



International Journal of Multidisciplinary Research Transactions

(A Peer Reviewed Journal)

www.ijmrt.in

Static Structural Analysis of Cantilever Beam using Different Tools

Shanmugavalli M^{1*}, Priyadharshini M², Sharmila Rani A³, Swetha R⁴

¹ Professor, Department of Instrumentation and Control Engineering, Saranathan College of Engineering, Tamil Nadu, India.

^{2,3,4} Student, Department of Instrumentation and Control Engineering, Saranathan College of Engineering, Tamil Nadu, India.

*Corresponding author

DoI: <https://doi.org/10.5281/zenodo.6623302>

Abstract

This article concerns about the static structural analysis of a cantilever beam. The cantilever beam is fixed to one end and the force is applied to the other free end. This action made the beam to vibrate. It is analyzed using different types of software and the displacement and stress values are compared for the beam. ANSYS, COMSOL and Solid Works are the software used to structural analysis of the beam and the respected changes of values are compared.

Keywords: Cantilever Beam, Structural Analysis, ANSYS, COMSOL and SOLID WORKS.

1.Introduction

Flexible structures usually have low flexible rigidity and small material damping ratio. A little excitation may lead to destructive large amplitude vibration and long settling time. These can result in fatigue, instability and poor operation of the structures. Vibration control of flexible structures is an important issue in many engineering applications, especially for the precise operation performances in aerospace systems, satellites, flexible manipulators, etc. The vibration measurement technique using Digital Image Correlation as an alternative method to

determine the Young's modulus of materials. The principle is based on the linear transverse vibrations of cantilever beam with clamped-free boundary conditions. The dynamic responses of three materials were discussed. For each material, the beam was subjected to an initial static transverse displacement at its free end and then allowed to vibrate freely [1]. The mathematical and numerical formulation of recently articulated finite element method for flexible structures was discussed. The paper also presents planar 2-dimensional case of Euler Bernoulli's finite element formulation for comparison and static analysis of cantilever beam [2]. A comparative study of a thickness tapered beam and width tapered beam harvesters for potential vibration-based energy harvesting applications. Prior to the power output analysis, mathematical models are developed to evaluate strain distribution of the two configurations for proper implementation as substrates for unimorph harvester designs. In addition, vibration analysis was conducted to evaluate and compare the Eigen frequency [3]. A model for the widely used cantilever beam machine is developed to describe the dynamics of the end-effector by capturing the rotation and coupled dynamics between axes. The proposed model is validated through experiment and systematically reduced to switched linear time-invariant models for controller design [4]. The elastic pendulum based on a metal cantilever beam can convert a small linear motion into a large angle of rotational motion to achieve amplification of the low frequency vibration amplitude. Electromagnetic and piezoelectric hybrid power generation designed can improve energy harvesting efficiency [5]. The vibration characteristic analysis of the spinning cantilever beam is studied. The two linear partial differential equations governing two transverse vibrations are derived by the extended Hamilton principle. The ordinary differential dynamic equations are obtained by using the modal expansion of Galerkin method. The natural frequencies, complex modes, and stability of the spinning cantilever beam are investigated by solving the standard eigenvalue problems of the linearized system directly [6]. The behavior of gallery in a concert hall under forced vibration without damping to avoid deformations in the

structure leading to potential damage. The natural frequency of the Gallery of two proposed designs is then determined followed by finding the best design based on their natural frequencies by comparing with exact natural frequencies of a uniform cantilever beam to find the Transient Response for the given load at the tip of the Cantilever Beam [7]. This provides a weigh-in-motion device based on multi-cantilever beam structure. The device uses a piezoelectric cantilever beam to realize self-energy supply. The experimental results show that this structure can greatly improve the bandwidth. Finally, the structure parameters of the cantilever beam and mass block are obtained through finite element analysis [8]. The active vibration control of smart structures has received a lot of attention in the field of vibration suppression. A smart active control system comprising of piezoelectric materials, signal conditioning circuits and the embedded hardware is proposed in the paper [9]. The vibration measurement technique using Digital Image Correlation as an alternative method to determine the Young's modulus of materials, and the results processed and presented in the form of displacement versus time, following which the natural frequencies of the beam can be obtained using the method of Fast Fourier Transform [10]. However, in this paper the static structural analysis of the cantilever beam made up of aluminum is made to analyze the static structure. The change in deflection and von misses stress values of the beam when a force is applied to one end is discussed. The displacement occurred due to applied force values are also compared from three different software's namely ANSYS, COMSOL and Solid works.

2.Materials and Methods

2.1 Materials

2.1.1 Cantilever beam

In this project a cantilever beam is used for the vibration control. A cantilever beam is one of the examples for the flexible structures. It is made up of aluminium material and piezoelectric sensor is attached to it. The piezo actuator gets the signal when the beam is vibrated. This

cantilever beam is fixed at one end to a stand set up and the other end is free. The following tabulation Tab.1 describes about the dimensions and material properties of the cantilever beam which is used in this experiment to calculate the natural frequency of the beam based on its given dimension values.

Table 1: Dimensions and Properties of Cantilever beam

S. No	SYMBOL	PARAMETERS (with unit)	VALUES
1.	L	Length (m)	0.35
2.	B	Width (m)	0.025
3.	H	Thickness (m)	0.003
4.	P	Density (kg/m ³)	2700
5.	E	Young's Modulus (N/m ²)	7.1×10^{10}
6.	V	Poisson's Ratio (unit less)	0.34

2.2 Methods

2.2.1 Analysis using Ansys

- First open the Ansys APDL window. Set the Preferences as Structural as we need options related to Structural Analysis only.
- Now go to Pre-processor where we have to select the element type then go to Add/Delete/Edit there click Add then select Beam 2nodes 188 and then click OK. After that close the window.
- Now select Material Properties there go to Materials Models select Structural, Linear, Elastic, Isotropic where give Young's Modulus value in EX as (7.1×10^{10}) and then Poisson's ratio was 0.3 then close the window.
- Select Sections in Main Menu then select Beam and Common Sections give Width(B= 0.025), Thickness(H= 0.003) in the Beam Tool.
- Now select Modeling and select Create then click Key points and then to In Active CS. In that give 1 to Key point number and the x, y, z Coordinates as (0, 0, 0) or simply left. It will take Origin. For the second time give 2 in Key point number and 0.2 in X location coordinate.
- Now go to lines then go to Straight line click on the two sides a line will be created and then click OK on the box.
- Now we will do Meshing for that click Meshing now go to Mesh Tool click Mesh in the rectangle tool box and select the line and click OK. Give No. of element divisions as 4 then click OK.

- Now Set the Analysis as Static by click on the Solution and then set the Analysis type.
- Now go to Define Loads then click Apply in that select Structural and then go to Displacement and select On Nodes and select the 1st set of the line and click OK. Then select All DOF in Apply U, ROT on Nodes and click OK.
- Now go to Force/Moment then select the other side of the line and click OK. Then select FY and give Force/moment value as (-250) then click OK.
- Now go to Solve in Solution in that select Current LS after that you will get one window click OK there.
- Then Solution is done dialogue box will display as shown in fig.1.

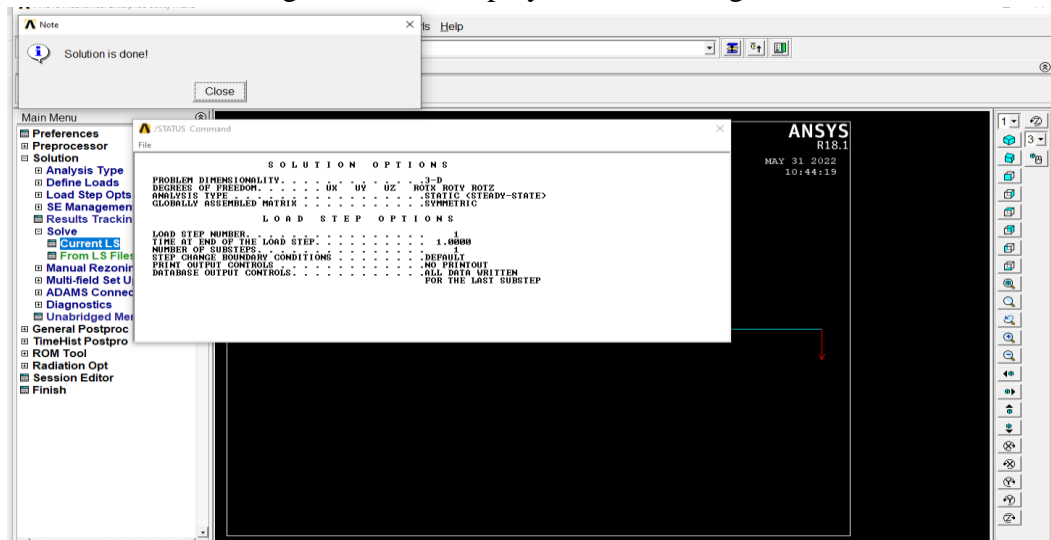


Fig.1 Solution is done for given changes

- Now go to General Postproc then select Results Summary. The result will display for the designed model is in data set file where the frequency values are recorded as shown in fig.2.

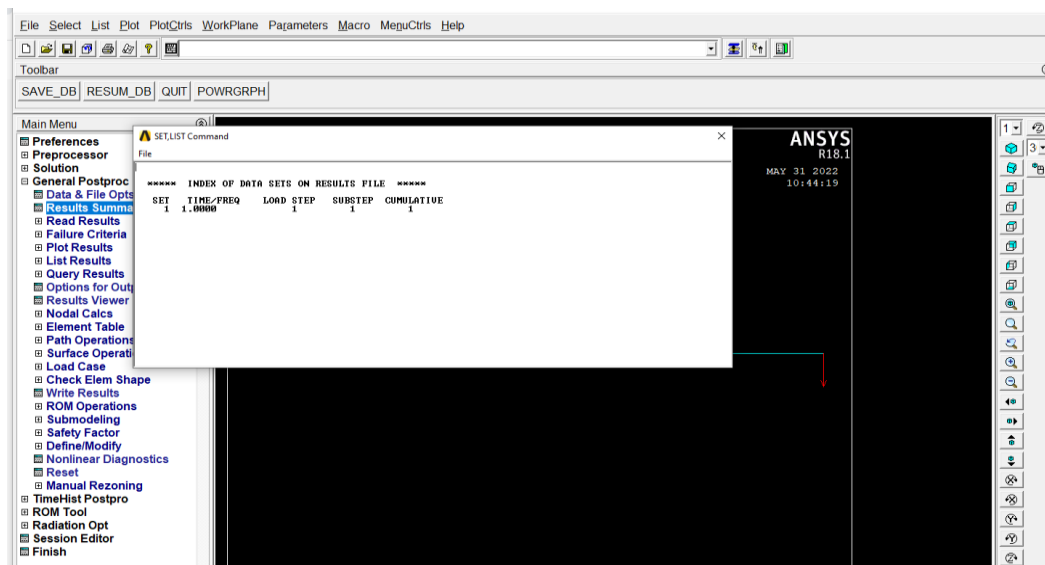


Fig.2 Data set values of the beam

- Now go to Plot Results select Deformed Shape there select Def + undef edge and then click OK. The displacement for the first set will display for the beam as shown in fig.3.

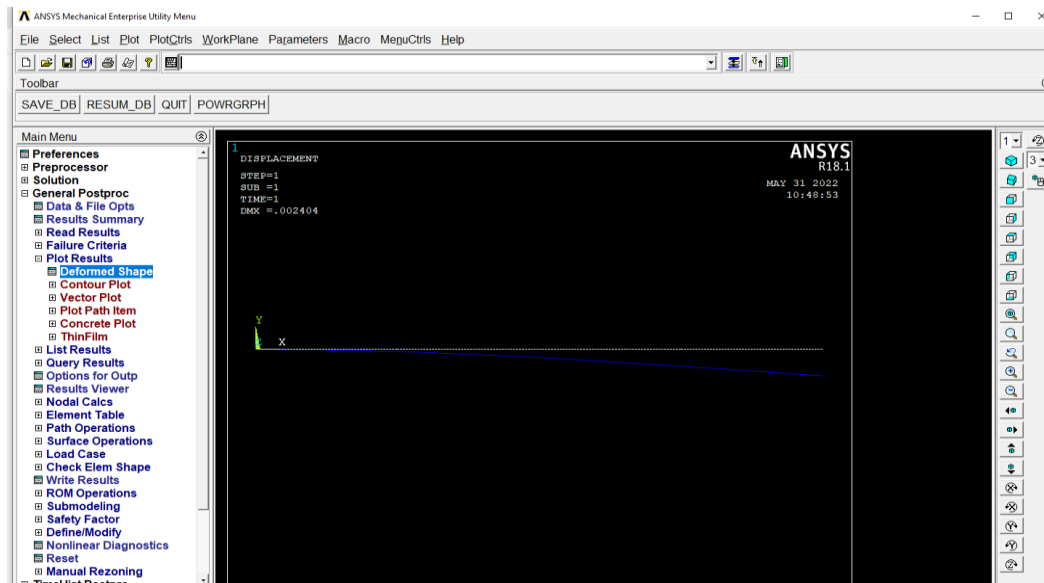


Fig.3 Displacement of the beam for the first mode

- Now select Contour then go to Nodal Solution. In nodal solution go to DOF solution there select Y-Component of displacement and click OK. Nodal solutions will display as shown in below fig.4.

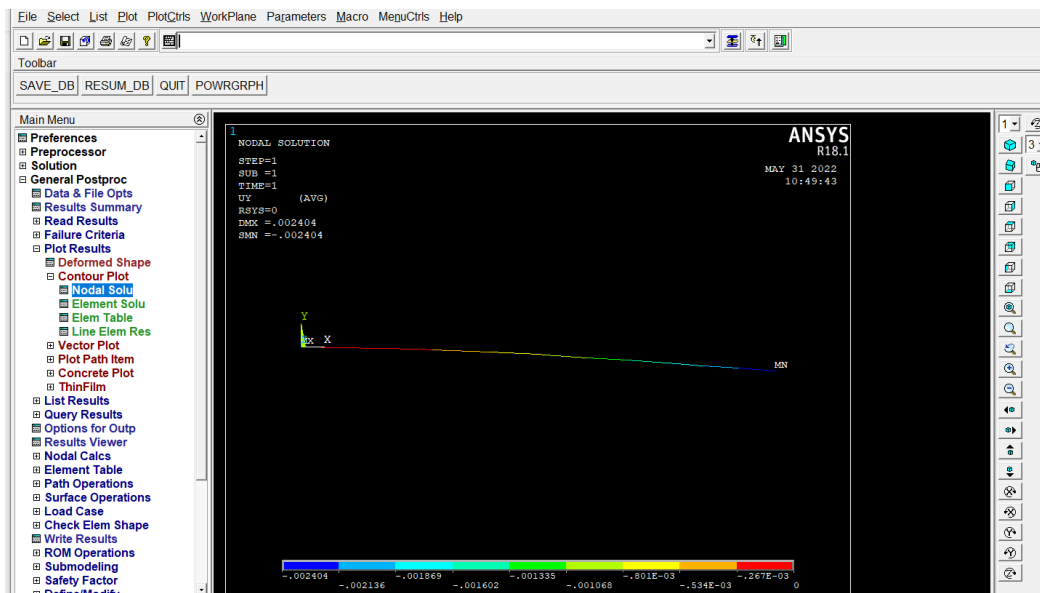


Fig.4 Nodal solution result of the beam

- Now go to List solutions then select Element solution there select 1st principle stress then click OK. It shows the Stresses at Beam Nodal Points where we can see the Maximum and Minimum Stresses as shown in below fig.5.

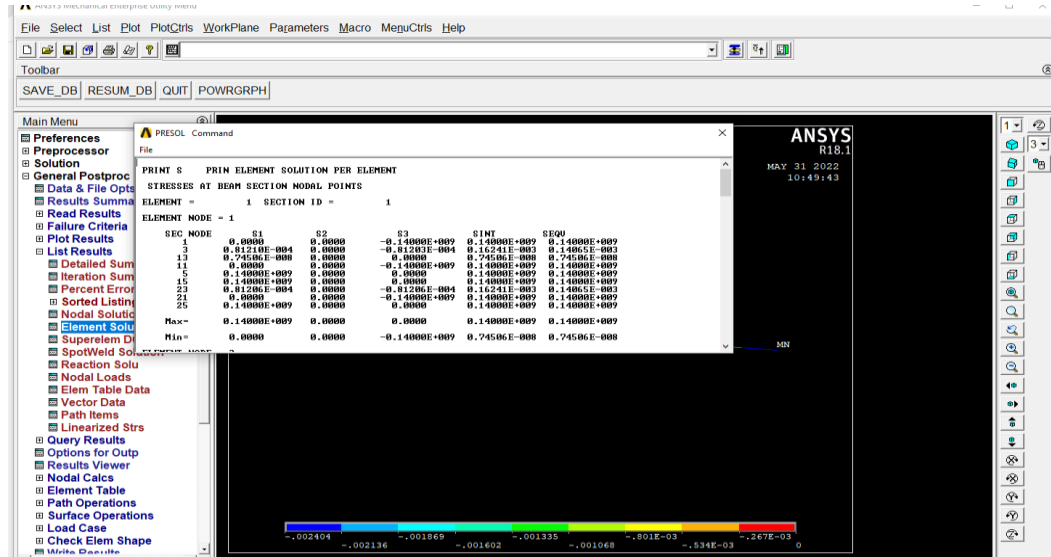


Fig.5 The Stresses at the specific sections.

2.2.2 ANALYSIS USING COMSOL

- First create a new file in the Comsol software. And then choose model wizard icon. To select the space dimension click 3D and select done.
- To select physics, click solid mechanics in the column of structural mechanics and click study. Select stationary which is used when field variables do not change over time.
- In geometry, select block and choose the length unit in m and give the respective dimensions in the width, depth and height column in meters and click build selected. A model of cantilever beam will display in the Graphics.
- To add the material of the cantilever beam, go to add material section and select the aluminum in the built in column.
- For physics, choose the side 1 for fixed constraint where the one side of the cantilever beam is fixed and the load is applied at the other end. To apply load, select the side 6 for boundary load where the pressure is applied. Give the force value -1000N in the load type section.
- To mesh the designed model, click the mesh icon and select normal and build all for meshing the entire beam.
- To make the changes of the deflection of beam for the applied pressure, select study and compute the designed model with the respective dimensions.

- The designed model of cantilever beam will deflect for the applied change is given below in the fig.6.

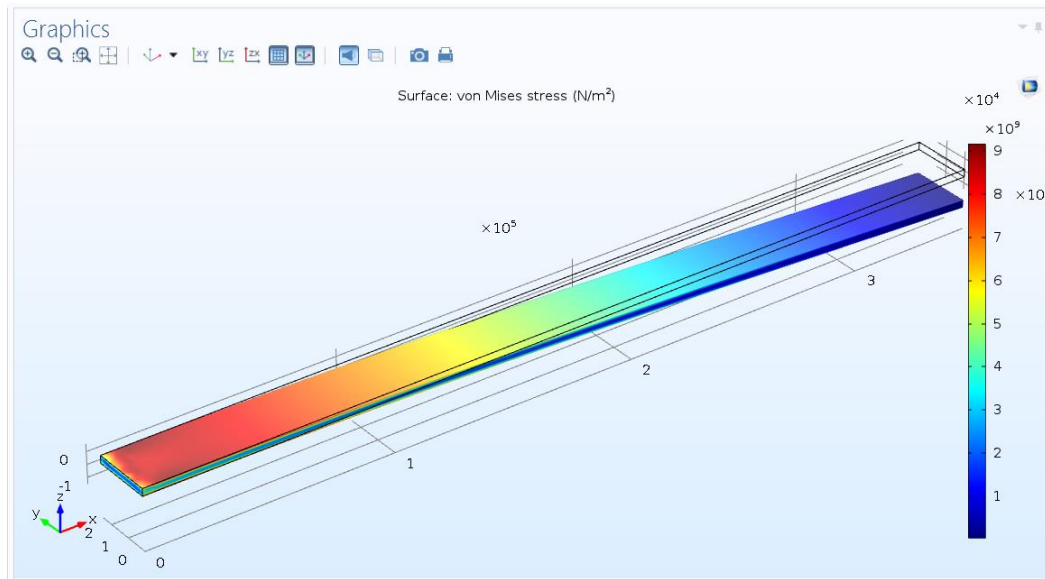


Fig.6 Deflection of the beam for applied changes

- To obtain the displacement values, go to result section and choose 3D plot group and volume and select plot, the displacement value of the beam will display as shown in fig.7.

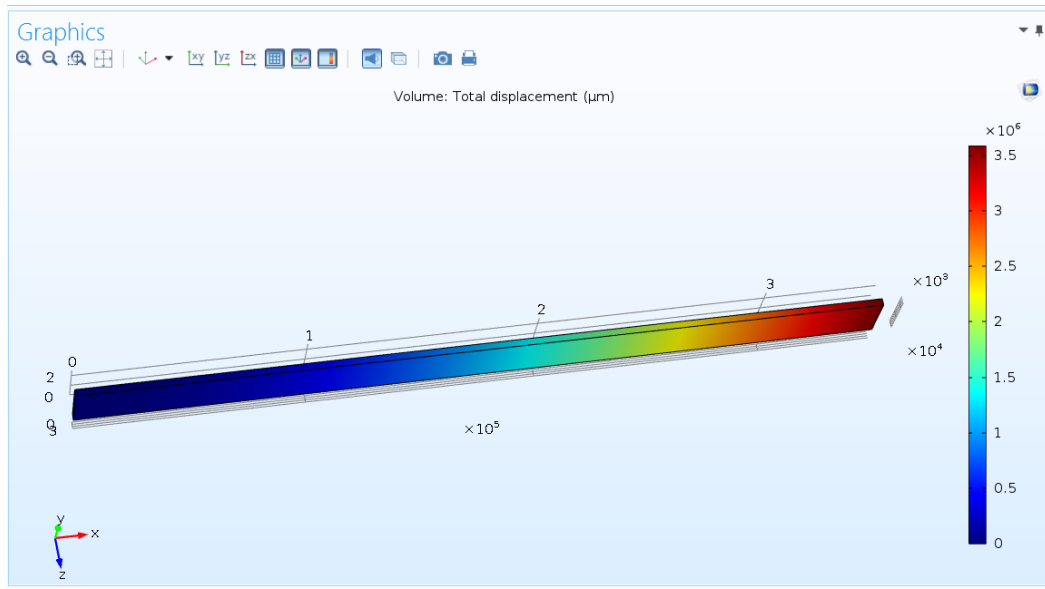


Fig.7 Displacement values of the cantilever beam

- In result, select 1D plot graph and choose line graph. The line graph will display the total elastic strain energy and arc length for the designed cantilever beam as shown in fig.8.

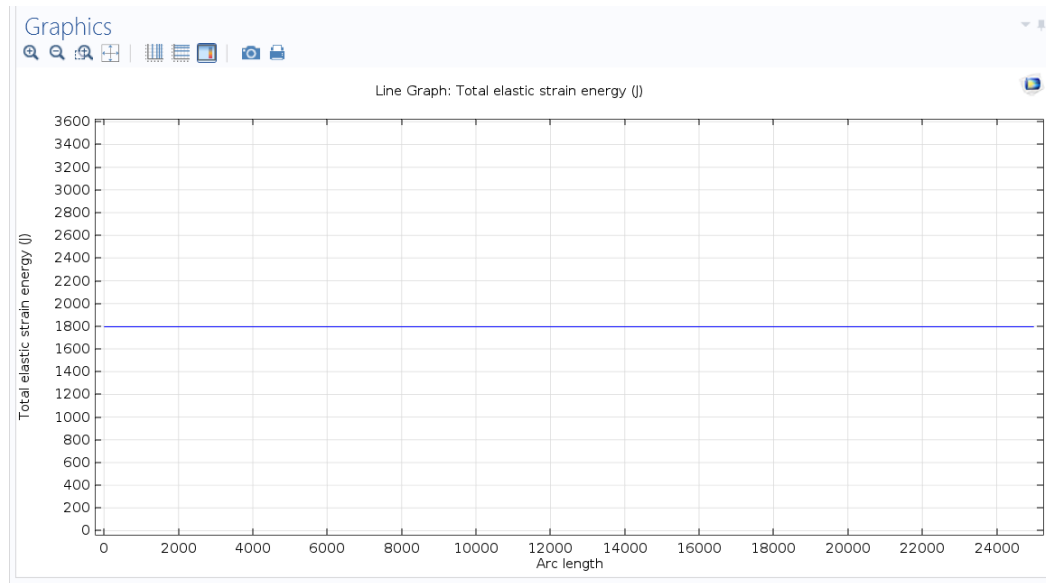


Fig.8 Line Graph for Arc length and total elastic strain energy

2.2.3 Analysis using Solidworks

- As a first step, draw the top plane. Then click the rectangle to draw a rectangle shaped cantilever beam. After that, click a smart dimension and give the dimension of our cantilever beam and click the tick symbol.
- Then go to features and then click extruded boss/base, then give depth value and enter it. A model will be displayed in the graphics as shown in fig. 9.

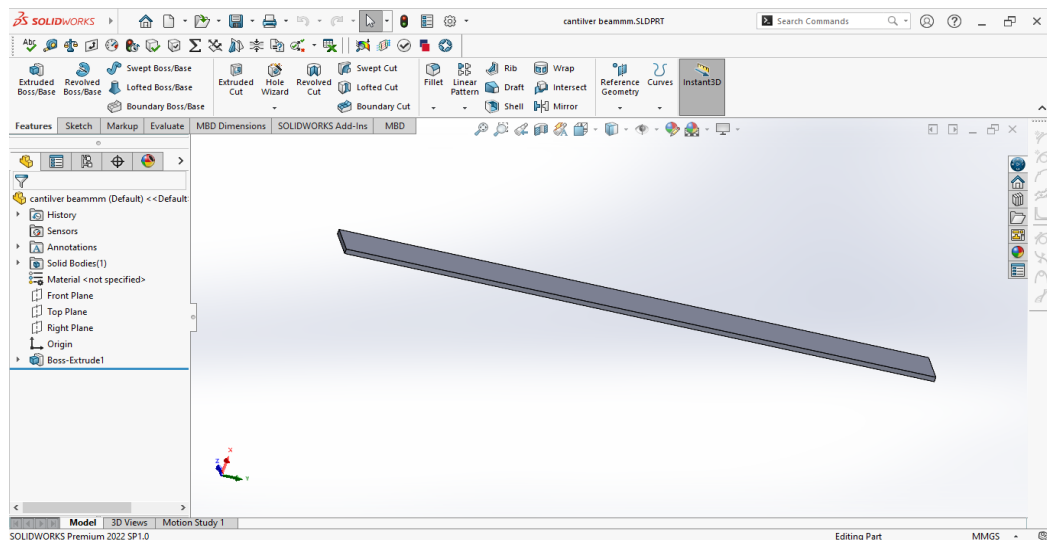


Fig.9 Designed model of beam in solid works

- Save the model in the PC by creating a new folder. Go to simulation and then click a study advisor and then go to new study and choose the static analysis and click a tick symbol.
- For fixing the end, we need to choose the face and go to fixtures advisor and click fixed geometry. And we need click a tick symbol by checking the face.

- For applying the load in another end, we need to go to external loads and click force. Then choose the particular edge and give a force for selected direction of 2N and click a tick symbol. It will be display as shown in fig. 10.

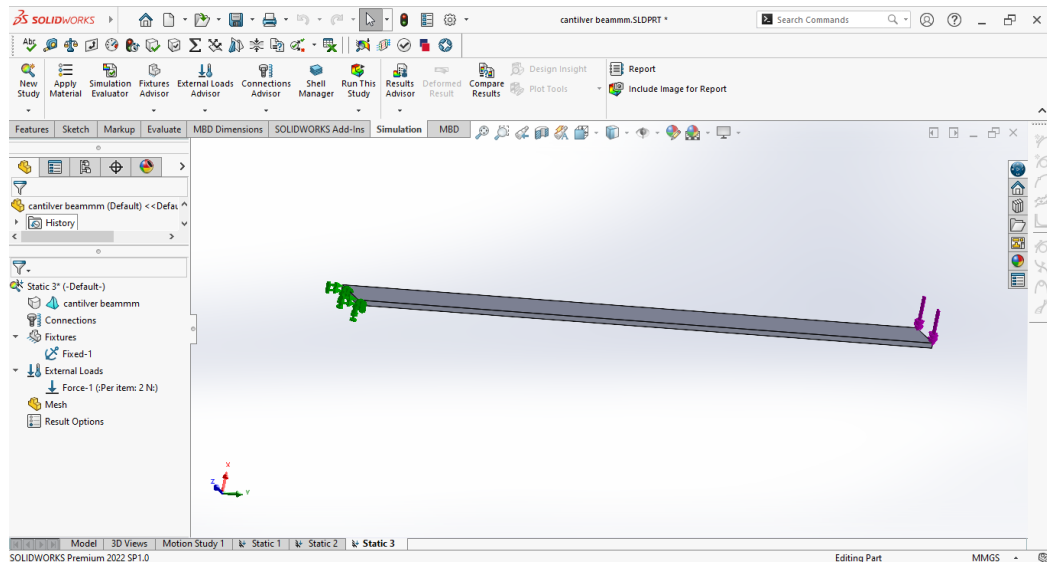


Fig.10 Load applied in one end of the beam

- We need to choose a material for our cantilever beam, so we need to select apply material icon option and choose the aluminium alloy. Then apply it and cancel it.
- For meshing, we need to get to create mesh and click the tick symbol. The meshed form of beam will be displayed as shown in fig. 11.

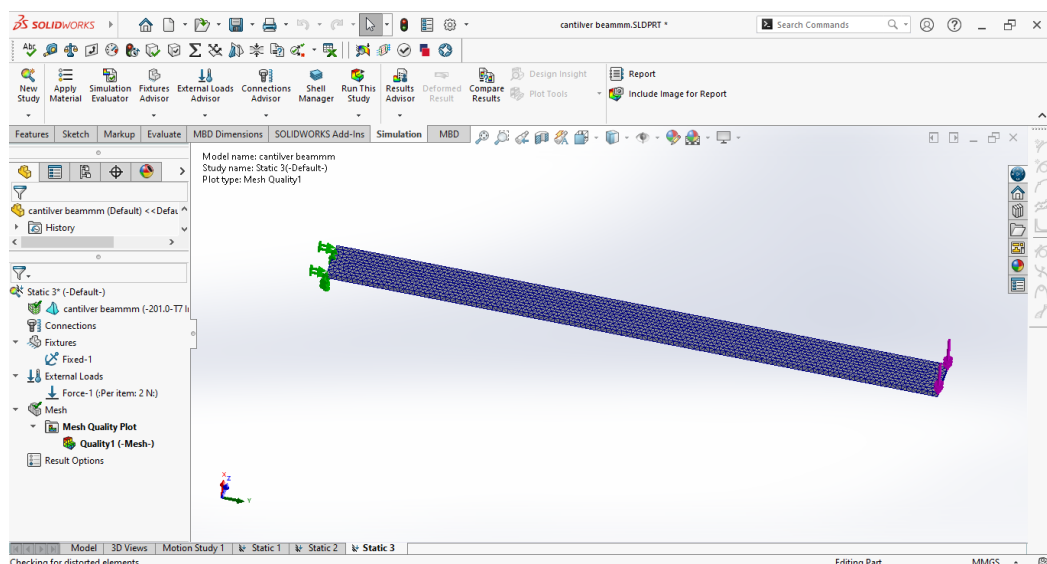


Fig.11 Meshed form of cantilever beam

- Then we need to run this study, we will get the deformed shape of cantilever beam as shown in fig.12 . Then animate the cantilever beam by clicking the animation option.

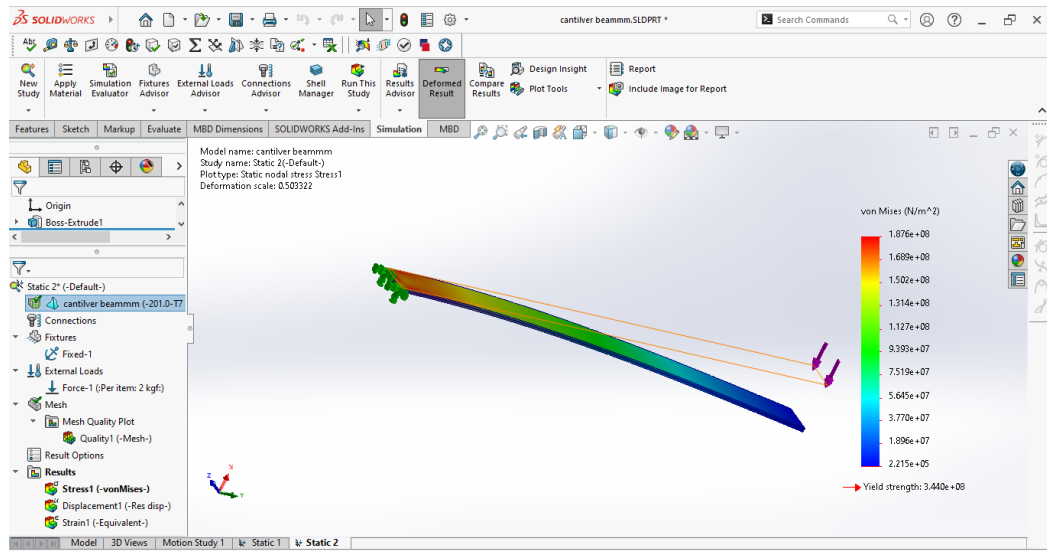


Fig.12 Displacement of cantilever beam

3.CONCLUSION

Thus the static structural analysis of the cantilever beam is done using different software like ANSYS where we got the von misses stress values and displacement values of the beam and using COMSOL the same data and the line graph is recorded and finally using solid works we got the results of the deflection of the cantilever beam in its static position when the force will be applied at the one end. The following table explains the comparisons of stress and displacement values of the cantilever beam in each tool.

PARAMETER	ANSYS	COMSOL	SOLID WORKS
VON MISSES STRESS (N/m ²)	0.002404	0.002372	0.002215

REFERENCES

1. S. Malik and T. Solaija, "Static and Dynamic Analysis of Absolute Nodal Coordinate Formulation Planar Elements," 2020 17th International Bhurban Conference on Applied Sciences and Technology (IBCAST), 2020, pp. 167-173.
2. D. S. Ibrahim, S. Beibei, O. A. Oluseyi and U. Sharif, "Performance Analysis of Width and Thickness Tapered Geometries on Electrical Power Harvested from a Unimorph Piezoelectric Cantilever Beam," 2020 IEEE 3rd International Conference on Electronics Technology (ICET), 2020, pp. 100-104.

-
3. M. Yuan, L. Li and Z. Xu, "Modeling and Contouring Control for Cantilever Beam Machine With Structural Flexibility," in *IEEE Access*, vol. 10, pp. 27809-27818, 2022.
 4. Z. Wang, G. Li and T. Yao, "An electromagnetic and piezoelectric coupled energy harvester using cantilever beam for low frequency vibration," 2019 6th International Conference on Systems and Informatics (ICSAI), 2019, pp. 350-354.
 5. J. -H. Yang and X. -D. Yang, "Analytical Investigation of Vibration Characteristic about Spinning Cantilever Beam," 2020 5th International Conference on Mechanical, Control and Computer Engineering (ICMCCE), 2020, pp. 81-84.
 6. S. A. Siddiqui, A. Ahmad, A. A. Siddiqui and P. Chaturvedi, "Stability Analysis of a Cantilever Structure using ANSYS and MATLAB," 2021 2nd International Conference on Intelligent Engineering and Management (ICIEM), 2021, pp. 7-12.
 7. X. Wang, B. -d. Bao, R. -q. Xu and Y. Wang, "Design of Device For Weigh-in-Motion Based on Multi-Cantilever Beam Structure," 2020 15th Symposium on Piezoelectricity, Acoustic Waves and Device Applications (SPAWDA), 2021, pp. 381-385.
 8. M. Cui, W. Tang, Y. Han and Z. Li, "Smart Active Vibration Control System Using Piezoelectric Materials," 2020 Chinese Control And Decision Conference (CCDC), 2020, pp. 2611-2615.
 9. T. B. Loh, M. C. Shin Rong, S. H. Goh, K. K. Hau, K. L. Goh and J. J. Chong, "Determination of Static and Dynamic Young's Modulus of A Cantilever Beam using Digital Image Correlation (DIC) Method," 2021 24th International Conference on Mechatronics Technology (ICMT), 2021, pp. 1-5.