



FINITE ELEMENT ANALYSIS OF THREE DIFFERENT SOLID MECHANICS PROBLEM

JALAJ GUPTA

(19310R002)

Table of Contents

| | |
|---|----|
| 1. INTRODUCTION..... | |
| 2. PROBLEM-1..... | 3 |
| 2.1. Published Solution..... | 3 |
| 2.2. FEA Simulation..... | 4 |
| 2.2.1. Geometry..... | 4 |
| 2.2.2. Material Properties..... | 4 |
| 2.2.3. Mesh Convergence Study..... | 4 |
| 2.2.4. Interaction..... | 6 |
| 2.2.5. Load and Boundary Conditions..... | 6 |
| 2.2.6. Results and Discussion..... | 7 |
| 3. PROBLEM-2..... | 8 |
| 3.1. Analytical Solution..... | 8 |
| 3.2. FEA Simulation..... | 8 |
| 3.2.1. Geometry..... | 8 |
| 3.2.2. Material Properties..... | 9 |
| 3.2.3. Mesh Convergence Study..... | 9 |
| 3.2.4. Interaction and Predefined Fields..... | 9 |
| 3.2.5. Results and Discussion..... | 10 |
| 4. PROBLEM-3..... | 11 |
| 4.1. Analytical Solution..... | 11 |
| 4.2. FEA Simulation..... | 12 |
| 4.2.1. Geometry..... | 12 |
| 4.2.2. Material Properties..... | 13 |
| 4.2.3. Mesh Convergence Study..... | 13 |
| 4.2.4. Load and Boundary Conditions..... | 14 |
| 4.2.5. Results and Discussion..... | 14 |
| 5. CONCLUSION..... | 15 |
| 6. REFERENCES..... | 15 |

ABSTRACT

This report shows the comparison between analytical and computational results for three different Finite Element Analysis (FEA) Problems. Out of these three problems, two are solved analytically by using Basics of Solid mechanics and one has results that are already published in a reputed journal.

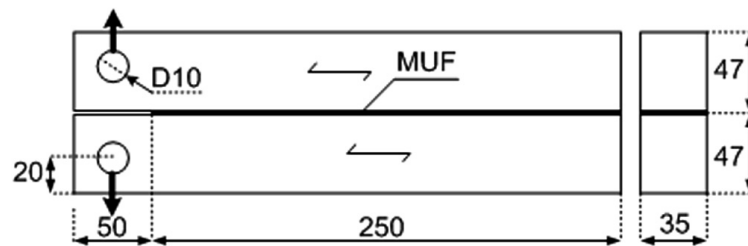
Finite element simulation is done using Abaqus Standard 6.13-1 version. First problem is to simulate Mode I crack failure of the glued timber by cohesive behavior interaction. A mesh convergence study is performed to validate results. There is very less difference between simulation results and published ones. The reason for the variation in results is explained in results and discussion part of the document. Second problem is to simulate two-dimensional truss subjected to loading. Abaqus automatically considers hinged joints between trusses until specified. The stresses are calculated analytically that matches with the simulated results. Third problem is transient behavior of sphere subjected to thermal boundary conditions. Lumped capacity analysis is valid as Biot Number is less than 0.1.

1. INTRODUCTION

Finite Element Analysis (FEA) is a computer-based technique that calculates how components are likely to perform in real-life applications. This advanced technique can determine how components may be impacted by exposure to extremes in temperature and pressures, particular stresses and aggressive chemical environments. Engineers use it to reduce the number of physical prototypes and experiments and optimize components in their design phase to develop better products, faster. There are many FEA packages such as NX NASTRAN, ANSYS, ABAQUS, PATRAN, SOLIDWORKS (in build FEA package) and many more are available. For this report. ABAQUS 6.13-1 version is used for FEA simulation. While doing FEA simulation it is very important to verify it using analytical solutions if possible. Finite element analysis (FEA) is now an integral part of most structural analyses. In fact, we not only use FEA in daily analysis, we also use FEA to optimize our structural designs. It saves times, allows for quick variation in designs and is often used to bring out a lean product.

2. Problem-1

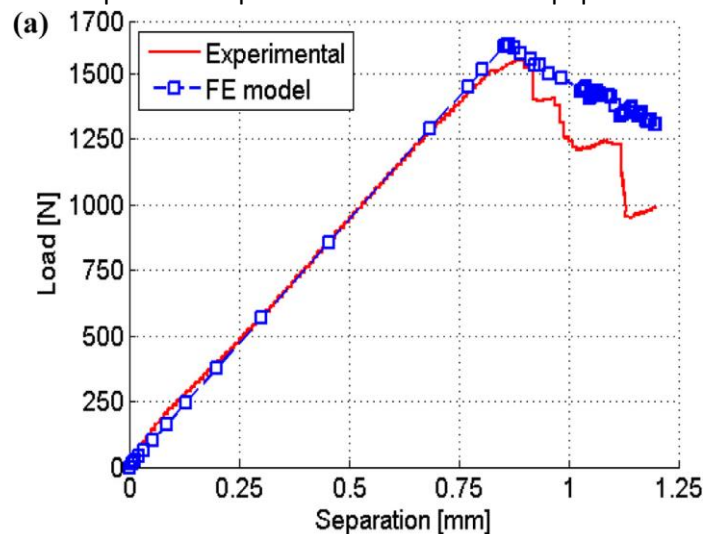
This problem has been taken from a research paper named "[Experimental and numerical analyses of the structural response of adhesively reconstituted beech timber beams](#)" by Van-Dang Tran, Marc Oudjene and Pierre-Jean Méausoone. The dimensions of the standard specimen are as shown below in the Figure. We will determine Load vs Displacement curve and verify it with published plot.



Schematic representations of the modified DCB used in fracture mode I.

2.1. Published Result

Load vs Displacement plot that is presented in the research paper is shown below.

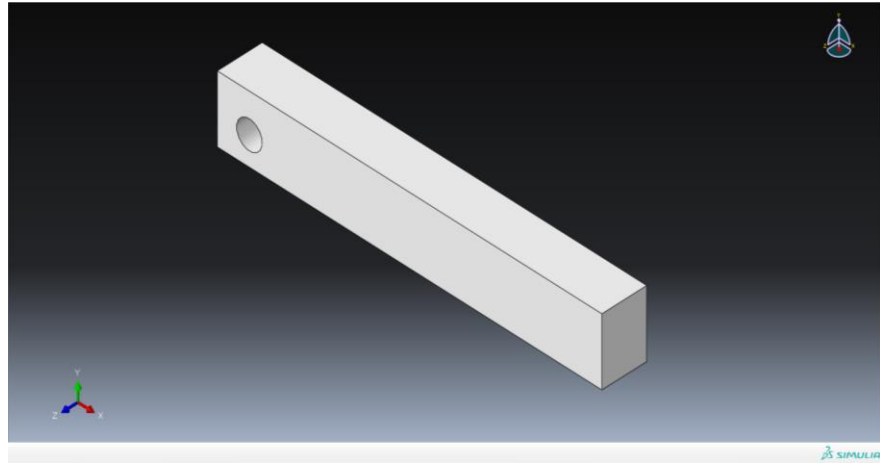


Comparison between mean experimental and numerical load–displacement curves for mode I

2.2. FEA Simulation

2.2.1. Geometry

A solid simplified Double Cantilever Beam (DCB) is modelled using one half of the specimen as the DCB is symmetric about an x-z plane passing through upper surface of specimen.



2.2.2. Material Properties

Elastic properties of a beech timber (anisotropic) is shown below.

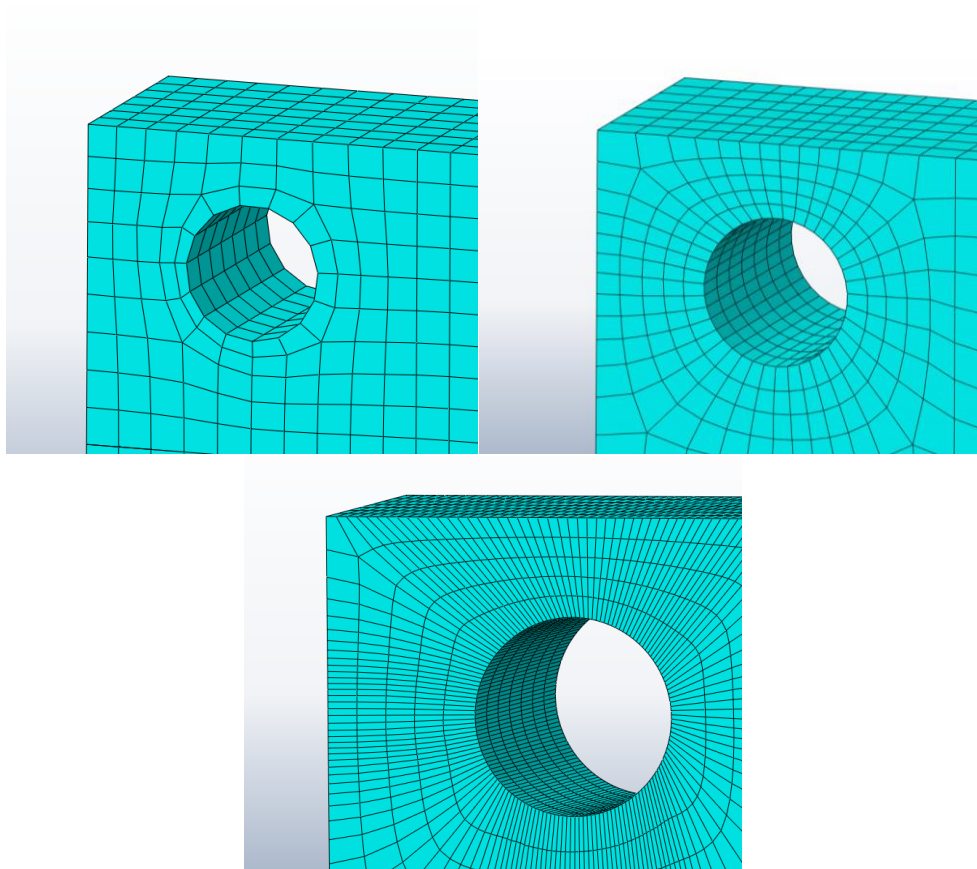
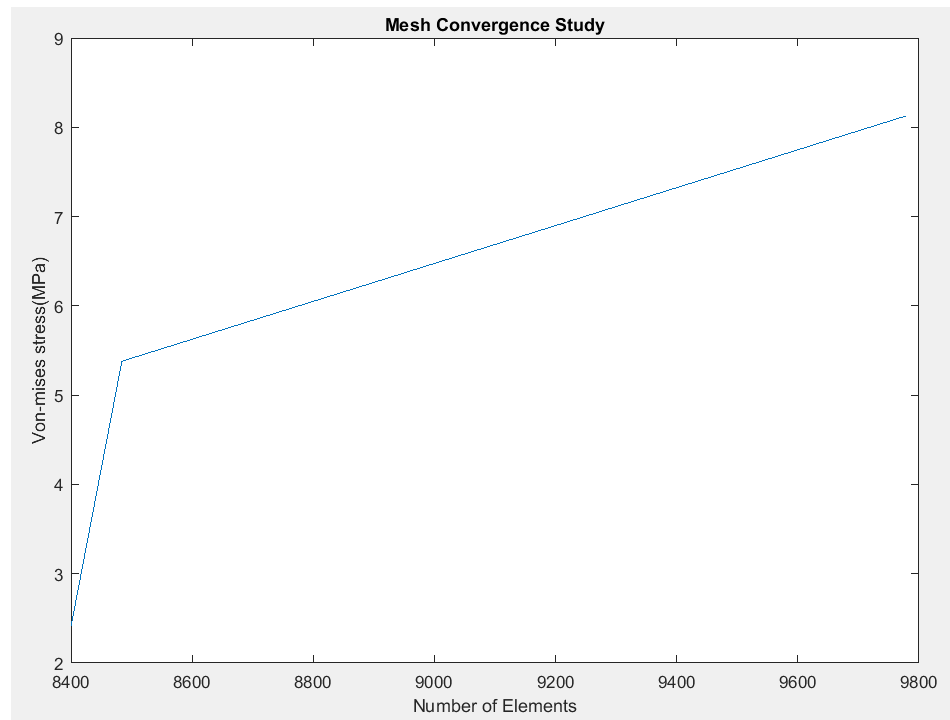
| Elastic properties | Values |
|--------------------|--------|
| E_1 (MPa) | 14788 |
| E_2 (MPa) | 1848 |
| E_3 (MPa) | 1087 |
| ν_{12} | 0.39 |
| ν_{13} | 0.46 |
| ν_{23} | 0.67 |
| G_{12} (MPa) | 1220 |
| G_{13} (MPa) | 971 |
| G_{23} (MPa) | 366 |

Cohesive Properties (i.e. Stiffness of the Traction-Separation curve) is shown below.

| K_{nn} (N/mm) | K_{ss} (N/mm) | K_{tt} (N/mm) |
|-----------------|-----------------|-----------------|
| 10 | 30 | 30 |

2.2.3. Mesh Convergence Study

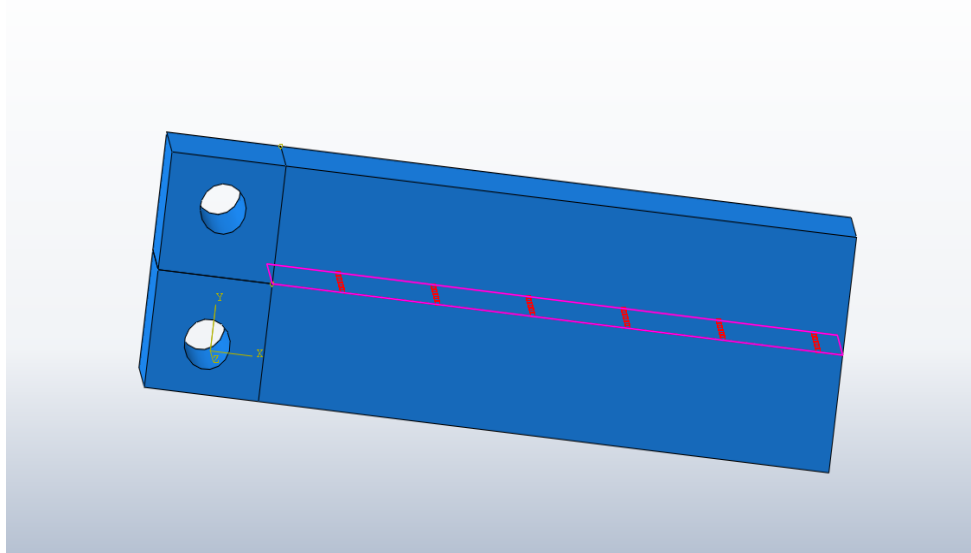
Entire body is meshed using C3D8R: An 8-node linear brick, reduced integration, hourglass control. In order to validate the results calculated in Abaqus, a mesh convergence study is performed. Total three cases are analysed with different element count. First a model with coarse mesh having total 8400 elements is analysed. Further mesh is refined having 8484 and 17899 elements. For each model, Von-mises stress at lower part of upper hole surface is plotted against number of elements in the model. The plot is shown below.



Local mesh convergence is performed in order to obtain the Mesh converged solution.

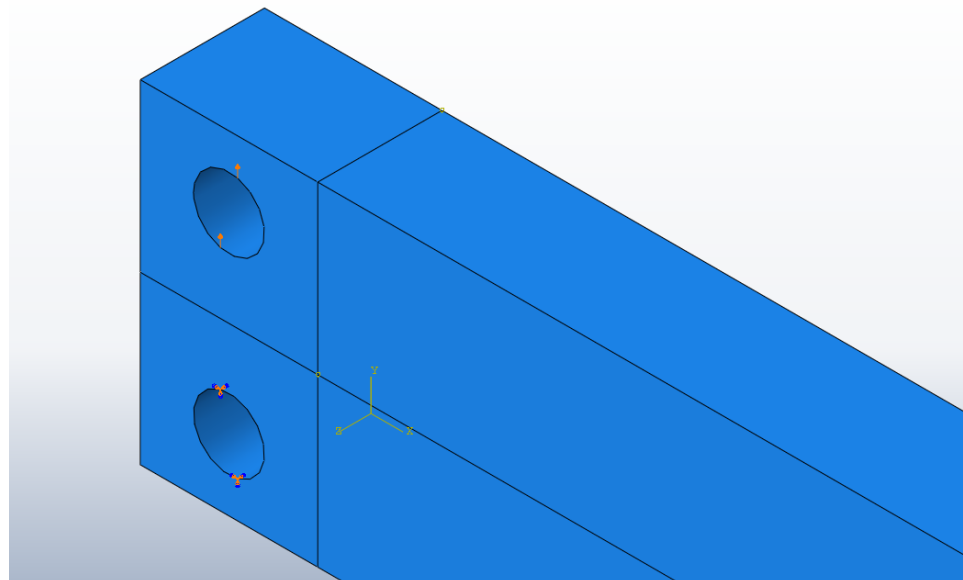
2.2.4. Interaction

A Surface-to-surface contact is applied between two halves of the specimen (representing glued timber). A contact property is defined to account for traction separation behaviour by specifying stiffness coefficients. For initiation of the damage, Maximum nominal stress criterion has been considered.



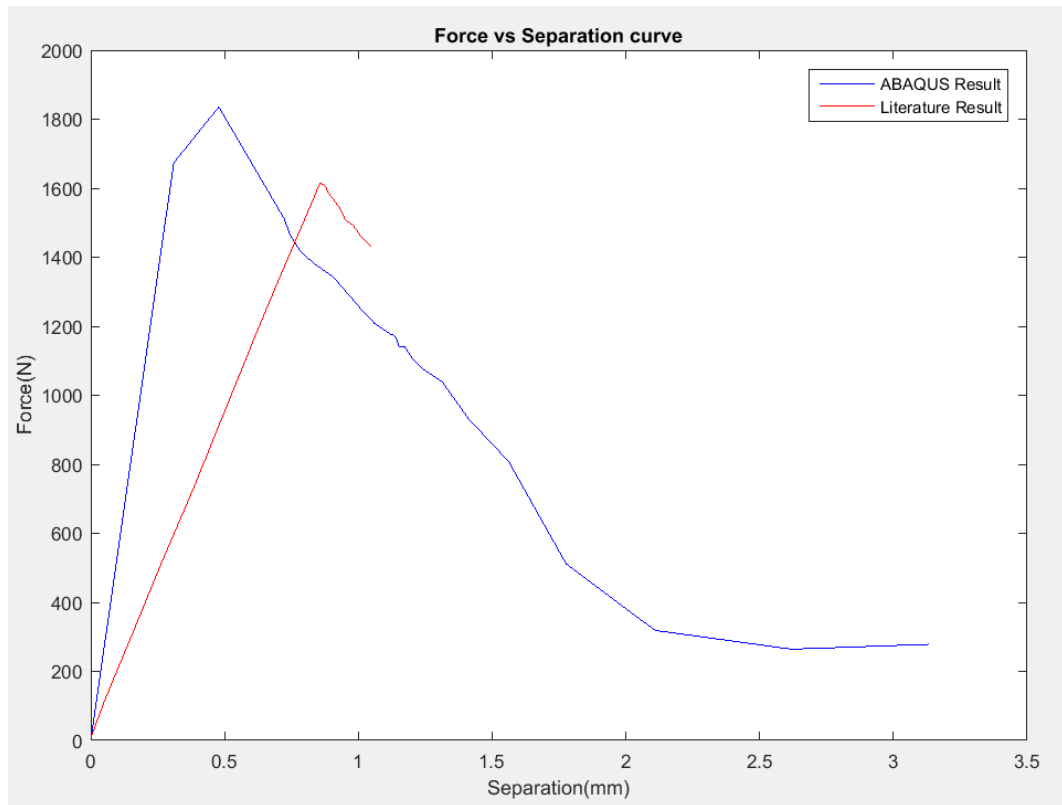
2.2.5. Load and Boundary Conditions

Two boundary conditions are applied to the specimen. First, the lower hole is fixed and second, the upper hole is given a displacement of 3mm in upward direction as shown below.



2.2.6. Results and discussion

Elastic material model has been assumed to describe the behaviour of the beech timber, while the behaviour of glue-lines has been modelled with the help of the Cohesive Zone Model (CZM). While simulating this problem, Elastic properties are only taken into consideration. This can be the possible reason for difference between Abaqus and Literature results.



3. Problem-2

A small copper ball of 5 mm diameter at 500 K is dropped into an oil bath whose temperature is 300 K. The thermal conductivity of copper is 400 W/mK, its density 9000 kg/m³ and its specific heat 400 J/kgK. If the heat transfer coefficient is 250 W/m²K. We have to determine temperature of the sphere after 30 seconds (Lumped capacity analysis is assumed to be valid).

3.1. Analytical Solution

Temperature after t seconds is given by
$$\frac{(T - T_{amb})}{(T_o - T_{amb})} = \exp \left[\frac{-hAt}{(\rho \cdot V \cdot C)} \right] \dots (1)$$

Where, T- temperature after t seconds

T_o- initial temperature

T_{amb}- Ambient temperature

h- Heat transfer coefficient

A- Cross sectional area

t - Time

ρ - Density

V- Volume

C- Specific heat

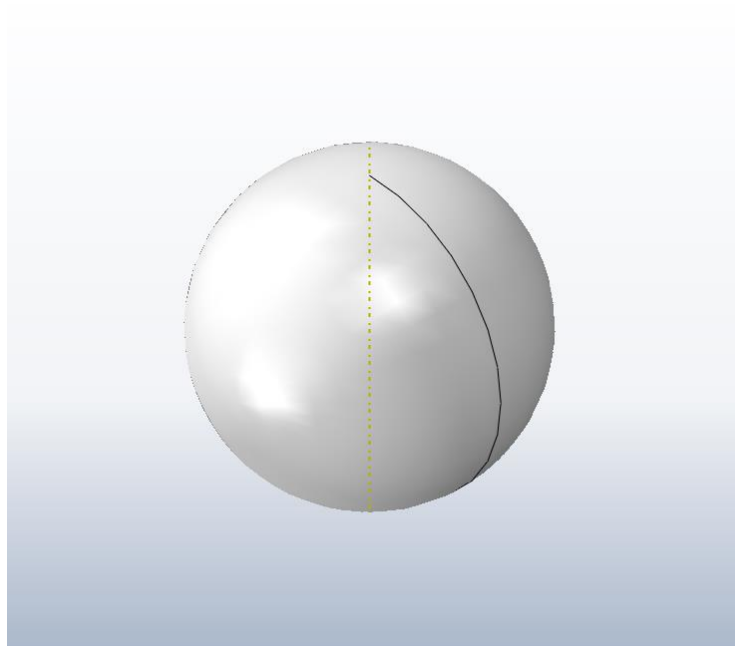
Equation (1) is valid as Biot No. = $\frac{(400 \cdot 0.005)}{(6 \cdot 400)} \ll 0.1$. Thus, Lumped capacity analysis can be applied.

On placing the values given in the problem statement and calculating for temperature after 30 seconds, T=316.41K.

3.2. FEA Simulation

3.2.1. Geometry

A sphere is modelled as shown in the figure. A semi-circle is drawn having radius 0.0025m which is revolved around the axis to obtain the required part.

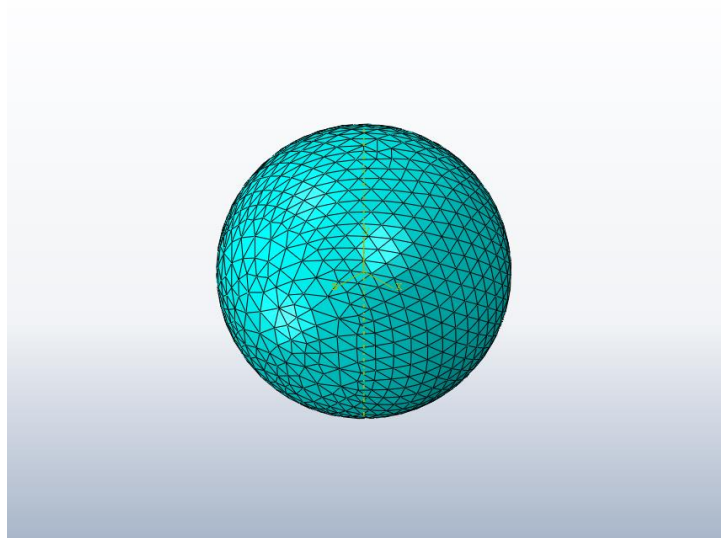


3.2.2. Material Properties

| Part | Density (kg/m ³) | Specific heat (J/kgK) | Thermal conductivity(W/mK) |
|--------|---------------------------------|--------------------------|-------------------------------|
| Sphere | 9000 | 400 | 400 |

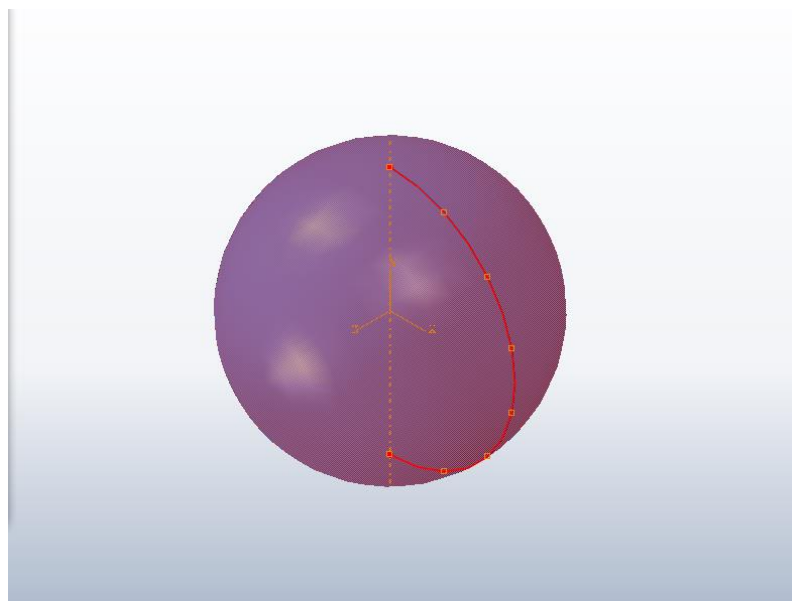
3.2.3. Mesh Convergence Study

Entire body is meshed using DC3D4: A 4-node linear heat transfer tetrahedron. There is no mesh convergence study required as temperature (which is our concern) is constant throughout the body (as lumped capacity analysis is valid).

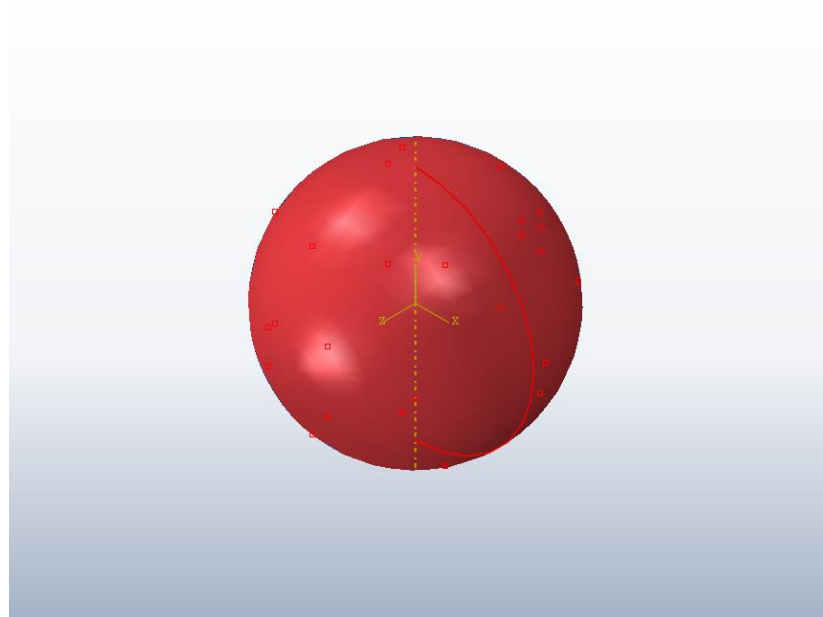


3.2.4. Interaction and Predefined Fields

Surface film condition has been provided to the sphere with film coefficient $250\text{W/m}^2\text{K}$. A uniform sink temperature of 300K is prescribed as per the given problem. A predefined temperature field of 500K is given to the sphere constant throughout the body.



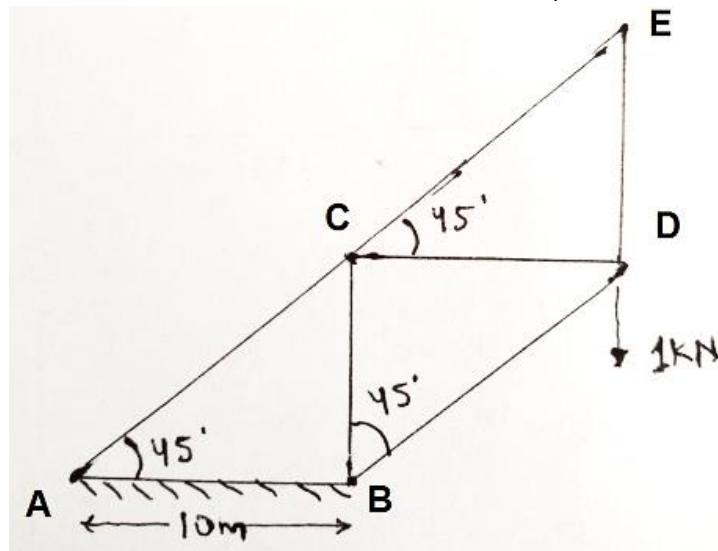
Predefined field



The temperature distribution in the body is shown in the figure below. There is no temperature gradient setup within the body as lumped mass analysis is valid. So, the results seems to be as expected. The temperature value after 30 seconds obtained from FEA simulation is almost equal to analytical value calculated in Section 3.1.

4. Problem-3

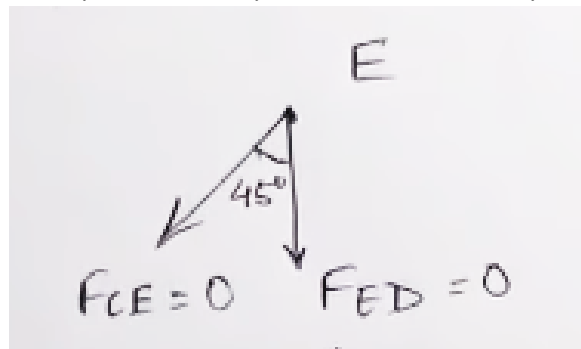
To find the stresses in the members of the truss subjected to load as shown below. (All the members have same cross sectional area with radius = 0.01m).



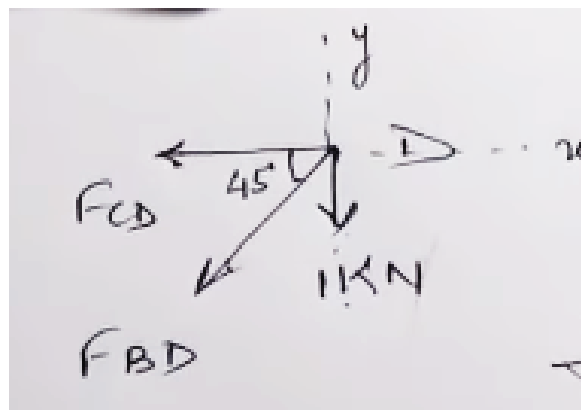
4.1. Analytical Solution

The solution for the given problem can be obtained as follows:

At point E, F_{ED} and F_{CE} is equal to zero as per balance of force at point E.



At point D, there is 1kN load acting in downward direction and other forces assumed as shown below.



Applying Force balance in Y-direction, we get

$$-1 - F_{BD} \sin(45) = 0$$

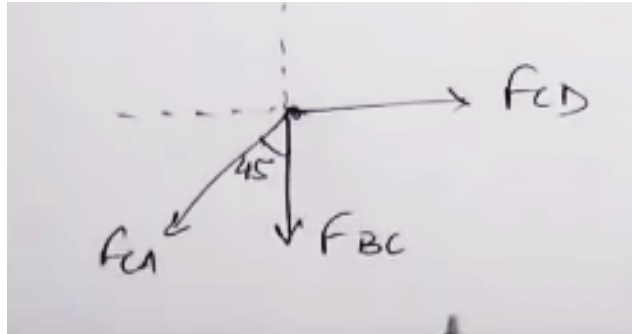
$$\text{Or } F_{BD} = -1.414 \text{ kN} \dots (1)$$

Applying Force balance in X-direction, we get

$$-F_{CD} - F_{BD} \cos(45) = 0$$

$$\text{Using (1), } F_{CD} = 1 \text{ kN} \dots (2)$$

At point C,



Applying Force balance in X-direction, we get

$$F_{CD} - F_{CA} \sin(45) = 0$$

$$\text{Or } F_{CA} = 1.414 \text{ kN}$$

Applying Force balance in Y-direction, we get

$$-F_{BC} - F_{CA} \cos(45) = 0$$

$$\text{Or } F_{BC} = -1 \text{ kN}$$

So, we have obtained forces in all the members. Now, stresses can be calculated by dividing cross sectional area.

$$A = \pi(0.01 \times 0.01) = 3.14 \times 10^{-4} \text{ m}^2$$

σ_{CE} and σ_{ED} is zero as loads in these members is zero.

$$\sigma_{BD} = -1.414 \times 1000 / 3.14 \times 10^{-4} = -4.5032 \text{ MPa}$$

$$\sigma_{CD} = 1 \times 1000 / 3.14 \times 10^{-4} = 3.184 \text{ MPa}$$

$$\text{Similarly, } \sigma_{CA} = 4.5032 \text{ MPa}$$

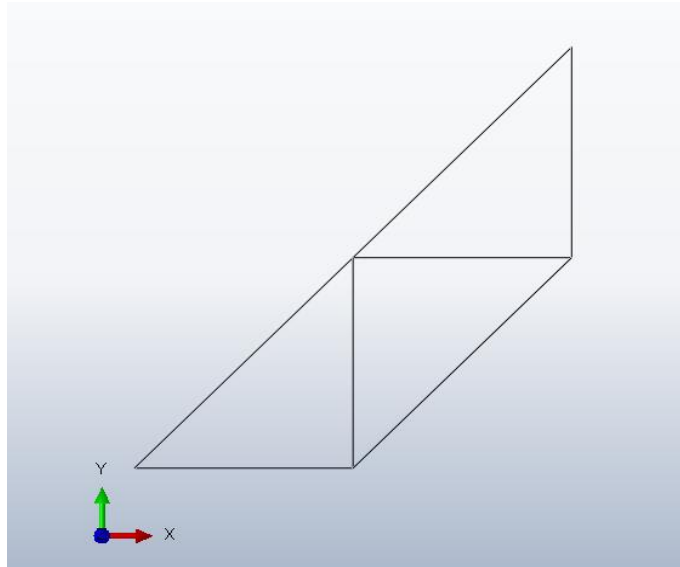
$$\sigma_{BC} = -3.184 \text{ MPa}$$

Here, negative sign implies that the stress in that member is compressive.

4.2. FEA Simulation

4.2.1. Geometry

A simplified geometry of given 2-D truss is modelled as shown below.



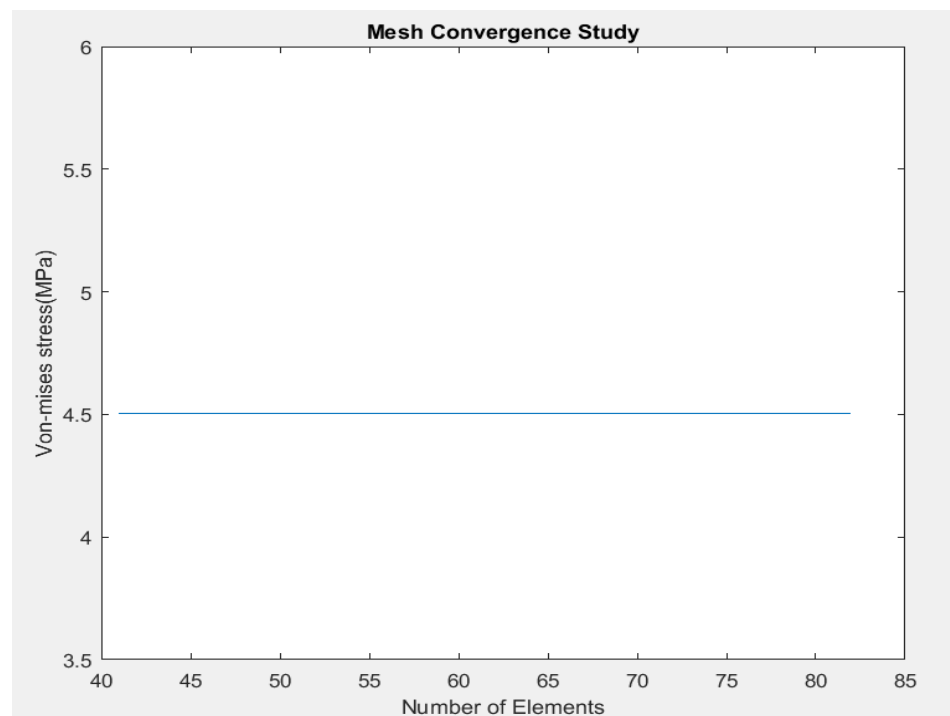
4.2.2. Material Properties

All the truss members have same material properties with material being SS_304 as shown below.

| Part | Density (kg/m ³) | Young's Modulus(GPa) | Poisson's ratio |
|-------|---------------------------------|-------------------------|-----------------|
| Truss | 8000 | 193 | 0.29 |

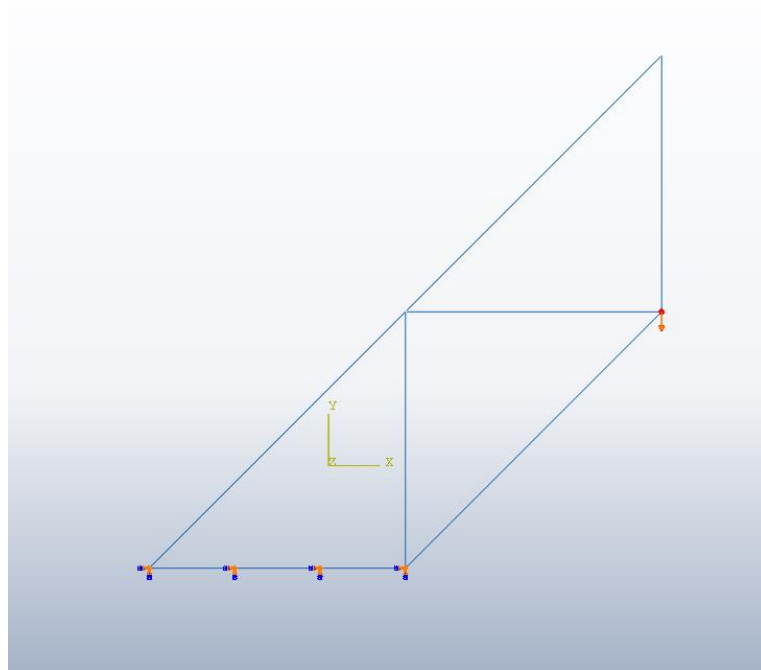
4.2.3. Mesh Convergence Study

Entire body is meshed using T2D2: A 2-node linear 2-D truss. In order to validate the results calculated in Abaqus, a mesh convergence study is performed. Total three cases are analysed with different element count. First a model with coarse mesh having total 41 elements is analysed. Further mesh is refined with 58 and 82 elements. Exactly same results are obtained in each of the cases.



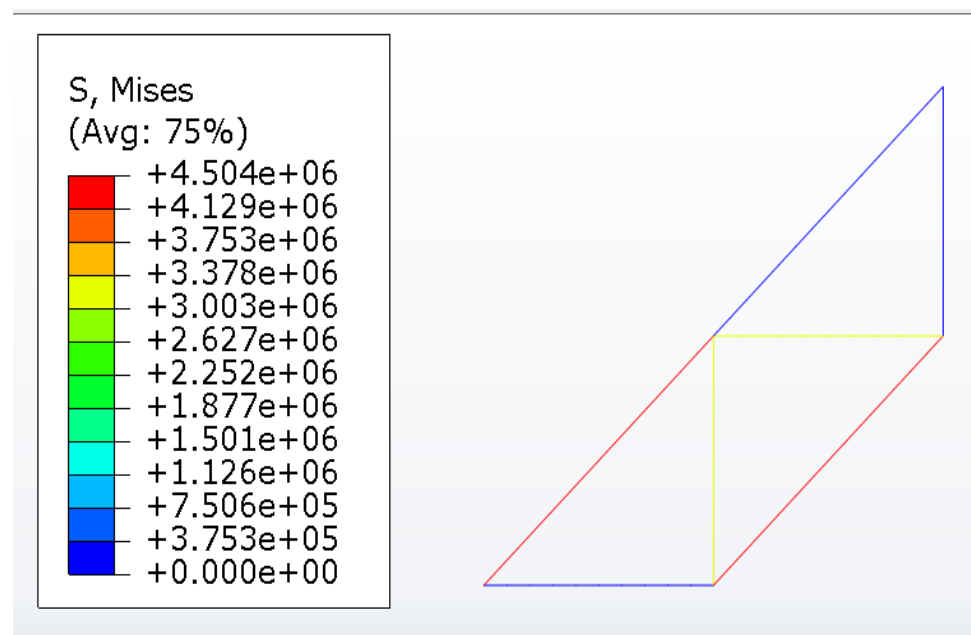
4.2.4. Load and Boundary Condition

A load of 1kN is applied in negative Y-axis direction at the node D (highlighted in orange color). The member AB (bottommost horizontal member) is fixed as shown below.



4.2.5. Results and Discussion

Stresses in various members of the truss is shown in the figure below. The stress values obtained from FEA simulation are almost equal to the analytical value. For e.g. Stress in member BD is 4.5032MPa which is in conjunction with value 4.504e+06Pa. Refer Section 4.1 for other stress values.



5. Conclusion

Two problems are solved using analytical and FEA simulation whereas one has published results. Analytical/Published and computational results are compared and reasons of variation in result is highlighted. It is concluded that there are many reasons for variation in FEA results and analytical results. When we solve solid mechanics problem we consider many assumptions, while finite element analysis considers real life conditions. Mesh density at critical location is also responsible for variation in results. So, it is very important to perform mesh convergence study whenever using FEA simulation if possible.

6. References

- [1] Experimental and numerical analyses of the structural response of adhesively reconstituted beech timber beams.
https://www.researchgate.net/publication/266148615_Experimental_and_numerical_analyses_of_the_structural_response_of_adhesively_reconstituted_beech_timber_beams
- [2] Properties of materials SS-304.
<https://www.azom.com/article.aspx?ArticleID=965>
<http://asm.matweb.com/search/SpecificMaterial.asp?bassnum=mq304a>
- [3] SIMULIA How-to Tutorial for Abaqus | Heat Transfer Analysis.
<https://www.youtube.com/watch?v=lanlaV03ZxE>
- [4] SIMULIA How-to Tutorial for Abaqus | Analysis of a 2D Truss.
<https://www.youtube.com/watch?v=u8yXeyFIWjY>