

Finite Element Method Using COMSOL Multiphysics©

Components and Devices Lab

Smart Systems

Winter semester 2019 – 2020

Dinesh Poluru Baskar (264374)

Contents

1. Introduction
2. Methodology
 - 2D Cantilever beam
 - 3D Cantilever beam
 - Silicon hot plate
3. Conclusion
4. Reference

Introduction :

In the past decade in the industrial manufacturing and also in the scientific research and engineering, simulation methods have become an important part of the development. Simulations of the physical behaviour of new products help to reduce costs in the manufacturing process in industry. It also helps in the understanding of complex relations in nearly all kind of scientific research, where analytical solutions of equations cannot be found. One of these simulation methods is very often used because of its basic ability to find approximatively solutions for problems which are described by partial differential equations. The finite element method (FEM) is a numerical method for the solution of partial differential equations. It is a widespread modern calculation method in engineering and is the standard tool for the bulk simulation. The method provides an approximate function of the exact solution to the differential equation, the accuracy can be improved by increasing the degrees of freedom and hence the amount of computation. The FEM is a basic numerical method for solving differential equations by approximation. Thus different physical problems from different disciplines such as structural mechanics, fluid dynamics, magnetic and electrical fields, acoustics and heat transfer can be calculated and solved. First, the computational domain is divided into an arbitrarily large number of elements. These elements are "finally" (finite) small. Dividing the area into a certain number of elements of finite size, it can be described with a finite number of parameters, which gave the name to the method: "finite element method". Within these elements approximation functions are defined. Substituting these trial functions into the differential equation to be solved, we get together with the initial, boundary and transition conditions a system of equations, which is usually solved numerically. The size of the system of equations to be solved depends largely on the number of finite elements. His solution is ultimately the numerical solution of the considered differential equation. Different commercial FEM software is available and commonly used: Two of these are ANSYS© and COMSOL. In other Software packages like Solid Works© a FEM tool is part of it. Also some non-commercial FEM programs as for example Abaqus are in use. In this practical course we want work with the software COMSOL Multiphysics©, which is together with ANSYS one of the mostly used in industry [1].

Methodology :

The task which was given to us is to create a 2D and 3D cantilever beam and run simulations and compare the values between them. This methodology is divided into two parts, Part 1 is going to discuss about the 2D and 3D cantilever beam and the results about those cantilever beams and in part 2 we will be discussing about the silicon hot plate. For creating the 2D cantilever I was given with 950/100/20 dimension.

A. Part 1 : 2D Cantilever Beam & 3D Cantilever Beam

In 2D cantilever beam we were asked to build a model of silicon cantilever beam with beam geometry (L/W/H) = (1000 μ m/100 μ m/20 μ m) with material silicon and boundary condition as one end of the beam is fixed. The force to be produced is $F = 10\text{mN}$ at the tip of the beam.

- I created the 2D cantilever beam using COMSOL Multiphysics and simulations for different elements and different mesh grids are run and compared by using excel sheet. The data which are shown below are the *normal triangular mesh* for 2D cantilever beam and the running time is also noted.

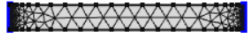
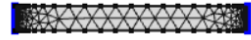



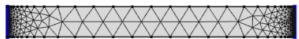
No. of elements	Mesh	Results	Time (s)	Deformation
10		Mesh – 222 domains, 64 boundary elements	1	9.3808
15		Mesh – 320 domains, 78 boundary elements	1	9.3816
25		Mesh – 538 domains, 104 boundary elements	1	9.3822
30		Mesh – 658 domains, 118 boundary elements	1	9.3824
40		Mesh – 874 domains, 140 boundary elements	1	9.3824
50		Mesh – 1128 domains, 164 boundary elements	1	9.3826

Table 1 : Shows mesh details according to no. of elements and also the deformation.

The above tables shows the different mesh elements and also their deformation data. If we draw a graph between the number of elements and the deformation we get :

The graph chart below shows the deformation with respect to number of elements for 2D cantilever beam. The above table shows the result of normal triangulated mesh which is available in COMSOL Multiphysics software. But COMSOL provide us with various option to change the way simulation works by changing the meshing type. There different types of mesh available which includes : *Automated free triangular mesh from extremely coarse to extremely fine and Manual mapped mesh*

These meshes are used for 3D cantilever beam too which are later will be discussed in this report.

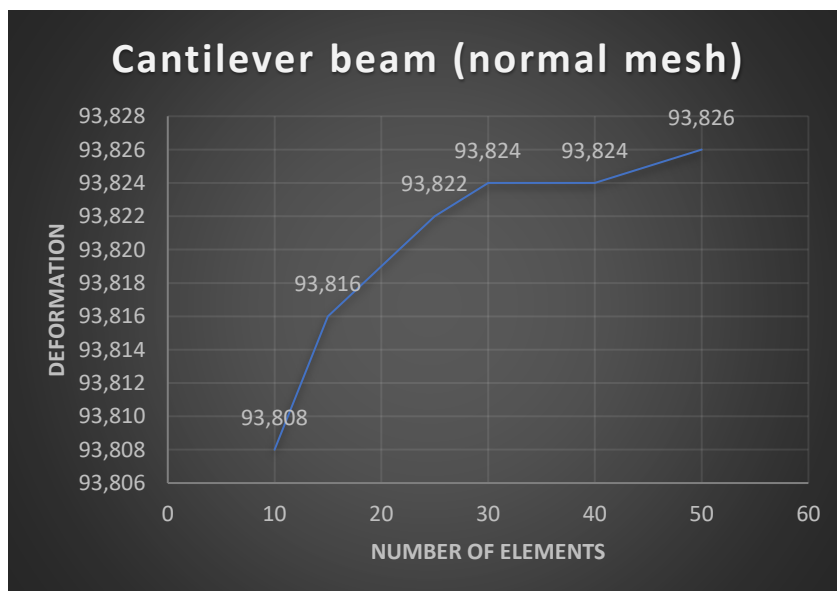


Chart 1 : Number of elements vs Deformation

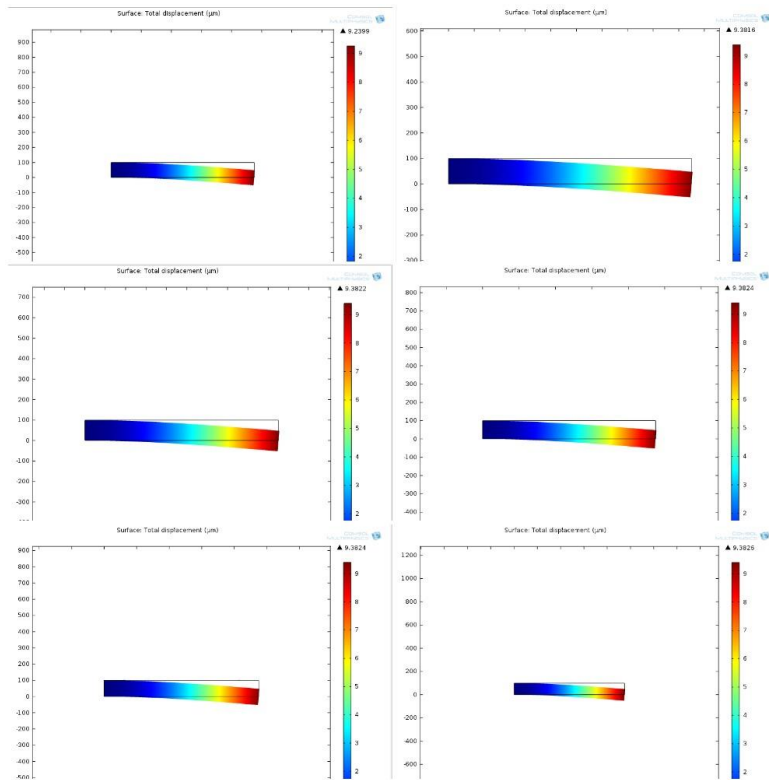


Figure 1 : Deformation for normal mesh from 10 – 50 no. of elements

- Now that we have looked about the normal triangular mesh and their results and now I have created the same beam with “Automated free triangular mesh from extremely coarse to extremely fine”. For this I changed the mesh details to *physics-controlled mesh* to play from extremely coarse to extremely fine meshing. The below tables shows the results for this method.

Coarse - Finer	Mesh	Results	Time (s)	Deformation
Extremely coarse		Mesh – 14 domains, 16 boundary elements	1	9.2399
Extra coarse		Mesh – 16 domains, 18 boundary elements	1	9.2605
coarser		Mesh – 20 domains, 18 boundary elements	1	9.3166
coarse		Mesh – 40 domains, 22 boundary elements	1	9.3298
Normal		Mesh – 70 domains, 36 boundary elements	1	9.3668
Fine		Mesh – 116 domains, 42 boundary elements	1	9.3669
Finer		Mesh – 218 domains, 60 boundary elements	1	9.3751



Extra fine		Mesh – 718 domains, 112 boundary elements	1	9.3806
Extremely fine		Mesh – 2628 domains, 222 boundary elements	1	9.3826

Table 2 : Shows the mesh details according to automated free triangular mesh and also the deformation details.

The mesh grids are created automatically as it available in the software from extremely coarse to extremely fine. Now below chart shows the comparison between the mesh types results.

This chart is the comparison between the deformation between the normal mesh and automated triangle mesh type.

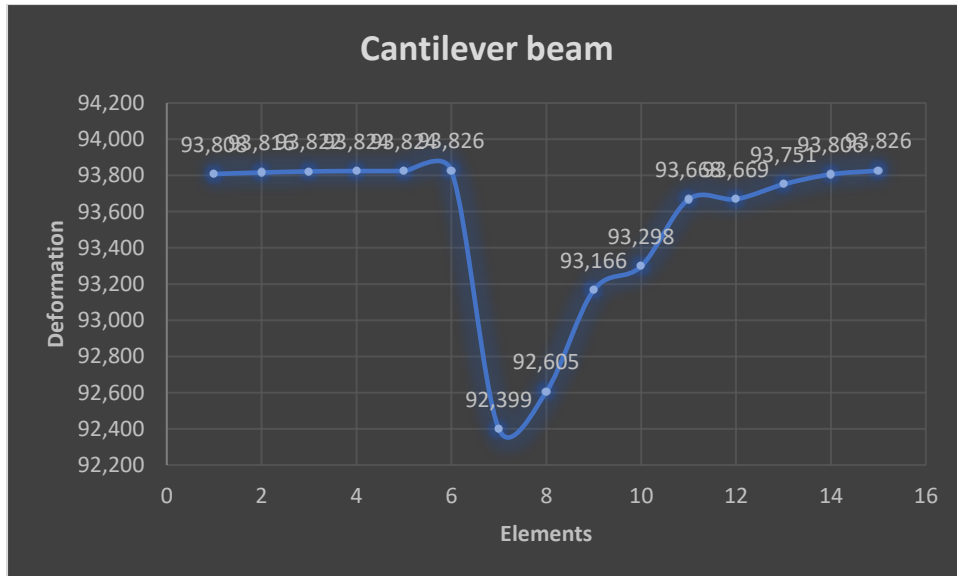
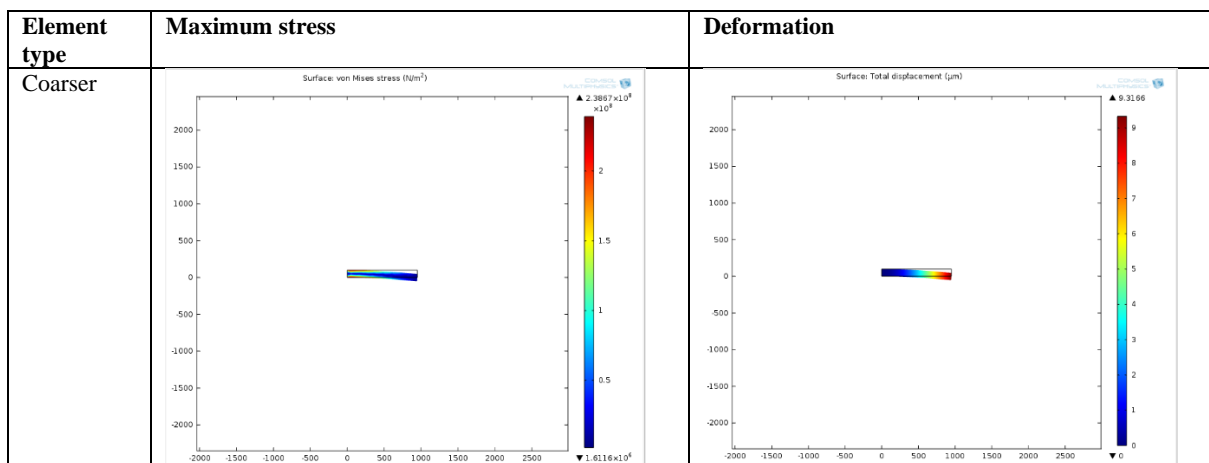


Chart 2 : comparison between the two mesh type deformation information



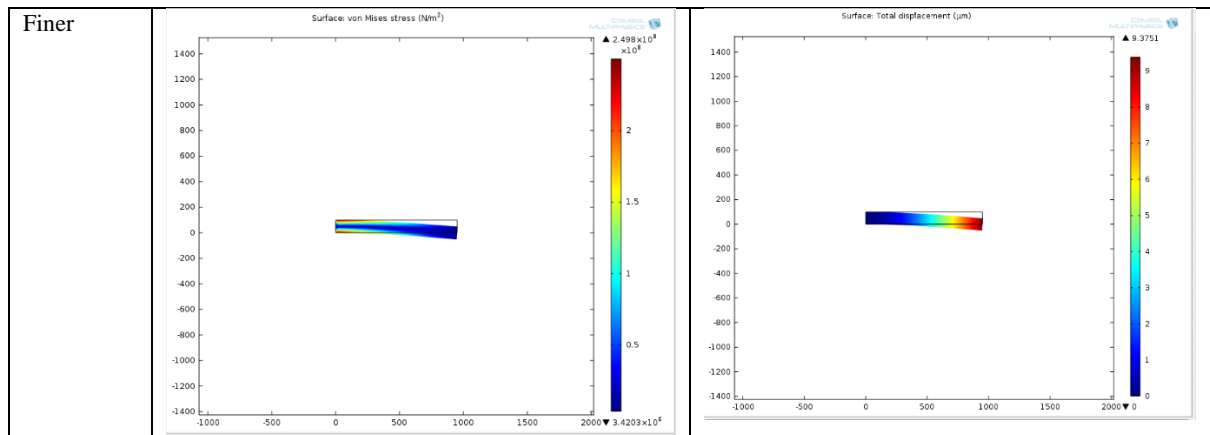


Figure 2 : Shows the automated free triangle mesh results and deformation

From the table, we can say that as the mesh is changed from extremely coarse to extremely finer, the stress values as well as deformation values increases gradually.

- Now that we have looked about normal mesh type and also the automated triangle mesh type and compared the results between them. Below shows the *mapped mesh* in which the mesh grids are divided by rectangular form. Here the mesh grid values ranges from 2 – 1000 elements in order to find where the blow up results happen.

No. of elements	Mesh	Results	Maximum stress	Time (s)	Deformation
2		Mesh – 4 domains, 8 boundary elements	1.9512e8 - 4.2608e5	1	8.8669
20		Mesh – 400 domains, 80 boundary elements	2.5107e8 - 2.4942e6	1	9.3713
50		Mesh – 2500 domains, 200 boundary elements	2.805e8 - 2.2156e6	1	9.3809
80		Mesh – 6400 domains, 320 boundary elements	3.2067e8 - 1.3227e6	2	9.3824
100		Mesh – 10000 domains, 400 boundary elements	3.421e8 - 1.2431e6	3	9.3828
500		Mesh – 250000 domains, 2000 boundary elements	5.4283e8 - 2.803e5	60	9.3838
700		Mesh – 490000 domains, 2800 boundary elements	5.9665e8 - 7.034e5	139	9.3839
850	failed				

Table 3 : Shows the mapped mesh stress, time and deformation of various elements

As we can see clearly that after 700 number of elements the simulation has failed to process because of the capacity of the computer process.

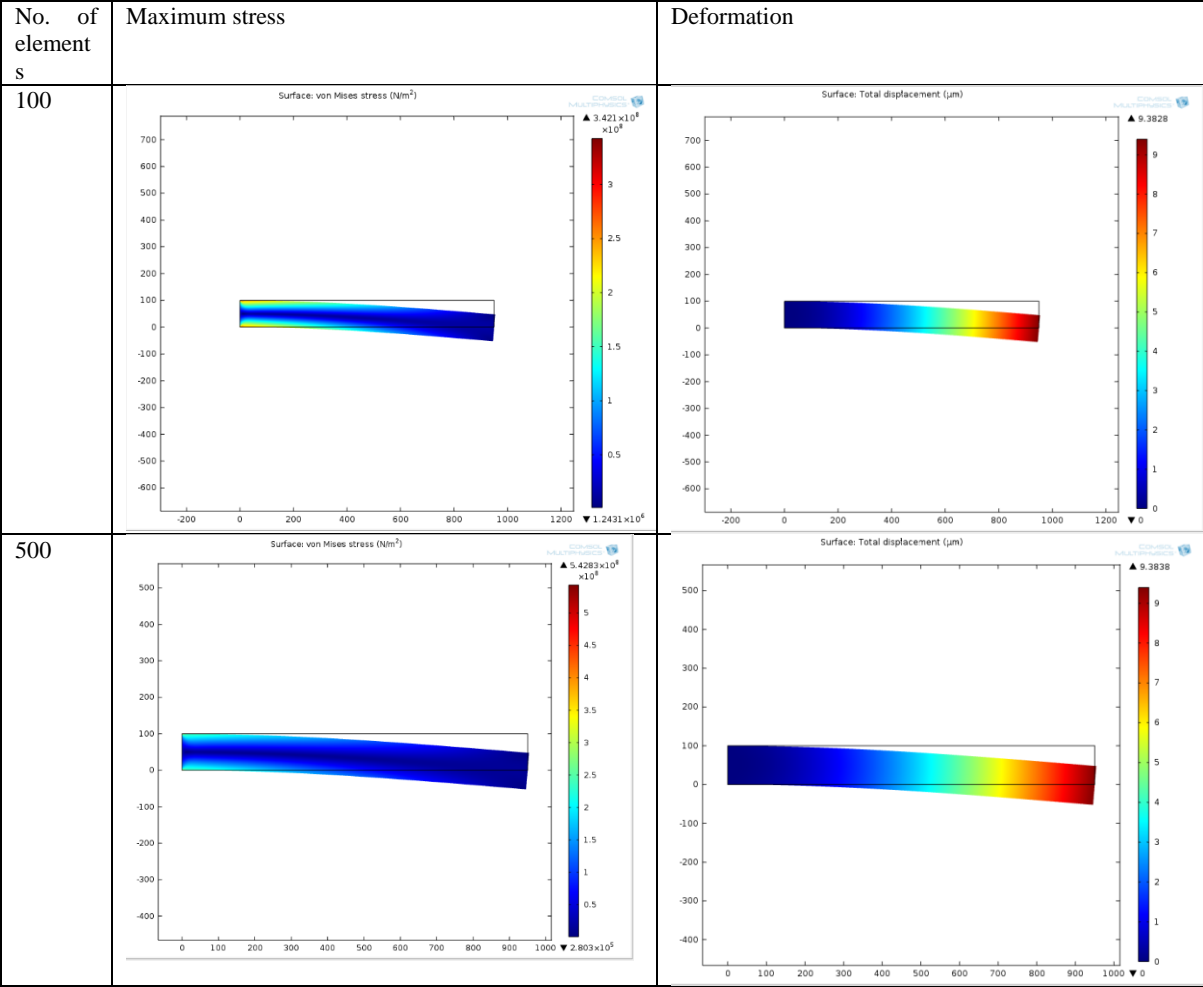


Figure 3 : Shows some of the deformation and maximum stress details

Below are the three graphs which shows the information about number of elements vs maximum stress, time and deformation.

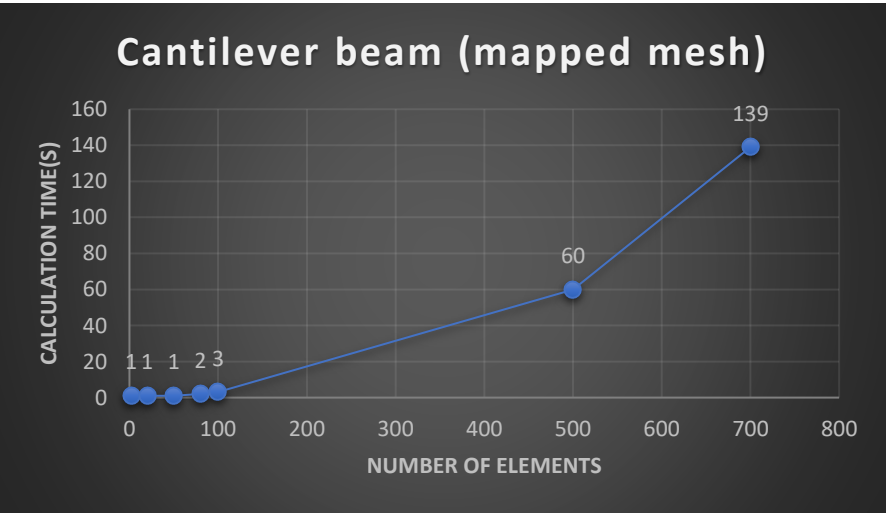


Chart 3 : Shows the information of number of elements vs time as it increases

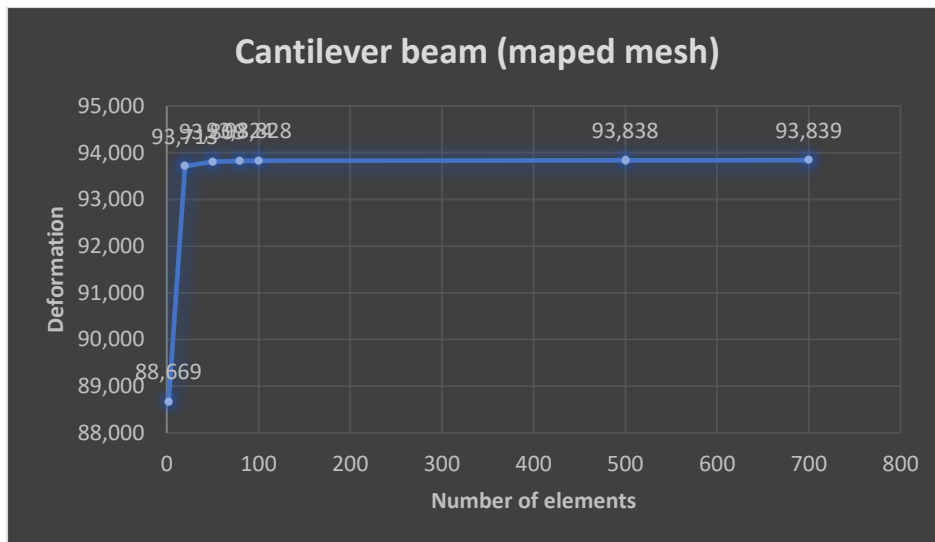


Chart 4 : Shows the information on number of elements vs deformation

The charts clearly shows that as the number of elements increases the deformation and the calculation time increases. The calculation time increases drastically and the deformation values becomes almost constant at some point.

- 3D cantilever works in the same way as the 2D cantilever beam. Modelling and simulation of 3D cantilever beam is also the same way as the 2D cantilever beam. Here we have to choose 3D option rather than 2D and continue with the same steps. Here we compare the same results with 2D cantilever beam and then solutions are recorded.

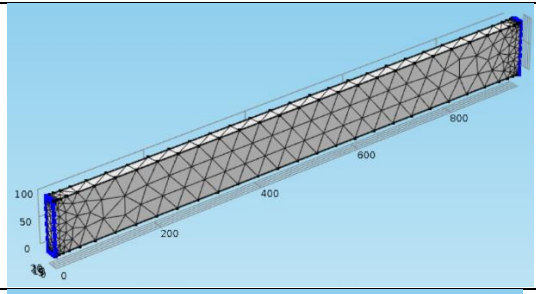
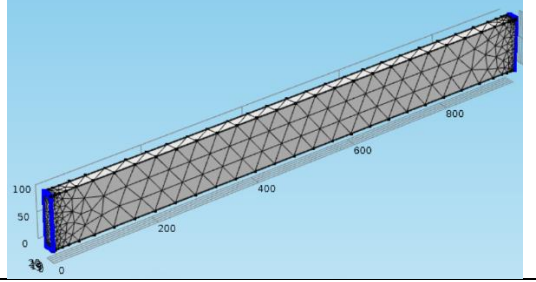
No. of elements	Mesh	Time (s)	Maximum stress	Deformation
10		2	5.8836e8 3.406e6	10.176
15		2	6.5352e8 3.0404e6	10.177

Table 4 : Shows the 3D cantilever beam deformation and stress details

Now we have to compare 2D and 3D values for better understanding. We create a table which shows the values for normal mesh, automated triangle mesh and mapped mesh and compare with it.

Normal mesh details for 2D and 3D.

No. of elements	Maximum stress 2D & 3D	Deformation 2D & 3D	Time 2D & 3D
10	2.9404e8 - 2.0248e6 ; 5.8836e8 - 3.406e6	9.3808 ; 10.176	1 ; 2
15	3.1999e8 - 2.7602e6 ; 6.5352e8 - 3.0404e6	9.3816 ; 10.177	1 ; 2

Table 5 : Shows the comparison between 2D and 3D normal mesh details

- Now that we have looked Normal mesh and their comparison. The below table shows the automated free triangular mesh for 3D.

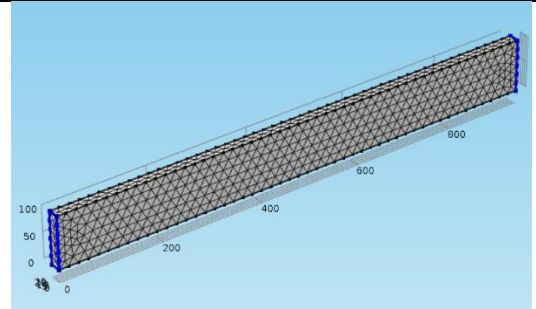
Element type	Mesh	Maximum stress	Time (s)	Deformation
Extremely fine		2.9335e8 - 4.37e6	2	10.172

Table 6 : Shows the automated free triangular mesh for 3D beam

Now comparison of 2D and 3D values

Element type	Maximum stress 2D & 3D	Deformation 2D & 3D	Time 2D & 3D
Extremely fine	2.9364e8 - 1.7222e6 ; 2.9335e8 - 4.37e6	93,826 ; 10,172	1 ; 2

Table 7 : Shows the comparison between 2D and 3D free triangular mesh details

- Now we have looked the results for the automated triangular mesh. Below shows the mapped mesh details.

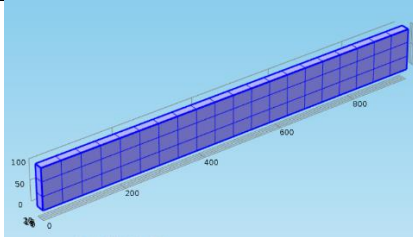
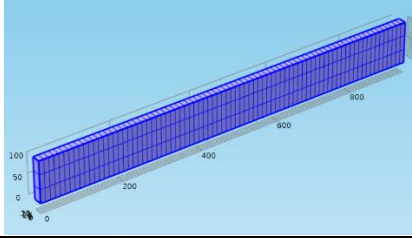
No. of elements	Mesh	Maximum stress	Times (s)	Deformation
20		2.8367e8 1.4077e6	- 1	10.151
80		3.0206e8 5.3647e6	- 1	10.173

Table 8 : Shows the details of mapped mesh of 3D beam

Now comparison of 2D and 3D values

No. of elements	Maximum stress 2D & 3D	Deformation 2D & 3D	Time (s)
20	2.5107e8 - 2.4942e6 ; 2.8367e8 - 1.4077e6	9.3713 ; 10.151	1 ; 1
80	3.2067e8 - 1.3227e6 ; 3.0206e8 - 5.3647e6	9.3824 ; 10.173	2 ; 1

Table 9 : Shows the comparison between 2D and 3D mapped mesh details

B. Part 2 : Modelling and simulation of silicon hotplate

Now that we have looked about the stress and deformation of a 2D and 3D cantilever beam using COMSOL Multiphysics. The second task would be the creating of 3D silicon hot plate which are used in sensors. The concept of silicon hot plate is that when we apply heat there will be change in resistivity of the plate, so here we are going to provide heat transfer through the plate.

Geometry of the silicon hot plate :

Plate : Length – 6mm

Thickness - 100µm

Beam : Length - 2000µm

Width - 600µm

Wire : Thickness - 10µm

Width - 100µm

Material constants :

Electrical Conductivity of Si(c) : 1e-14

Relative permittivity of Pt : 1

Boundary Conditions :

Ends of the beam fixed

Ends of the beam at room temperature (293.15K)

Heat flux through the surrounding air : Heat transfer coefficient = $5\text{W}/(\text{m}^2\text{K})$

In this method we are selecting both solid mechanics and joule heating options available on the COMSOL. The joule heating option is under the electromagnetic option. In order to create the required shape for the silicon hot plate the block option is selected to design the hot plate and silicon material is given to the plate and the beam. Platinum material is given to the wire and the electrical conductivity and the relative permittivity of platinum is changed in the material contents.

The above given dimensions are used to provide the exact diagram and the simulation is run to find out the heat transfer. The ends of the beams are fixed. The temperature of the beam is assigned to 293.15K and also the heat flux option is selected under joule heating and heat transfer coefficient of $5\text{ W}/\text{m}^2\text{K}$.

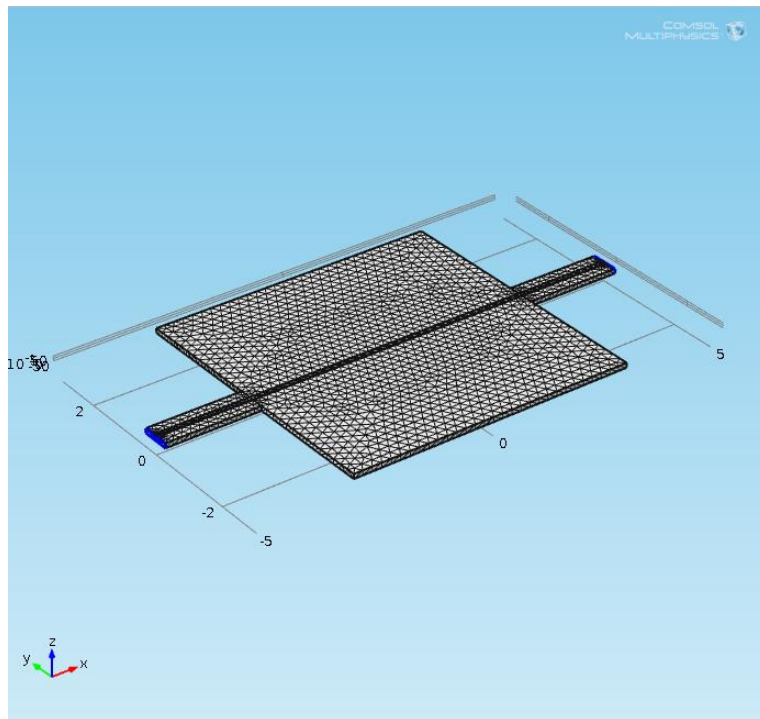


Figure 4 : Shows the meshed details of silicon hot plate

When 2V is given along the platinum wire, due to the temperature change, stress acts in the wire which is around $7.4376\text{e}8$

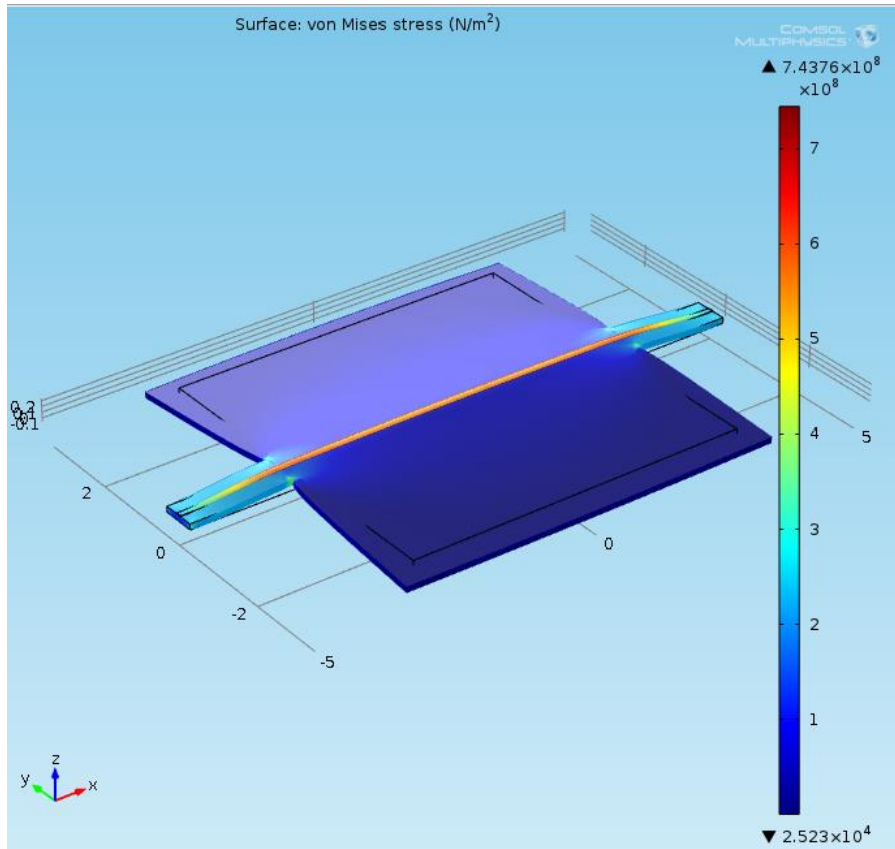


Figure 5 : shows the stress involved in the hot plate

Now that we have seen the stress for the hot silicon plate, followed by we have created the heat flux and temperature change which is also analysed.

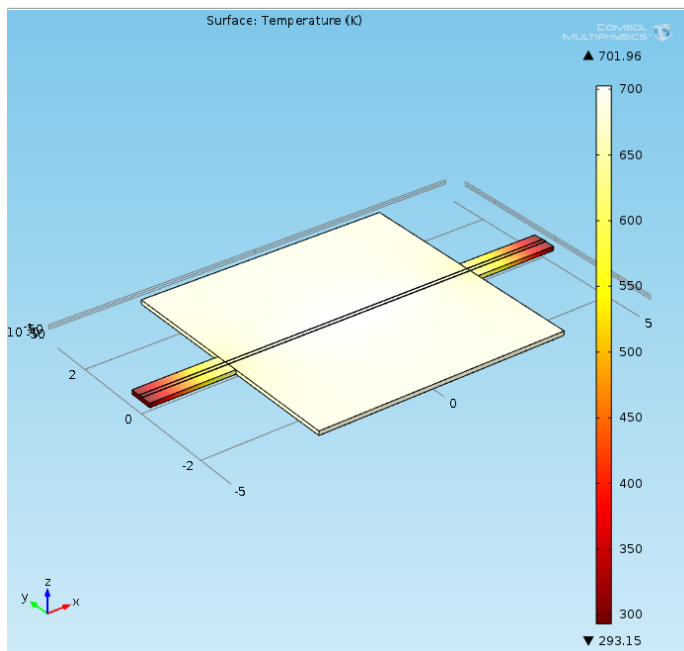


Figure 6 : Shows the heat flux in the hot silicon plate

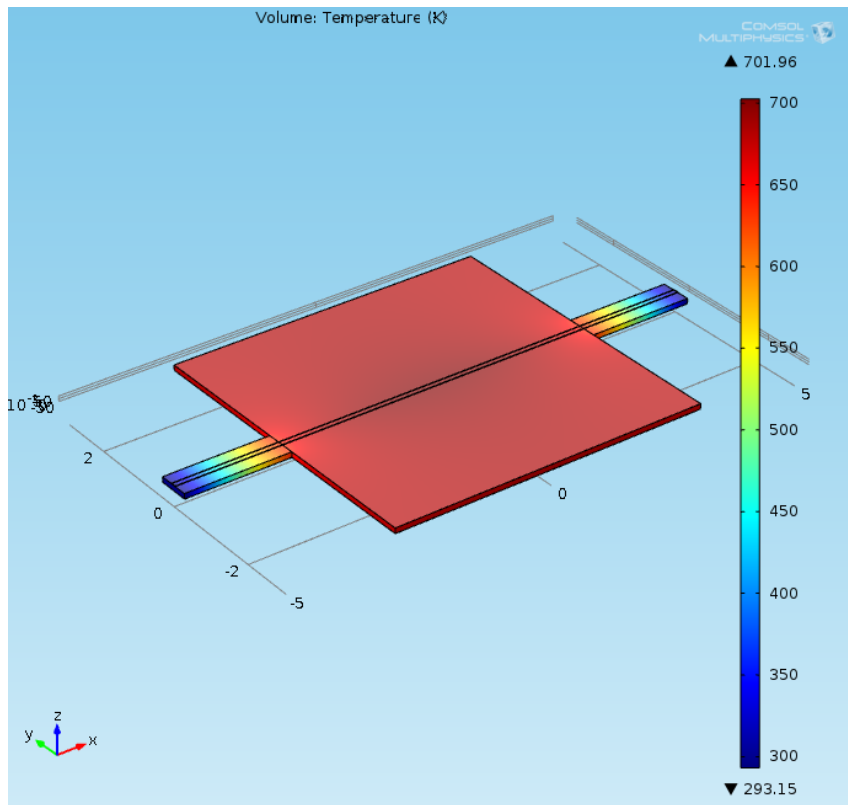


Figure 7 : Shows the heat transfer through the silicon hot plate

Conclusion :

The given task of creating 2D and 3D using COMSOL Multiphysics with desired dimensions is done. Here three types of meshing is used in order to get the desired results and all the results are compared accordingly. This report shows the exact details about the 2D and 3D comparison of maximum stress, deformation and calculation time. As the mesh grids increases the results approaching to the desired value also increases.

References :

[1]. Components and Devices (Dr. Wolfgang Kronast, 09.10.2015).