0.389 = 324.815 \text{ Kg}$$

We know,

$$F = mg = 324.815 * 9.8 = 3183.187 \text{ N}$$

For Concrete,

$$l_1 = 3.1m, \rho = 2400 \text{ Kg/m}^3, w = 1m, h = 0.1m \text{ \& } g = 9.8 \text{ ms}^{-2}$$

$$\text{Here, } V = l_1 * w * h = 3.1 * 1 * 0.1 = 0.31 \text{ m}^3$$

$$\text{Now, } m = \rho V = 2400 * 0.31 = 744 \text{ Kg}$$

We know,

$$F = mg = 744 * 9.8 = 7291.2 \text{ N}$$

For live load,

$$\text{Area} = 3.1 * 1 = 3.1 \text{ m}^2$$

$$F = 5000 * 3.1 = 15500 \text{ N}$$

$$\text{Total Force} = 3183.187 + 7291.2 + 15500 = 25974.387 \text{ N}$$

Now,

$$\text{Uniform load per unit length, } W = \frac{25974.387}{3.1} = 8379.52 \text{ N/m}$$

$$\text{Here, Moment of inertia, } I = \frac{\pi d^4}{64} = \frac{\pi (0.4)^4}{64} = 0.001257 \text{ m}^4$$

We know,

$$\delta_{max} = \frac{5wl_1^4}{384EI} = \frac{5 * 8379.52 * 3.1^4}{384 * 20 * 10^9 * 0.32} = 0.000401 \text{ m}$$

Maximum Deflection for l_2

$$l_2 = (5 + 0.05 * ID)m = l_2 = (5 + 0.05 * 105)m = 10.25m$$

Similarly For Jarrah wood,

$$l_2 = 10.25 \text{ m}$$

$$V = \pi r^2 l_2 = \pi (0.2)^2 * 10.25 = 1.288 \text{ m}^3$$

$$m = \rho V = 835 * 1.288 = 1075.52 \text{ Kg}$$

$$F = mg = 1075.52 * 9.8 = 10540.14 \text{ N}$$

Similarly for Concrete,

$$\text{Here, } V = l_2 * w * h = 10.25 * 1 * 0.1 = 1.025 \text{ m}^3$$

$$\text{Now, } m = \rho V = 2400 * 1.025 = 2460 \text{ Kg}$$

$$F = mg = 2460 * 9.8 = 24108 \text{ N}$$

For live load,

$$\text{Area} = 10.25 * 1 = 10.25 \text{ m}^2$$

$$F = 5000 * 10.25 = 51250 \text{ N}$$

$$\text{Total Force} = 10540.14 + 24108 + 51250 = 85898.14 \text{ N}$$

Now,

$$\text{Uniform load per unit length, } W = \frac{85890.14}{10.25} = 8379.52 \text{ N/m}$$

$$\text{Here, Moment of inertia, } I = \frac{\pi d^4}{64} = \frac{\pi(0.4)^4}{64} = 0.001257 \text{ m}^4$$

We know,

$$\delta_{max} = \frac{5wl_2^4}{384EI} = \frac{5*8379.52*(10.25)^4}{384*20*10^9*0.32} = 0.048 \text{ m}$$

FEA Setup in Abaqus Software for l_1 with 1D Beam Element

In order to calculate the displacement value, the Abaqus Software employs a systematic approach to achieve accurate analysis.

Initially, the part drawing is created with accurate geometry. Since the beam drawing is one-dimensional, just the length of the bridge span is emphasized. A 2D planar with deformable wire is utilized to obtain a 1D element in order to generate a part drawing. Simply draw a line based on the value of l_1 .

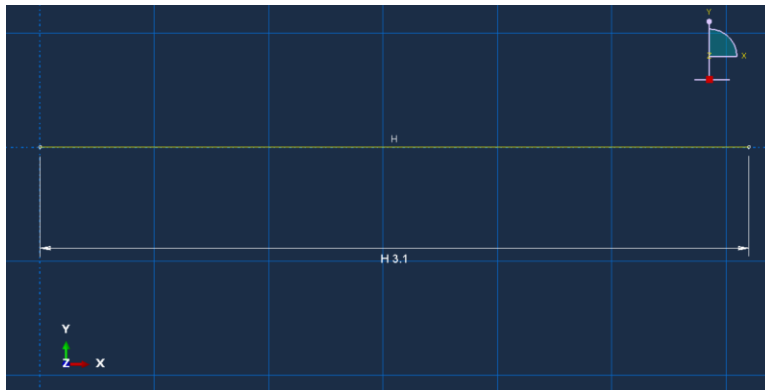


Figure 1: Sketch of Drawing.

Next, an appropriate material is defined in the property tab, with specific attributes assigned to it. For instance, the mass density is set to 835 kg/m³, the Young's Modulus is set to 20 GPa, and the Poisson Ratio is set to 0.32 for Jarrah hardwood. After that, a section is constructed and assigned by selecting the wire. Next, the assembly operation is performed by picking the part and creating an independent mesh. Next, a step is established to ensure the logical operation and precision.

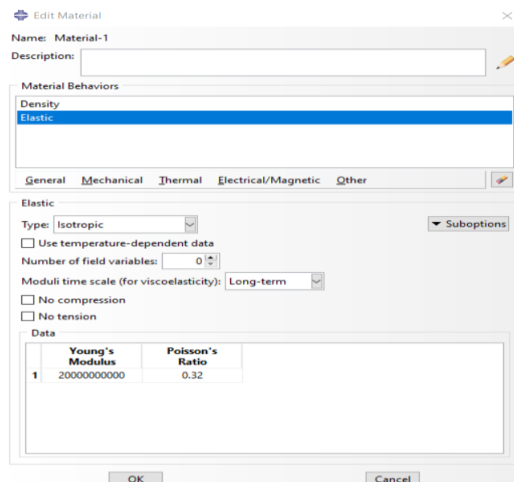


Figure 2: Property Selection and inserted data.

Furthermore, two distinct sorts of loads are exerted on the line. There are two types of loads: gravity load, which acts on the entire line, and pressure load, which acts uniformly throughout the length of the wire.

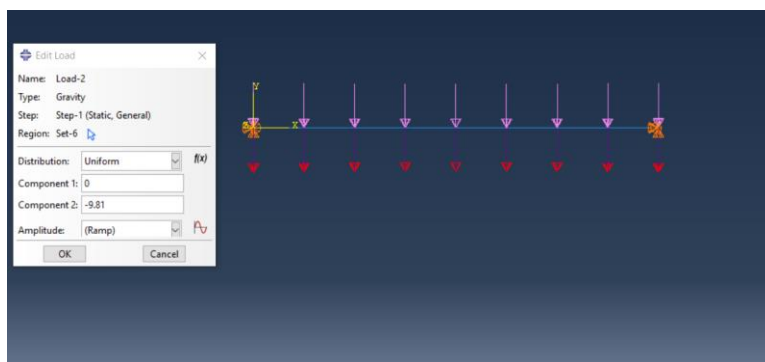


Figure 3: Red arrow denotes gravity load

During the manual calculation, we take into account three forces: one that comes from the concrete, another from the Jarrah hardwood, and the last one from the live load. In this scenario, the force exerted by Jarrah hardwood is disregarded due to the presence of live loads and the force exerted by concrete in the deck.

Hence, Total force = $7291.2 + 15500 = 22791.2\text{N}$

The wire is subject to a total force per unit length.

Uniform Load = $22791.2/3.1 = 7352\text{N}$

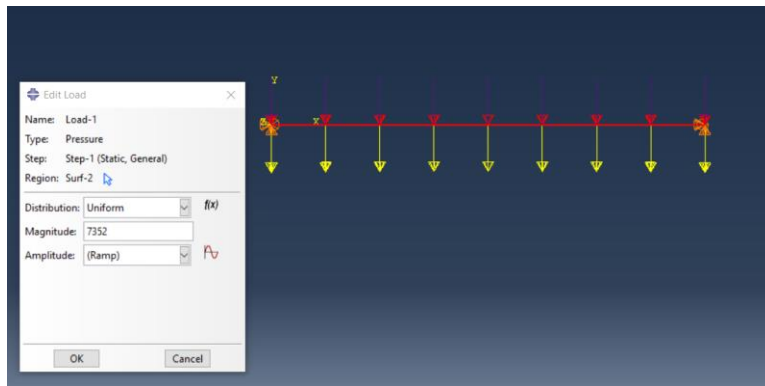


Figure 4: Red arrow denotes the uniform pressure load

Additionally, boundary conditions are imposed by following pinned condition on the two points located at opposing edges, which are held in a fixed position.

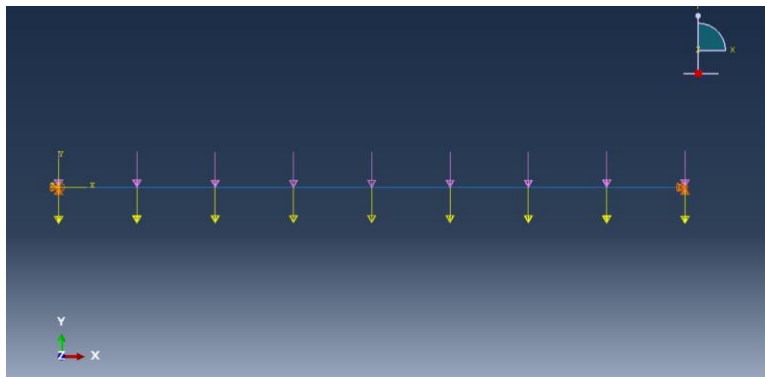


Figure 5: Orange arrow denotes the points, where boundary conditions are applied.

Furthermore, whole line is meshed by setting up the global size value 0.16. Next, a task is generated for submission. However, prior to submitting the work, we must assign the beam orientation in the property tab by choosing the wire.

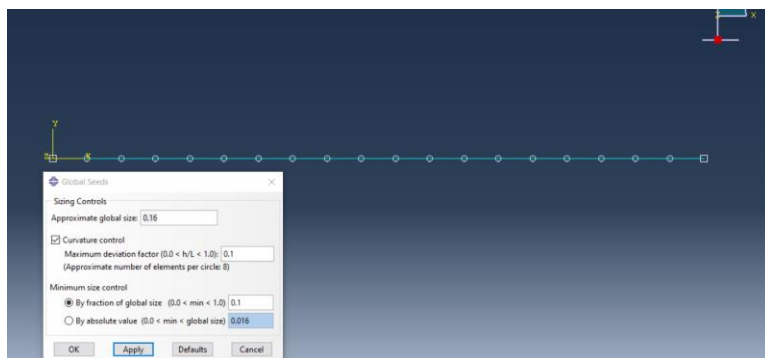


Figure 6: Seed part during Mesh

Upon completion of the submission, the ABAQUS Software displays the outcomes of stress and displacement. The outputs are shown below.

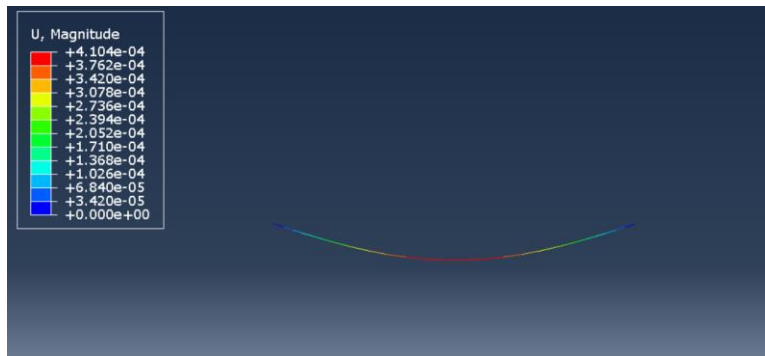


Figure 7: Magnitude for Displacement

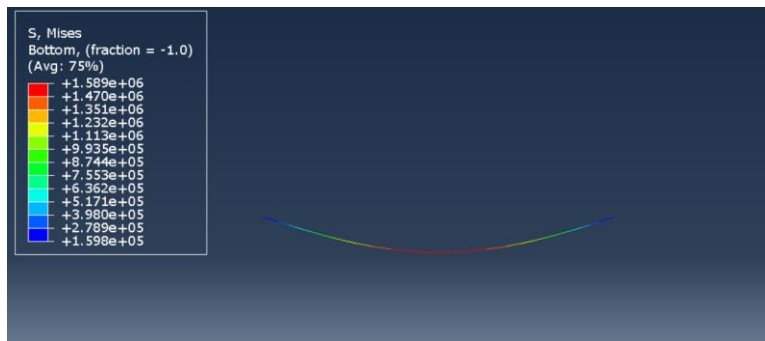


Figure 8: Magnitude for Stress

Figure 7 demonstrates that the maximum displacement value is $4.104 \times 10^{-4} \text{m}$ or 0.000401m , which coincides with the displacement value obtained from my manual computation. There are no errors in either the manual computation or the ABAQUS Software, as both displacement values are identical. Furthermore, figure 7 and figure 8 illustrate that the highest displacement and maximum stress occur at the midpoint when boundary conditions are given to the two points at opposite edges. These values gradually decrease as we go towards the two opposed edges.

FEA Setup in Abaqus Software for l_2 with 1D Beam Element

The displacement value in ABAQUS Software is determined in a similar manner as the preceding one, using a step-by-step technique. The 10.25-unit-length wire is depicted in Figure 9 by use of the 2nd planar with deformable wire option in the part drawing.

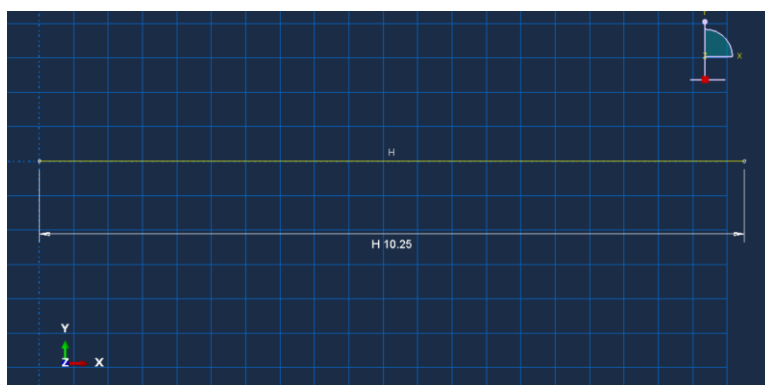


Figure 9: Part drawing for l_2

Furthermore, material properties such as mass density, Young's modulus, and Poisson's ratio can be defined on the property tab using the same characteristics as the preceding one. Next, the Section is generated and allocated by selecting the entire wire. Likewise, the assembly and stages are performed to obtain the structured analysis.

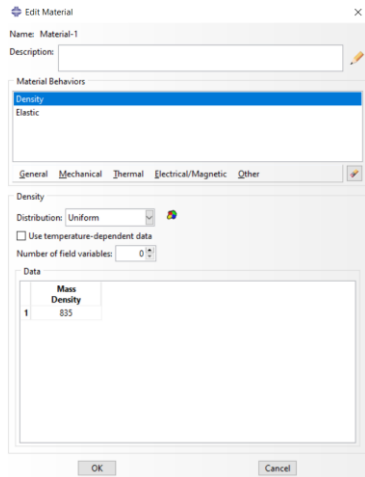


Figure 10: Created Material for Jarrah Hardwood

Similarly, two loads are applied to the wire: one due to gravity and another due to a uniform pressure load. In this situation, the force exerted by the Jarrah hardwood is disregarded since it does not contribute to the functioning of the deck.

Hence, Total force = $24108 + 51250 = 75358\text{N}$

The line is subject to a total force per unit length.

Uniform Load = $75358/10.25 = 7352\text{N}$

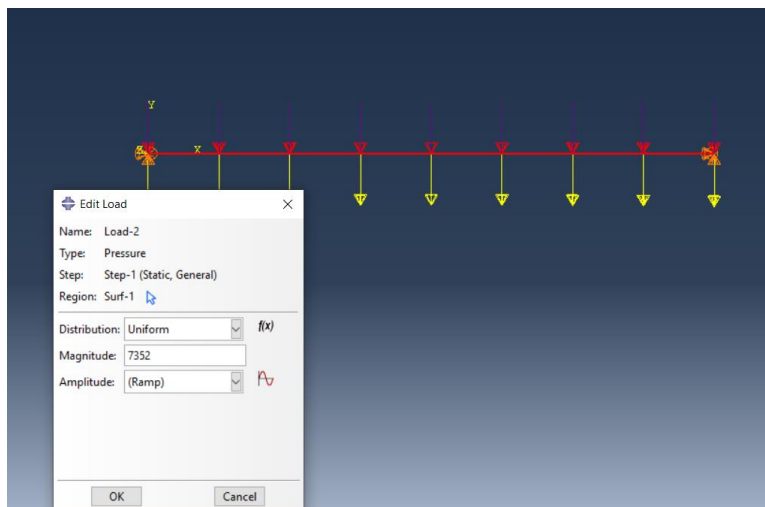


Figure 11: Red arrow denotes the uniform pressure load

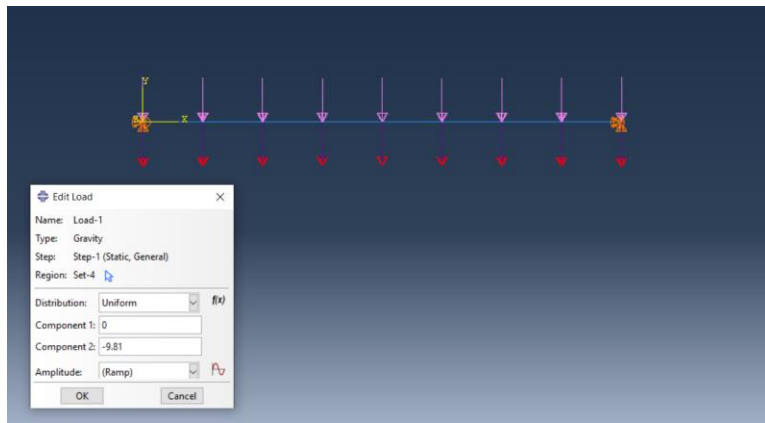


Figure 12: Red arrow denotes gravity load.

Similarly, boundary conditions are imposed by following pinned condition on the corresponding point at the opposite edge, much like in the preceding case.

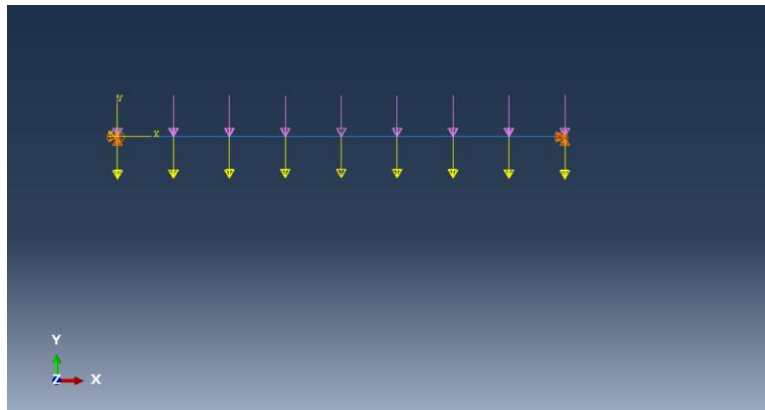


Figure 13: Orange arrow denotes the points, where boundary conditions are applied.

By configuring the global size value to 0.16, the entire wire is meshing. Submitting the work requires first assigning beam orientation in the properties tab by choosing the line, which must be done before a task can be produced.

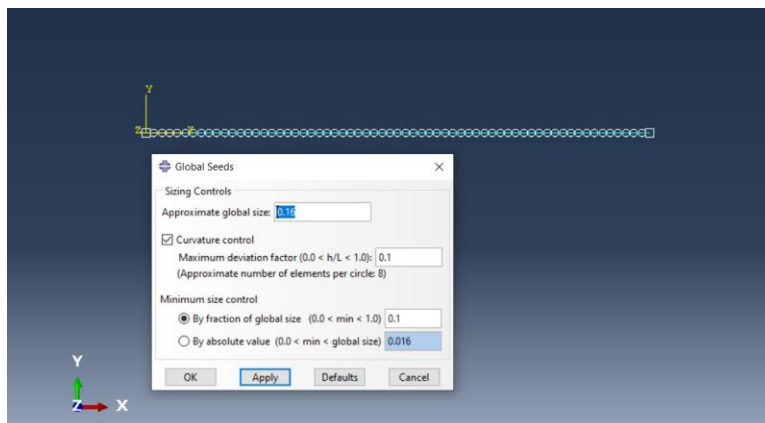


Figure 14: Seed part during Mesh

Once the job is submitted, the ABAQUS software displays the findings of displacement and stress. The results are displayed below.

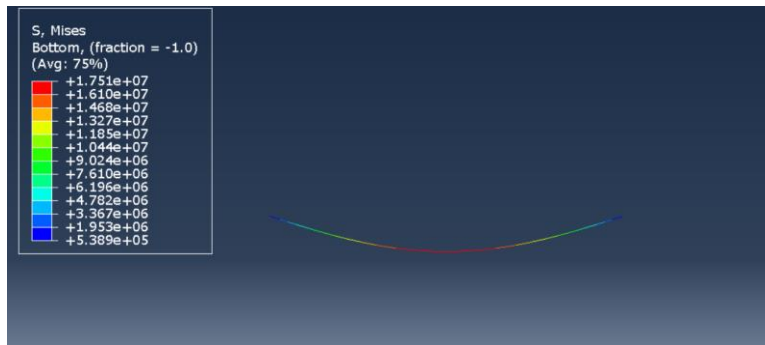


Figure 15: Magnitude of Stress in ABAQUS Software.

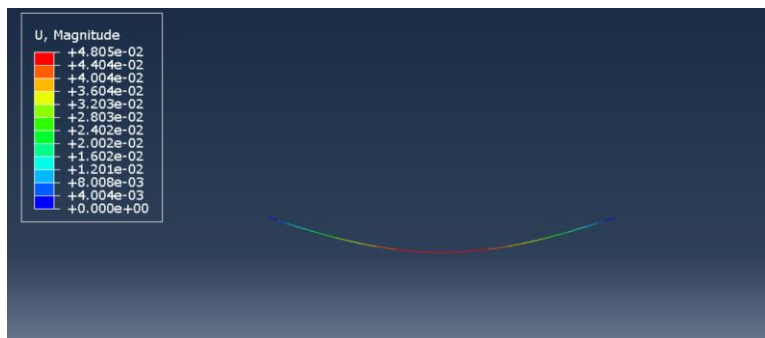


Figure 16: Magnitude of Displacement in ABAQUS Software.

Figure 16 demonstrates that the maximum displacement value is $4.805 \times 10^{-2} \text{m}$ or 0.048m . This result is consistent with the manual calculation and matches the displacement value obtained by the ABAQUS Software. Hence, it can be asserted that the manual calculation and operation in FEA software are completely accurate, with no discrepancy between them.

Additionally, Figures 15 and 16 depict the stress and displacement values at various points. They demonstrate that the maximum stress and displacement occur at the midpoint, where boundary conditions are imposed on two points at opposite edges. As one moves into two opposite edges, the stress and displacement gradually decrease.

Furthermore, in larger bridge spans, the stress magnitude is greater than in smaller bridge spans, and the displacement magnitude is likewise greater compared to the prior model. Consequently, larger bridge spans exhibit higher levels of stresses and displacements.

FEA Setup in Abaqus Software for l_1 with 3D Solid Element

In order to acquire an accurate analysis, Abaqus Software follows a series of systematic processes to find the displacement value for 3D Solid Element.

The initial step involves the creation of a part drawing by selecting a 3D solid deformable element. A circle with a radius of 0.2m has been drawn in the drawing. A line measuring 0.05m has been drawn on the top

surface of the circular to create a flat upper surface. This allows for the application of pressure load on that surface, as depicted in figure 17. Subsequently, solid extrusion has been applied by using l_1 .

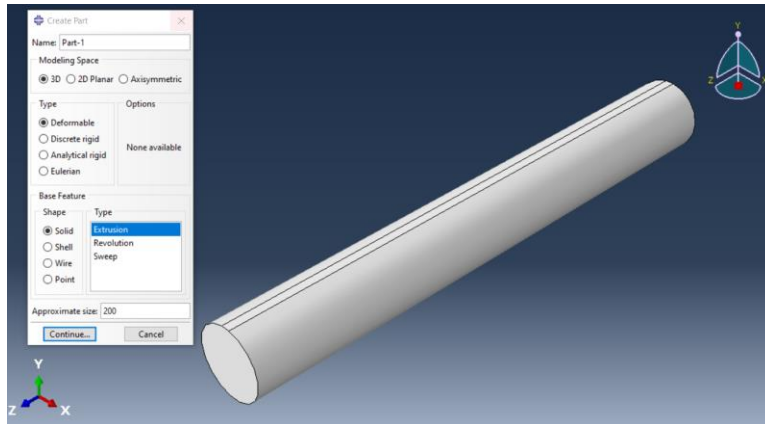


Figure 17: Part creation

Additionally, in the property tab, a material that is appropriate for Jarrah hardwood is constructed and its properties are entered, such as a mass density of 835 kg/m^3 , a Young's modulus of 20 GPa , and a Poisson ratio of 0.32 . The section is then created and assigned by choosing the solid surface. After that, the part and independent mesh are chosen and the assembly process is completed. Next, a step is developed to guarantee precision and logical operation.

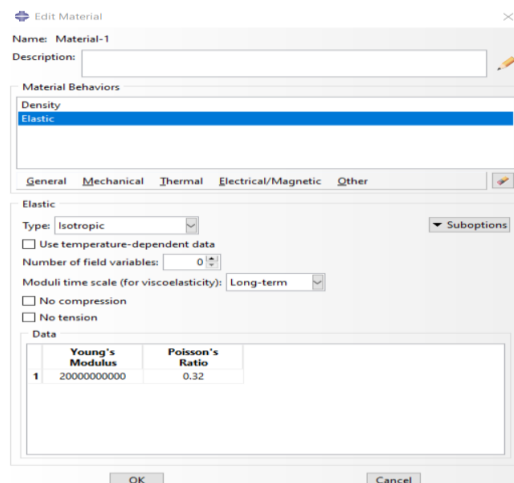


Figure 18: Property data Shown for Young's Modulus and Poisson's Ratio

Two forces are exerted on the solid surface. The first force is due to gravity, which acts on the entire solid element as depicted in figure 19. The second force is a pressure load, which acts on the upper portion of the circular surface in the flat area, as shown in figure 20.

In the manual calculation, we have already computed the force resulting from the concrete and live load. The force exerted by the jarrah hardwood, which does not contribute to the overall load on the plane, will be disregarded.

Here, $F_{concrete} = 7291.2 \text{ N}$

As the force is acting to the per square meter top flat surface.

Hence, uniform pressure load for Concrete = $7291.2/3.1 \times 0.05 = 47040 \text{ N/m}^2$

For live load, $F = 15500 \text{ N}$

Similarly, uniform pressure load for live load = $15500/3.1 \times 0.05 = 100000 \text{ N/m}^2$

Therefore, total uniform Pressure Load = $100000 + 47040 = 147040 \text{ N/m}^2$

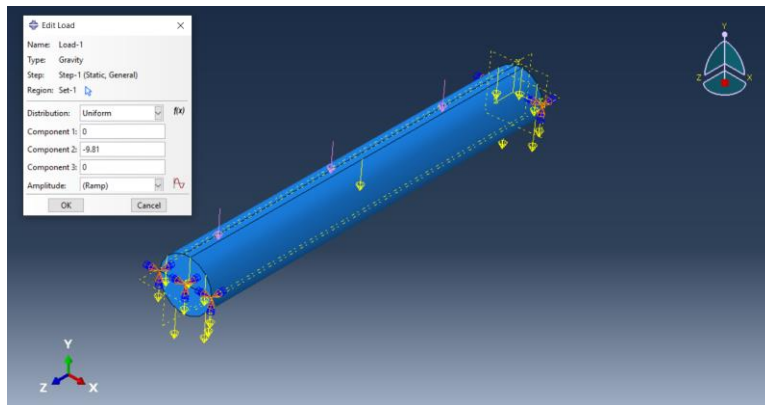


Figure 19: Yellow arrow denotes gravity load

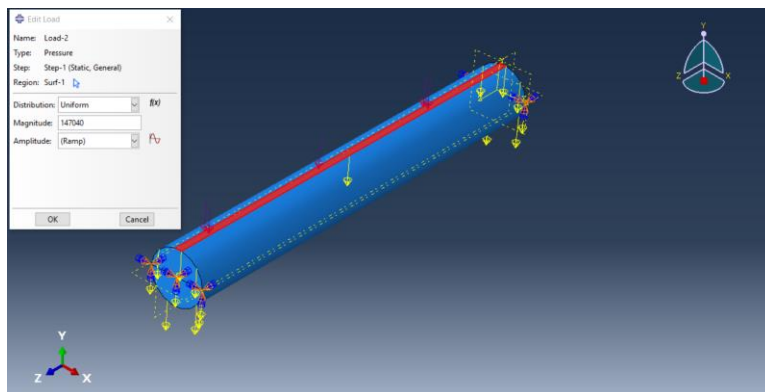


Figure 20: Red arrow denotes pressure load

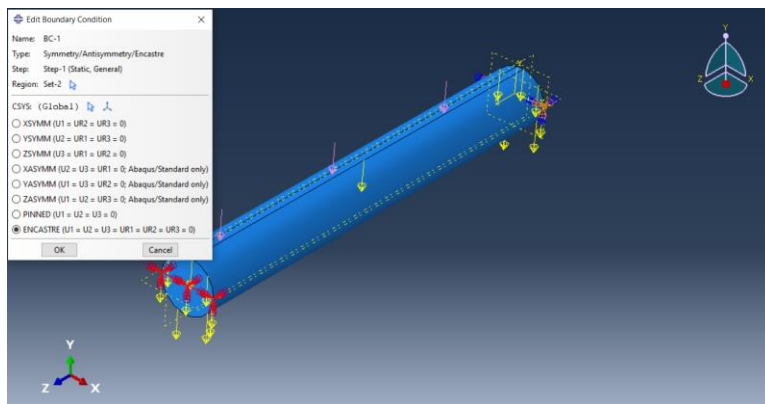


Figure 21: Boundary Condition Shown in one side.

Furthermore, boundary conditions are imposed on the two lines located at both ends of the cylinder by applying ENCASTRE condition, where these edges are held in a fixed position as shown in the figure 21. Two lines were generated by creating a datum plane. A plane was then constructed in the appropriate axis by specifying the coordinates. Finally, the partition face sketch tool was utilized to draw these two lines.

The seed part instance in the Mesh module is established by specifying a global size of 0.16, and the mesh control is set by the shape of the tet elements. Next, the solid element is discretized into a mesh.

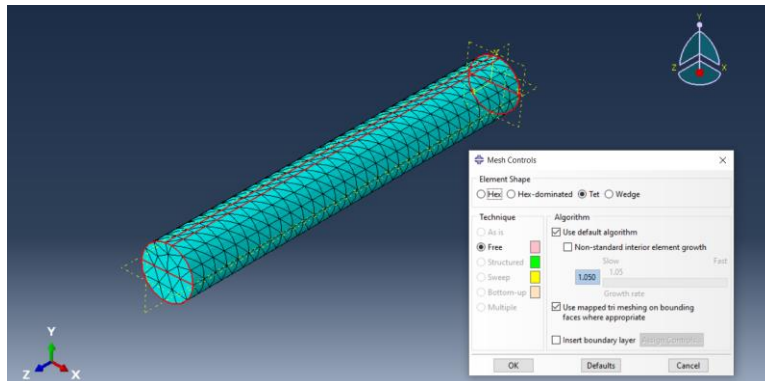


Figure 22: Assign Mesh Control

Moreover, a Job is generated and thereafter submitted for analysis. The stress and displacement outcomes are depicted in Figure 23 and 24.

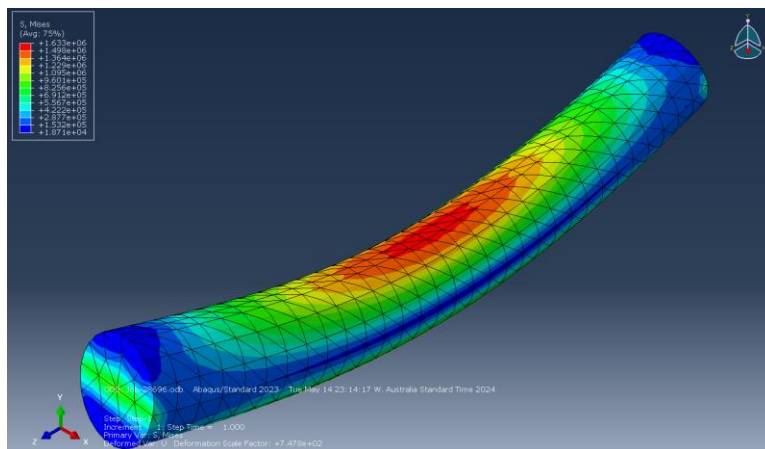


Figure 23: Magnitude of Stress

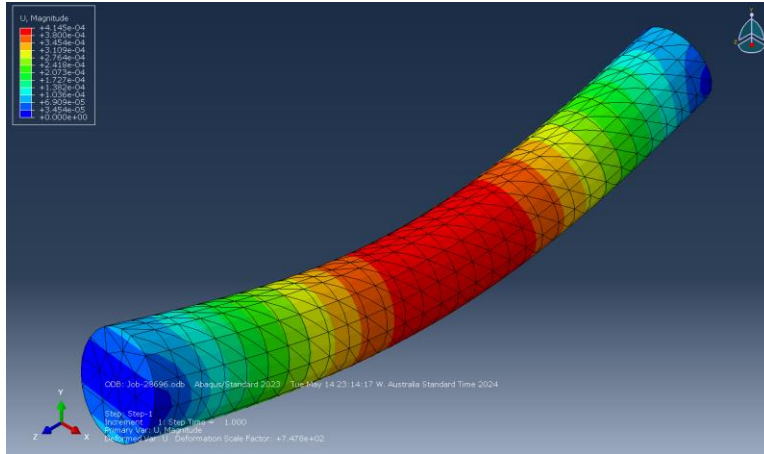


Figure 24: Magnitude of Displacement

Figure 24 demonstrates that the displacement value is 4.145×10^{-4} m, whereas in manual computation, the displacement value is 4.01×10^{-4} m. There is a little disparity in the displacement value between them.

$$\text{Error} = \frac{(4.145 - 4.01) \times 10^{-4}}{4.01 \times 10^{-4}} \times 100\% = 3.36\%$$

A minimal level of inaccuracy is detected in that particular model. I believe that utilizing rounded values can cause a little alteration in the uniform load pressure in ABAQUS Software, which in turn may affect the displacement value.

Furthermore, figure 24 demonstrates the displacement values at various points and highlights that the largest displacement and maximum stress occur at the midpoint. This is due to the application of boundary conditions at the two lines on opposite edges. The displacement gradually reduces as one moves towards the opposite edges.

Moreover, figure 23 demonstrates that the highest level of stress is exerted on the upper section of the solid surface. This stress gradually diminishes as it extends towards the lower portion of the surface, and also when it reaches the two opposing edges.

FEA Setup in Abaqus Software for l_2 with 3D Solid Element

Similarly, the displacement value in ABAQUS Software is determined in a step-by-step manner, just as the prior one, l_2 .

Picking out a 3D solid deformable element is the first stage in making a part drawing. In the drawing, a 0.2-meter-radius circle is shown. The top surface of the circular has been flattened by drawing a line that measures 0.05 m. As seen in figure 25, this establishes the way for the surface to be subjected to a pressure load. Then, l_2 was used to apply solid extrusion.

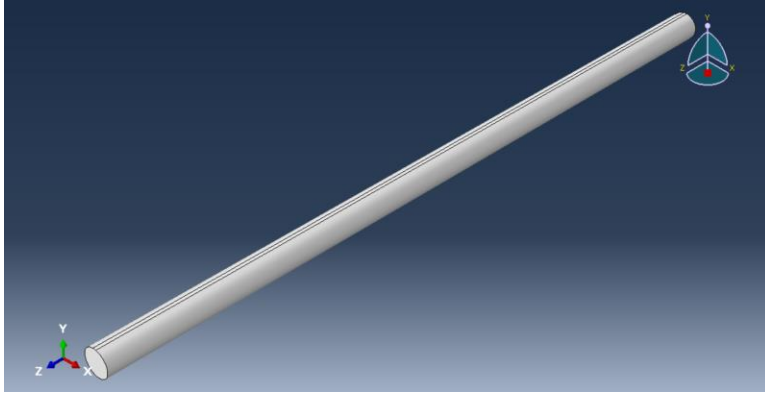


Figure 25: Part Drawing

In addition, the property tab calculates material properties such as mass density, Young's modulus, and Poisson's ratio using the same attributes as the previous tab. The section is established by choosing the complete surface. Likewise, step and assembly are executed in the same manner to obtain the structured analysis.

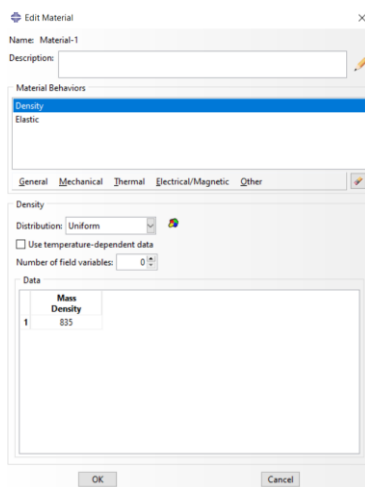


Figure 26: Created Material for Jarrah Hardwood

The solid surface is subjected to two forces. As seen in figure 27, the first force is the force of gravity acting on the whole solid element. Figure 28 shows that the second force is a pressure load acting on the top of the circular surface in the flat region.

The concrete and live load forces have previously been calculated in the manual calculation. The force applied by the jarrah hardwood, which is not included in the entire load carried by the flat surface, will be ignored.

Here, $F_{concrete} = 24108 \text{ N}$

As the force is acting to the per square meter top flat surface.

Hence, uniform pressure load for Concrete = $24108 / 10.25 * 0.05 = 47088 \text{ N/m}^2$

For live load, $F = 51250 \text{ N}$

Similarly, uniform pressure load for live load = $51250 / 10.25 \times 0.05 = 100000 \text{ N/m}^2$

Therefore, total uniform Pressure Load = $100000 + 47088 = 147088 \text{ N/m}^2$

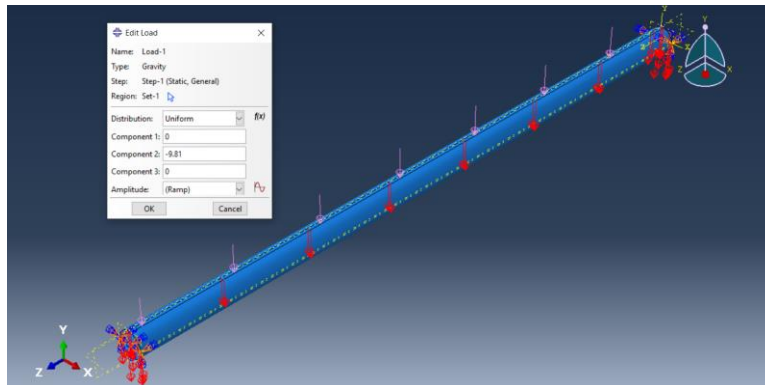


Figure 27: Red arrow denotes Gravity Load

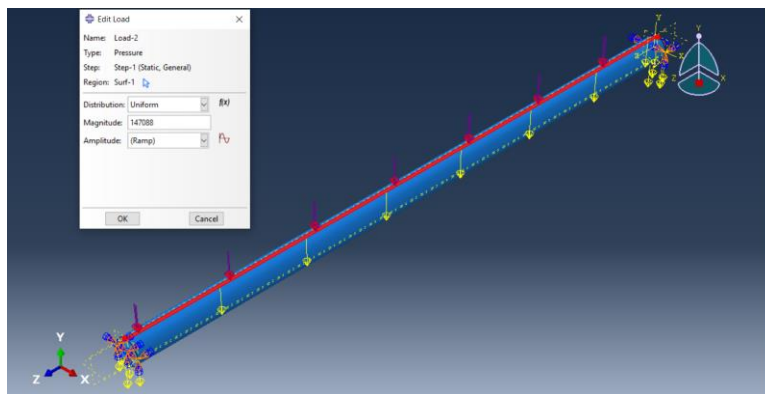


Figure 28: Red arrow denotes uniform pressure load

As seen in figure 21, ENCASTRE condition applies boundary conditions on the two lines at both ends of the cylinder. Creating a datum plane created two lines. Coordinates were used to build a plane on the appropriate axis. Finally, the partition face sketch tool drew these two lines.

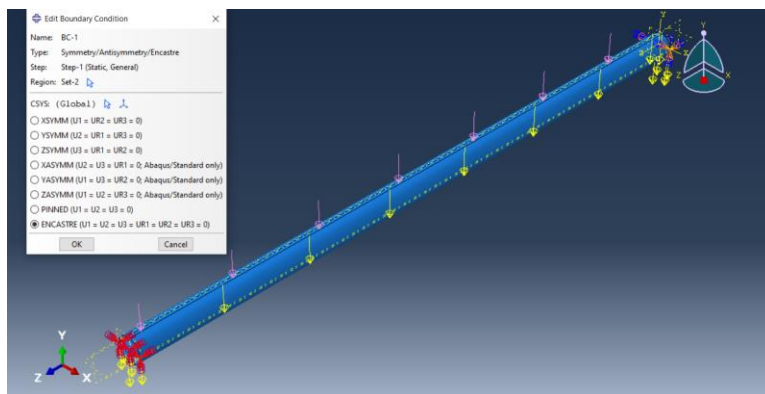


Figure 29: Boundary Condition Shown in one side.

The Mesh module's seed part instance is set up by giving it a global size of 0.16, and the form of the tet components sets the mesh control. The next step is to create a mesh by discretizing the solid component.

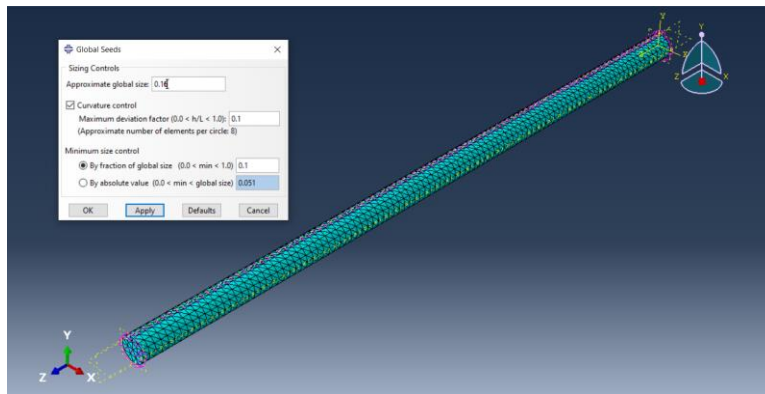


Figure 30: Seed Part Instance during mesh operation

In addition, a Job is created and then sent for evaluation. Figures 31 and 32 show the results of the stress and displacement.

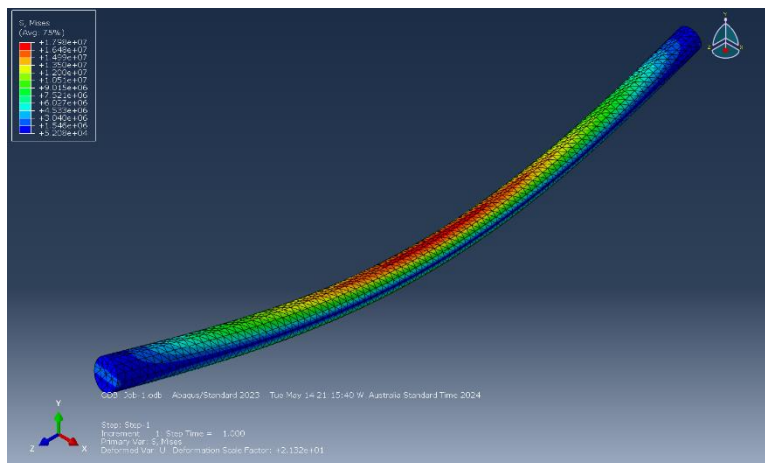


Figure 31: Magnitude of Stress

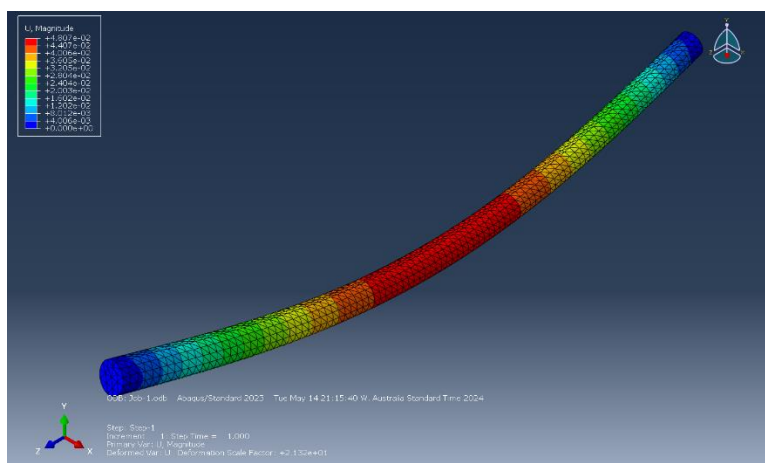


Figure 32: Magnitude of Displacement

The greatest displacement value is $4.805 \times 10^{-2} \text{m}$, or 0.048m , as shown in Figure 32. This outcome is in agreement with both the manual computation and the displacement value produced by the ABAQUS program. Therefore, there is not any reason for doubt that both the manual computation and the operation in the FEA software are entirely accurate.

In addition, Figure 32 depicts the displacement values at various locations and demonstrates that the maximum displacement and stress occur at the midpoint; when boundary conditions are applied to the two lines at opposite edges, the displacement gradually decreases as one approaches the other two opposite edges.

Furthermore, as illustrated in Figure 31, the maximal stress is exerted at the uppermost region of the solid surface. Furthermore, as the stress traverses two opposing edges, it decreases gradually at the lowermost region of the solid surface.

In addition, the magnitude of the stress is more in bigger bridge spans than it is in smaller bridge spans, and the magnitude of the displacement is also greater in comparison to the model that was used previously. Additionally, higher amounts of loads and displacements are exhibited by bridge spans that are larger in measurement.

Answer to the question no.2

The diagram depicts a longitudinal cross-section of a gravity dam, designed to contain water up to a specific height, denoted as "h". This issue focuses on analyzing the distribution of stress and displacement caused by water pressure in the dam. The ABAQUS Software is used to calculate the stress and displacement of the dam due to the water pressure.

Here, $h = 10 + 0.05 \cdot ID = 10 + 0.05 \cdot 105 = 15.25 \text{ m}$

In order to generate a part drawing, a 2D planar surface with a deformable shell is employed to obtain a 2D surface. The dam has been constructed according to geometric principles.

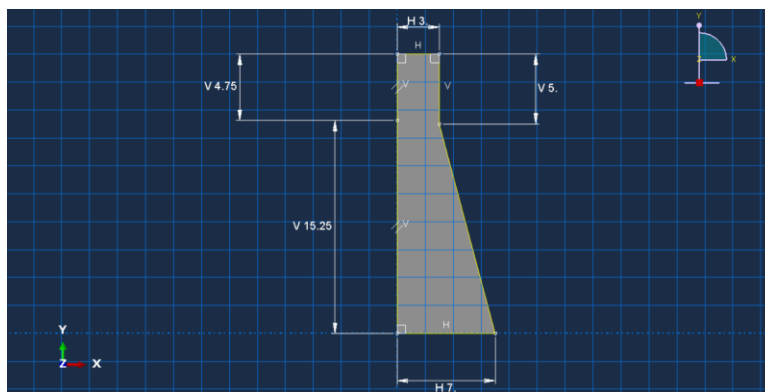


Figure 33: Geometry of dam

A appropriate material is defined in the property tab, with the following properties obtained from Google: mass density of 2400 kg/m^3 , Young's Modulus of 40 GPa , and a Poisson Ratio of 0.15 for Concrete. This material is then allocated to the selected wire surface to build and assign the section. Next, the assembly

operation is performed by choosing the component and creating a separate mesh for it. Next, a procedure is implemented to ensure the logical operation and accuracy.

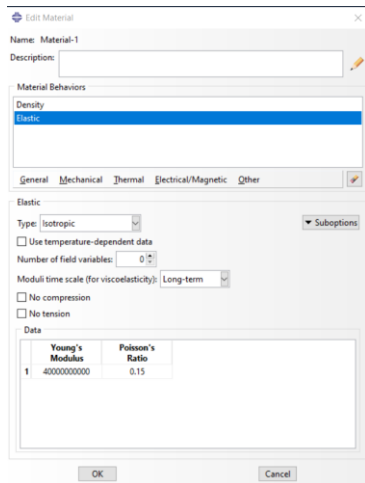


Figure 34: Property Selection and inserted data for Young's modulus and Poisson's Ratio

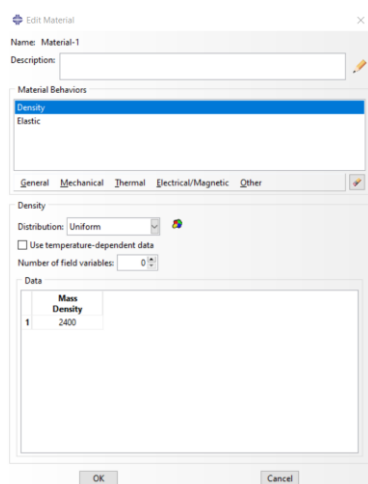


Figure 35: Data inserted for Mass Density in the property tab

Furthermore, 2 loads are acting there, one comes from the gravity load, which is acting the dam surface as shown in the figure 36, another one comes from the water pressure into the dam as shown in the figure 37.

Water pressure, $P = \rho gh = 1000 \cdot 9.81(15.25 - Y)$

The equation for Water pressure is made in Analytical field in the field tab as shown in the figure 38. Then created a load by selecting analytical field and magnitude is equal to 1, as shown in the Figure 37. In addition, boundary conditions are applied to the bottom lines of the dam, which is fixed in the ground, by using the ENCASTRE condition, as depicted in figure 39.

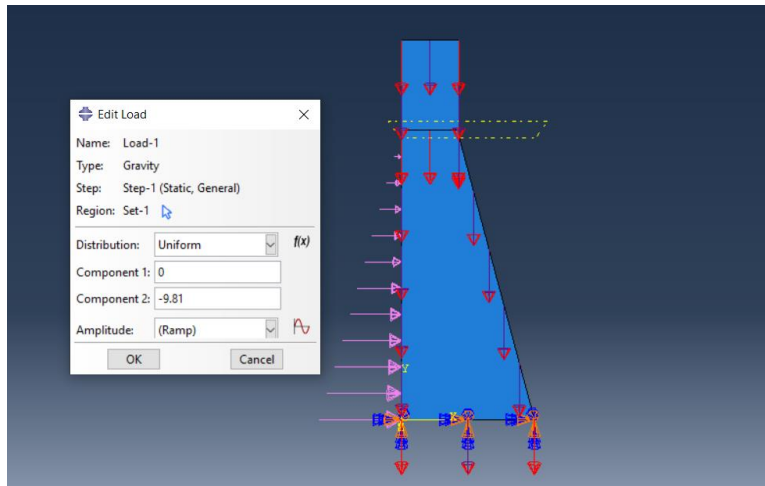


Figure 36: Red arrow denotes gravity load

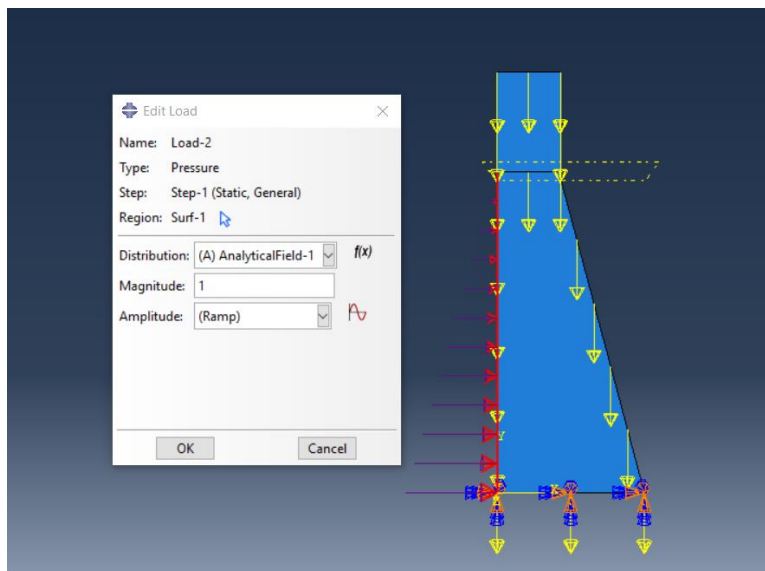


Figure 37: Red arrow denotes the water pressure load

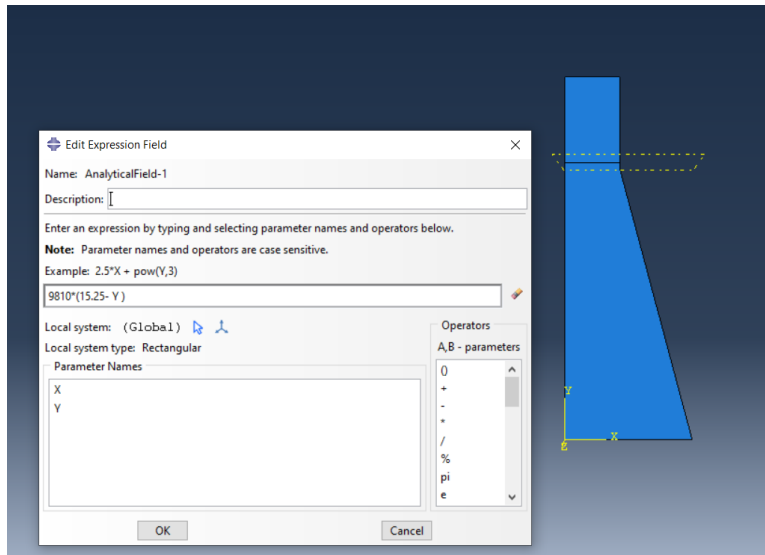


Figure 38: Equation for water pressure load

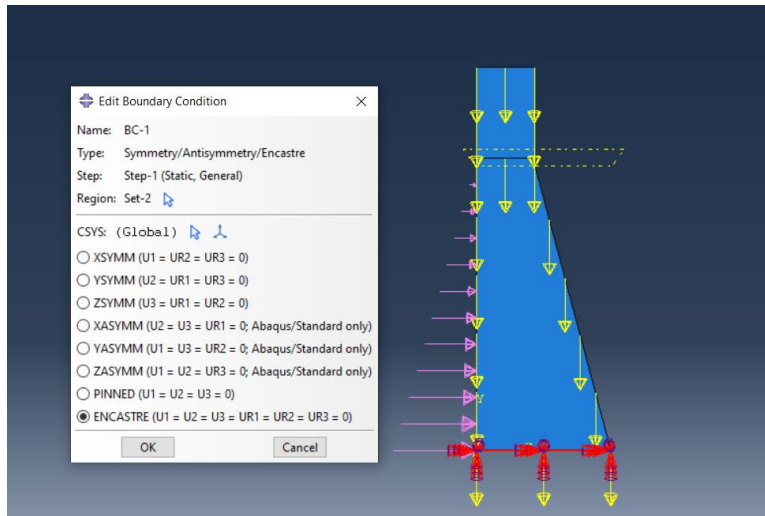


Figure 39: Red arrow denotes the line, where boundary conditions are applied.

The mesh control in the mesh module is established using a structured technique. The number of required seeds for a specific region determines the local seed selection. Subsequently, the element is assigned by encompassing the entire region. Next, the entire surface is meshed.

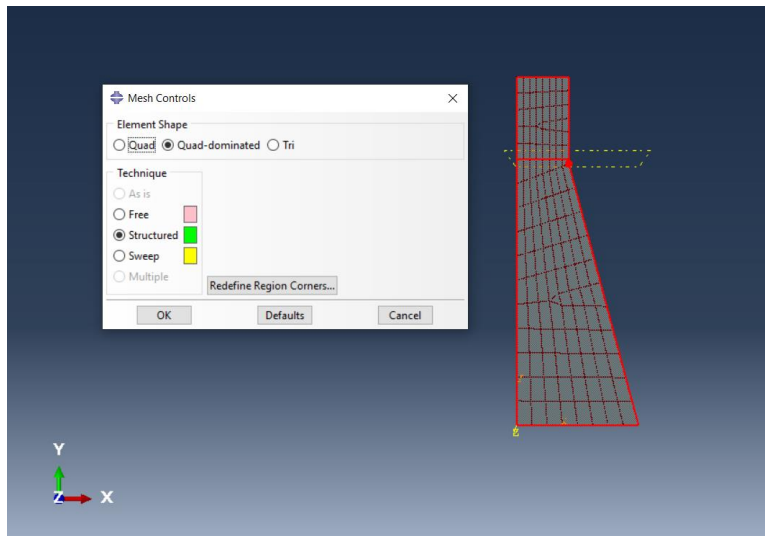


Figure 40: Mesh Control

In addition to this, a Job is created and then provided for evaluation after it has been created. Both Figure 41 and Figure 42 illustrate the results of the stress and displacement of the material.

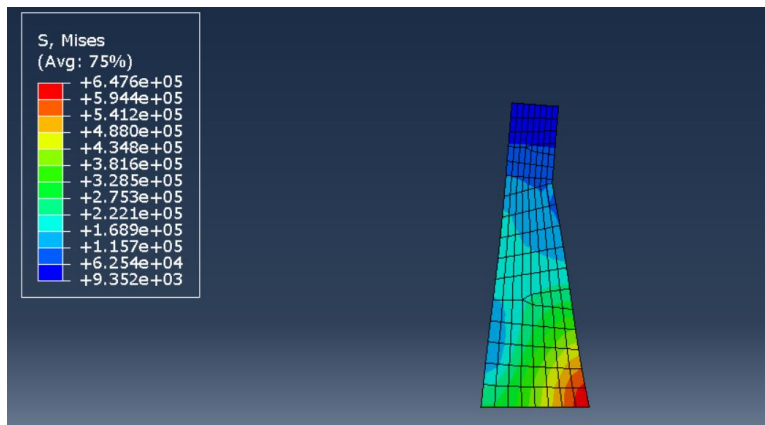


Figure 41: Magnitude of Stress

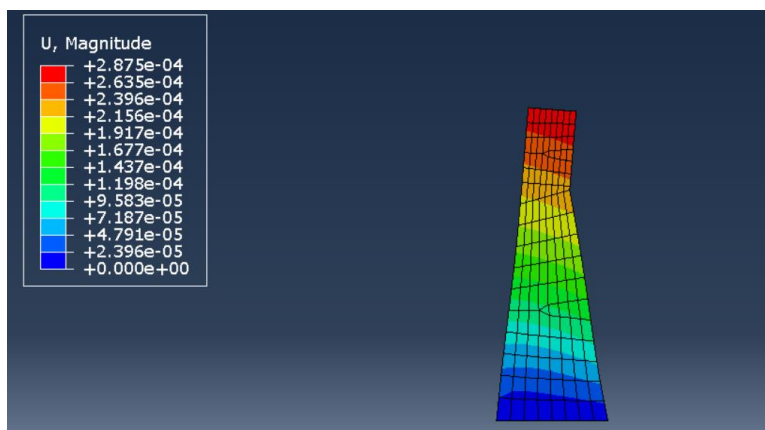


Figure 42: Magnitude of displacement

Figure 41 illustrates that the maximum stress on the bottom back of the dam is 6.476×10^5 , while the minimum stress is 9.352. As we move higher on the dam, the stress gradually decreases until it reaches the minimum stress at the top surface, which is not reached by the water level. This indicates that there is a lot of water pressure at depth, which raises stress levels.

However, figure 42 demonstrates that the greatest displacement is $2.875 \times 10^{-4} \text{m}$. This displacement is located at a higher position in the upper portion of the dam and gradually diminishes as we move down on the dam. A boundary condition is imposed at the base of the dam, resulting in lesser displacement in the lower section and a gradual rise as it approaches the upper section.

Reference

Alambra, K.& Swanson, N. (2024). *Omni Calculator*. Beam Deflection Calculator.

<https://www.omnicalculator.com/construction/beam-deflection>

BYJU'S. Poisson's Ratio.

<https://byjus.com/physics/poissons-ratio/>

Vasavan, Manilal (2017). *Linkedin*. Elastic Modulus of Concrete.

<https://www.linkedin.com/pulse/elastic-modulus-concrete-manilal-vasavan/>

