

Finite Element Analysis Of Thermal Loads Of Mechanical Structures

Mohammad Gogazeh¹, Hassan Al-Dabas², Nabeel Mousa³, Sayel M. Fayyad⁴

^{1,2,3} Mechanical Engineering Department, Faculty of Engineering, Philadelphia University

⁴Department of Mechanical Engineering, Faculty of Engineering Technology, Al-Balqa Applied University, Amman, Jordan

Abstract

This paper concentrates on mechanical structures. The design of static structures assumes they always have the same geometry (in fact, so-called static structures can move significantly, and structural engineering design must take this into account where necessary), but the design of moveable or moving structures must account for fatigue variation in the method in which load is resisted and significant deflections of structures. The forces which parts of a machine are subjected to can vary significantly and can do so at a great rate. The forces which a boat or aircraft are subjected to vary enormously and will do so thousands of times over the structure's lifetime. The structural design must ensure that such structures are able to endure such loading for their entire design life without failing. The main objective of this paper is to study the behavior of mechanical structural under thermal loads in a simple and logical manner and to apply modification. In this project the study of the behavior of mechanical structural under thermal load is based on the understanding of a few basic concepts, on the use of simplified models and simulation of mechanical structures. This approach makes it possible to develop all the necessary formulas in a rational and logical manner, and to clearly indicate the conditions under which they can be safely applied to the analysis and design of actual engineering structures and machine components. Throughout the paper, free-body diagrams are used to determine external or internal forces. The use of drawing and analysis program will also help understanding the superposition of loadings and resulting stresses and deformation.

Keywords: Thermal Loads, deflection, FEM, Mechanical Structures, Design.

Introduction

Finite Element Analysis (FEA) is a powerful computational technique used to simulate and predict the behavior of mechanical structures under various loading conditions. It involves discretizing a complex structure into smaller, simpler elements, which are then analyzed to determine how the structure responds to forces, heat, and other physical phenomena. When it comes to mechanical structures, FEA is commonly employed to evaluate both thermal and mechanical loads. These two types of loads can significantly affect the integrity and performance of a structure, and understanding their combined effects is essential for design, optimization, and safety assurance [1-2, 4-5].

1. Thermal Loads in Mechanical Structures

Thermal loads refer to the effects caused by temperature variations within a material or structure. Changes in temperature can induce thermal expansion or contraction, which may lead to stress development, deformation, or even failure. In FEA, thermal analysis is conducted to determine temperature distribution across a structure, and then mechanical stress and strain are calculated by considering the thermal gradients.

2. Mechanical Loads in Mechanical Structures

Mechanical loads refer to external forces or pressures applied to a structure, including static and dynamic loads such as weight, pressure, or impact. These loads can induce deformation, vibration, or failure if not properly accounted for. FEA helps in evaluating how mechanical forces interact with a structure, assessing factors such as stress, strain, and displacement.

3. Combined Thermal and Mechanical Loads

In many real-world scenarios, mechanical structures are subjected to both thermal and mechanical loads simultaneously. The interaction between these two types of loads can lead to complex behavior, including thermal stresses and strain caused by differential expansion, or mechanical deformation exacerbated by thermal effects. FEA allows for the simultaneous simulation of both mechanical and thermal loads, enabling a more comprehensive analysis and more accurate predictions.

4. Applications and Importance

Finite Element Analysis of thermal and mechanical loads is essential in the design of mechanical structures across a wide range of industries. Applications include aerospace engineering, automotive design, energy systems, civil engineering, and more. By simulating how structures behave under real-world conditions, engineers can optimize designs for performance, safety, and durability while minimizing costs and material waste. In summary, FEA offers a detailed and accurate method to predict the behavior of mechanical structures under the influence of both thermal and mechanical loads. This approach is indispensable for ensuring that designs meet required performance criteria and safety standards in complex and demanding environments [6-10]. Kim et al. (2022) introduced an overlapping finite element approach with an adaptive form function that takes temperature distribution into account to increase solution accuracy and minimize meshing effort in thermal stress analysis. The form functions are built adaptively by varying interpolation weights based on the temperature distribution within an element. The performance of the suggested method is assessed by solving 1D and 2D representative problems. It produces more accurate results than the original overlapping finite element method while incurring no additional computing costs. It is effective, especially when a coarse mesh is utilized in the region with high temperature gradients [1]. Traves et al. (2021) advocated doing simulation-based behavioral research of electronic components in a DC-DC converter for thermomechanical loads in order to develop more dependable electronic goods. It is said that temperature management is critical in any electronics package for optimal performance; thus, when these packages are subjected to structural loads at elevated temperatures, the resulting thermal strain and strains may cause these goods to fail. This study describes the substantial effort done using simulation tools to estimate the influence of thermomechanical loads on crucial electronic components, component leads, and other critical converter parts. The methods used to avoid damaging interactions caused by high temperatures and thermal stress are also highlighted [3].

-Beam Behavior Under Theoretical Structural Loads

This section will talk about the Derivation of the theoretical equation of beam deflection and study all parameters of equation and their effect on deflection value.

The Derivation of the theoretical equation of the deflection for simple supported beam.

Singularity method will be used to derive the deflection equation for a simple supported beam

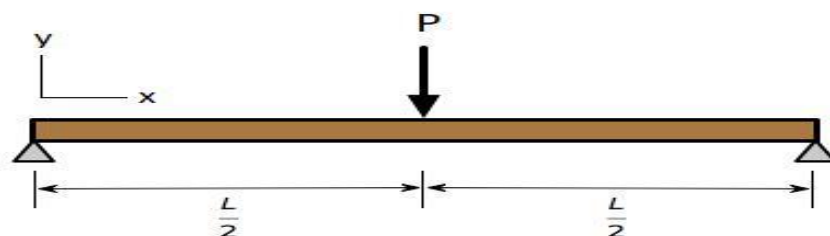


Figure1. simply supported beam

Verification of prokon software

The maximum deflection will be compute mathematically using the previous equations and compared with resulting computed by using prokon software. The following assumptions are used to computing mathematically maximum deflection:

Rectangular cross section with

Second moment of inertia is $I=5.27 \times 10^{-6} \text{ m}^4$

Modulus of elasticity (E) is equal 205 GPa

Beam length is equal 3 m

The value of x is equal 1.5 m

The single load (p) put at the center of the beam and its value is equal 100 KN

$Y(1.5) = -52.07 \text{ mm}$

For the purposes of the study the prokon software will be used to get the same results. Prokon software will be used to draw deflection diagram as show in figure 2 and calculate it value for simple supported beam.

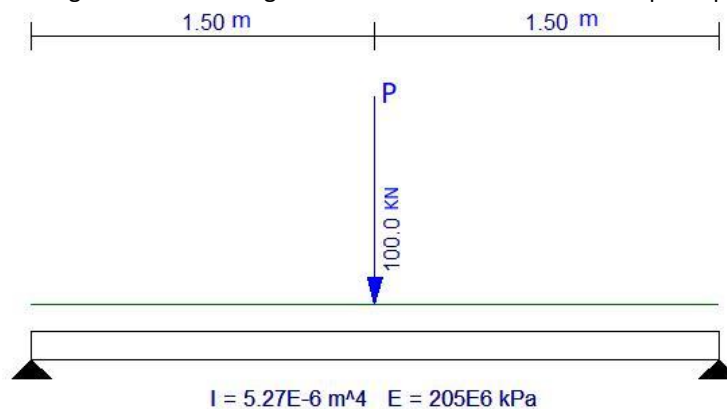


Figure 2 simply supported beam

-Finite Element Results

Finite element analysis (FEA) is a computational technique used to obtain approximate solutions; methods of this type are needed because analytical methods cannot cope with the real, complicated problems that are met within engineering. The method is now applied to problems involving a wide range of phenomena, including vibration, heat conduction, fluid mechanics and electrostatic. There are many of programs which works on finite element such (solid-work, ANSYS... etc) in this study, the solid-work was used as the main program. Mesh is a network that is formed of cells and nodes; it can have almost any shape in any size and is used to solve partial differential equation. Each cell of the mesh represents an individual solution of the equation, when increase the amount of mesh on the model, the solution close to exact solution (analytical). In the beginning, the model or beam is designed with specific dimensions and properties which include shape of cross section and the type of material. Then, the type of fixture is determined for beam whereas gravitational force is put on beam. Thereafter, the concentrated load is applied on the specific space on the beam Moreover, the beam is divided to fine mesh which is more accurate, in order to solve the equations automatically. Compare the finite results with mathematical results. Secondly, for the thermal analysis: The beam is exposed to thermal load rather than mechanical load; the values resulting are listed in table. Finally, the mechanical & thermal results are combined together [11-13].

-Deflection under thermal load using solid work by difference temperatures

The following assumptions will be used to draw the beam and help to put a load as quantity and direction as well as determine the type of material for simulation:

- Pine-pine
- beam length is 3 m.
- Moment of inertia is $I=5.27 \times 10^{-6} \text{ mm}^4$
- Reference temperature 25°C
- AISI 1020 steel cold rolled
- beam length is 3 m.

total node for:	-mesh type: solid mesh
rectangular (81811)	-total element for:
I-section (69332)	rectangular (53007)
C-section (97965)	I-section (35923)
	C-section (58183)

Table 1 relationship between change the temperature and deflection for rectangular section.

Temperature ($^\circ \text{C}$)	60°	65°	70°	75°	80°
Maximum deflection(mm)	-1.167	-1.334	-1.5	-1.667	-1.834

Effect changes the temperature on max deflection as shown figure 3.

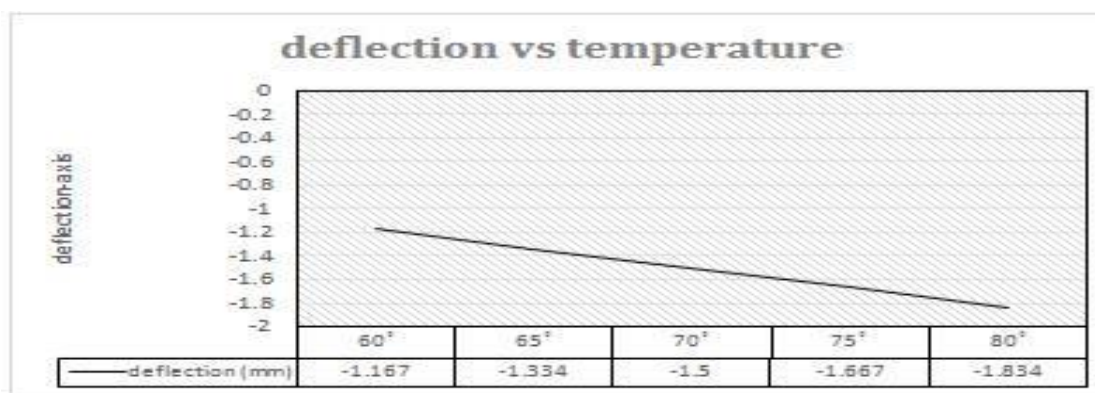


Figure 3 Explain the relationship between deflection & temperature which the x-axis represents temperature and y-axis represents the deflection.

Table 2 Comparison of deflection between different types of cross section by change temperature.

Temperature (c°)	60°	65°	70°	75°	80°
Deflection for rectangular section (mm)	-1.167	-1.334	-1.5	-1.667	-1.834
Deflection for l-section (mm)	-0.85	-0.97	-1.09	-1.215	-1.33
Deflection for c-section (mm)	-1.23	-1.14	-1.58	-1.76	-1.94

For comparison the deflection by change temperature between rectangular section , l-section and c- section as shown figure 4.

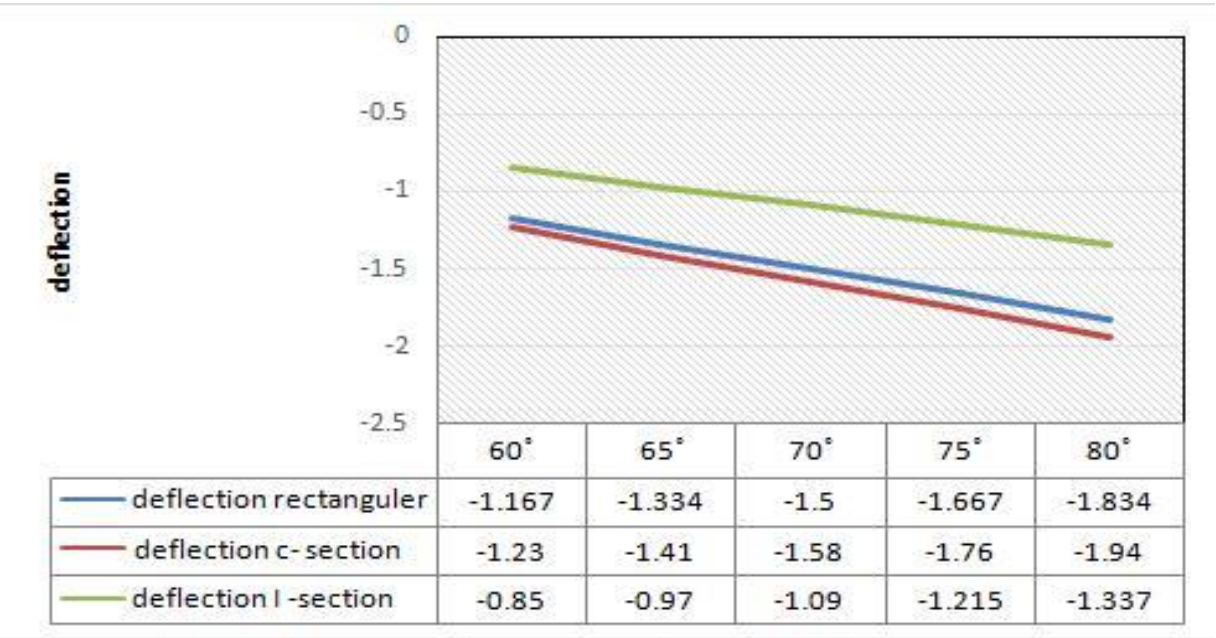


Figure 4 Explain the relationship between deflection & temperature which the x-axis represents temperature and y-axis represents the deflection.

This model is drawn on the solid-work as shown figure 5, it represents deflection which result from change temperature for rectangular section.

Figure 5 deflection profile which results from change temperature.

Figure 6 & figure 7 show the values of deflection resulting from changing the support of beam.

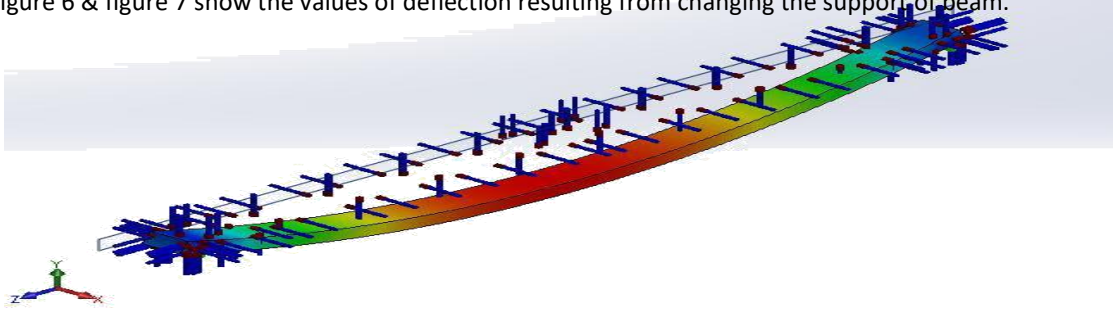


Figure 6 deflection with change temperature to 60°&65 ° respectively.

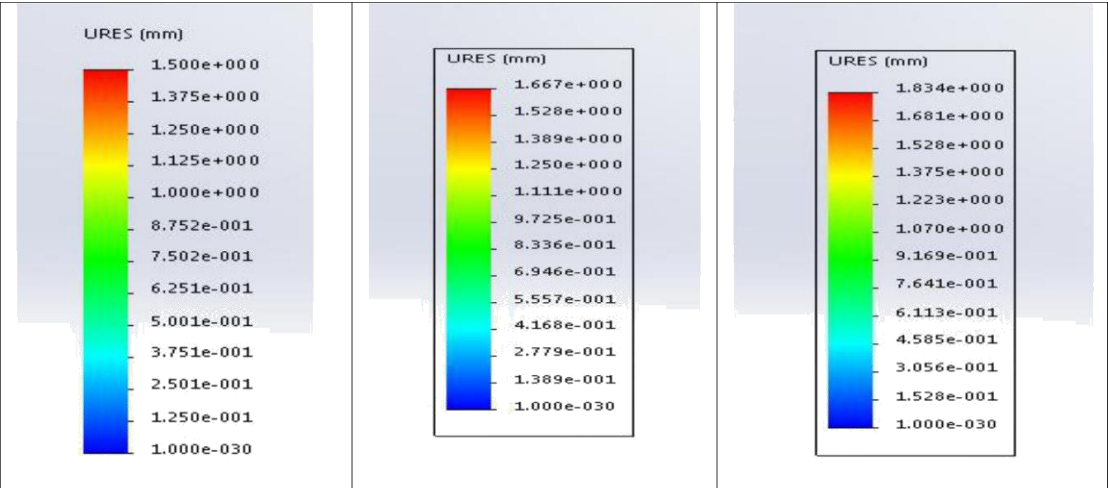
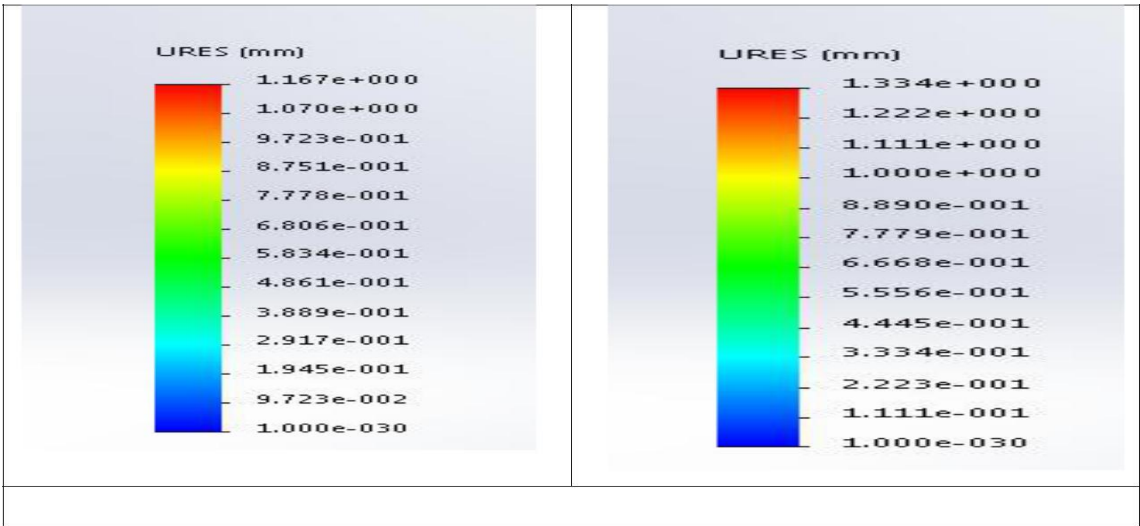


Figure 7 deflection with change temperature to 70 ° ,75° ,80° respectively

Deflection under thermal load using solid work by change material

The following assumptions will be used to draw the beam and help to put a load as quantity and direction as well as determine the type of material for simulation:

- Pine-pine
- beam length is 3 m.
- Moment of inertia is $I=5.27 \times 10^{-6} \text{mm}^4$
- Reference temperature 25 c°
- applied temperature at 60 c°

total node fore :	
	-mesh type : solid mesh
rectangular (81811)	
	-total element for :
I-section (69332)	
	rectangular (53007)
c-section (97965)	
	I-section (35923)
	c-section (58183)

Table 3 relationship between change the temperature and deflection for rectangular section.

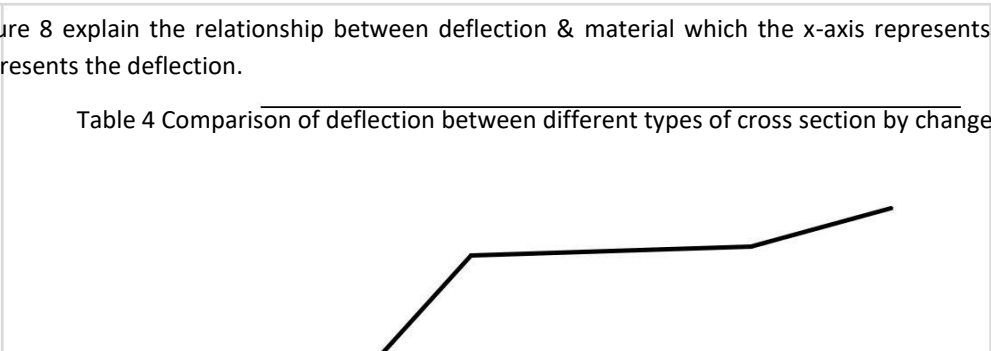
Type of material	Deflection (mm)
AL 1060 alloy	-2.394
Steel cold rolled	-1.167
Cu hot pressed	-1.137
Ductile iron	-1.097
Ti-3AL-8v-6cr-4Mo-4zr alloy	-0.7978

Effect changes the material on max deflection as shown figure 8.

deflection vs material

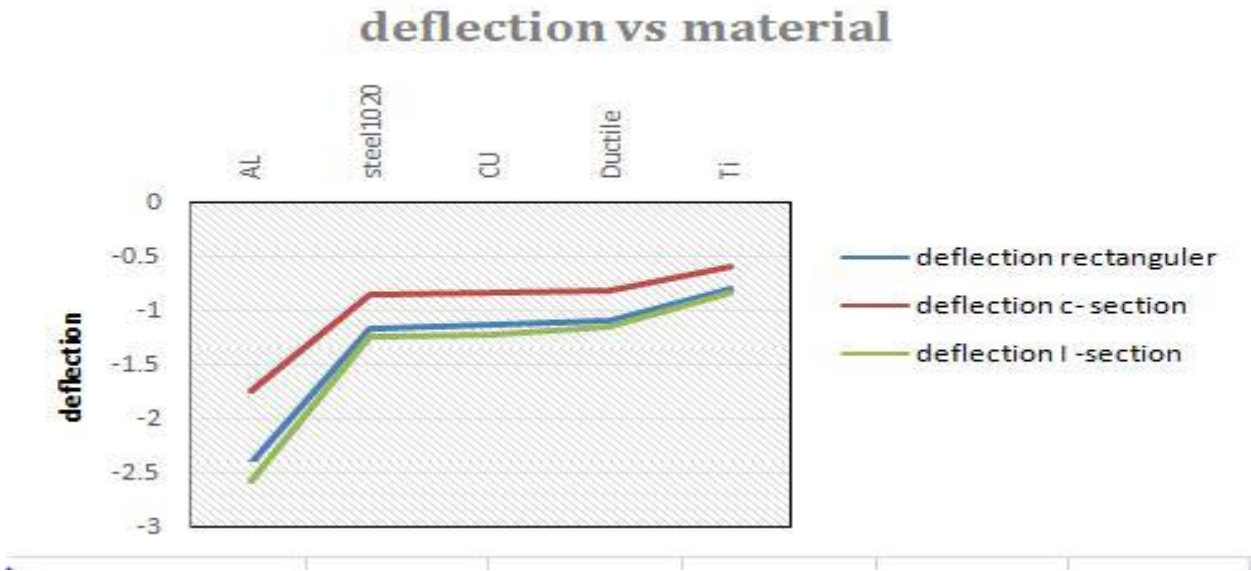
Figure 8 explain the relationship between deflection & material which the x-axis represents material and y-axis represents the deflection.

Table 4 Comparison of deflection between different types of cross section by change material.



Material	AL-1060	Steel1020 cold Rolled	Cu hot Pressed	Ductile iron	Ti-3Al-8v alloy
Deflection for rectangular section (mm)	-2.394	-1.167	-1.137	-1.097	-0.7978
Deflection for I-section (mm)	-1.74	-0.85	-0.828	-0.799	-0.5817
Deflection for c-section (mm)	-2.56	-1.23	-1.217	-1.154	-0.832

For comparison the deflection by change material between rectangular section, I-section and c- section as shown figure 9.



-1.5					
-2.5					
-3	2.4E-05	1.7E-05	1.4E-05	1.1E-05	8E-06
Deflection (mm)	-2.394	-1.167	-1.137	-1.097	-0.7978

Figure 9 explain the relationship between deflection & material which the x-axis represents material and y-axis represents the deflection.

This model is drawn on the solid-work as shown figure10, it represents deflection which result from change material for rectangular section.

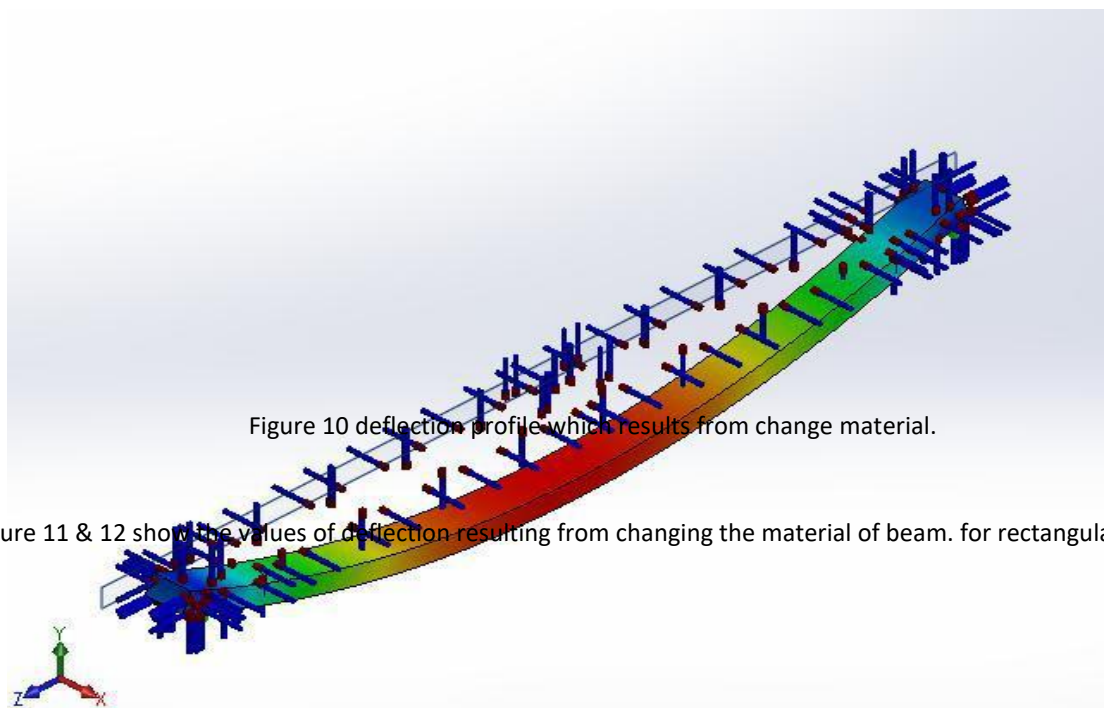


Figure 10 deflection profile which results from change material.

Figure 11 & 12 show the values of deflection resulting from changing the material of beam. for rectangular section.

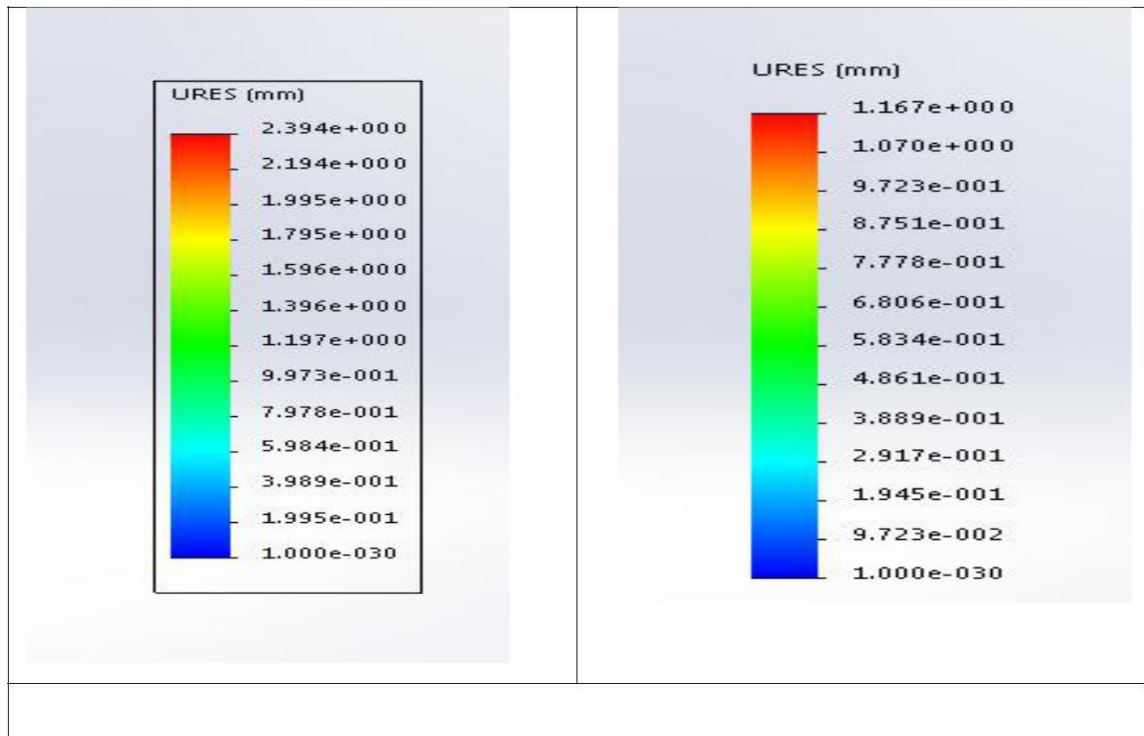


Figure 11 deflection with change material to AL & steel respectively.

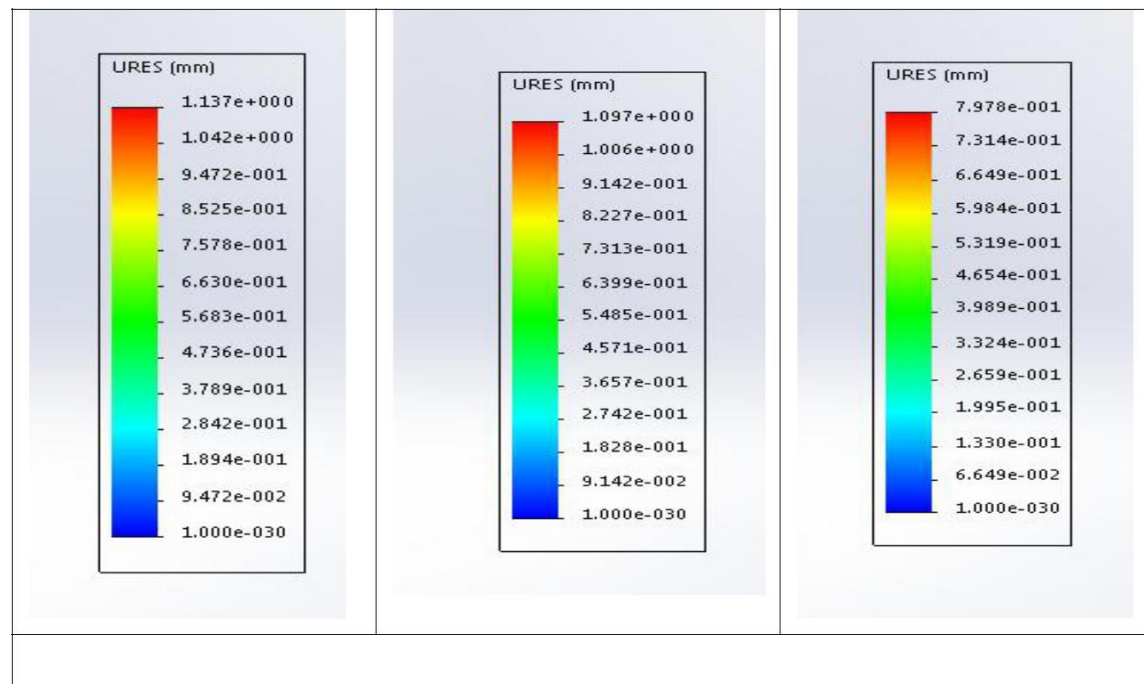


Figure 12 deflection with change material to Cu, ductile iron, Ti respectively.

AISI 1020 steel cold rolled

Pine-pine

beam length is 3 m.

Reference Moment of inertia is $I = 5.27 \times 10^{-6} \text{ mm}^4$

temperature 25 c°

applied temperature at 60 c°

-total node fore :	-mesh type : solid mesh
rectangular (81811)	-total element for :
I-section (69332)	rectangular (53007)
C-section (97965)	I-section (35923)
	C-section (58183)

Deflection under thermal load using solid work by change cross section

The following assumptions will be used to draw the beam and help to put a load as quantity and direction as well as determine the type of material for simulation:

Table 4 relationship between change cross section and deflection.

Cross section	rectangular	I-section	C-section
Maximum deflection(mm)	-1.167	-0.8506	-21.83

Effect changes cross section on max deflection as shown figure 13.

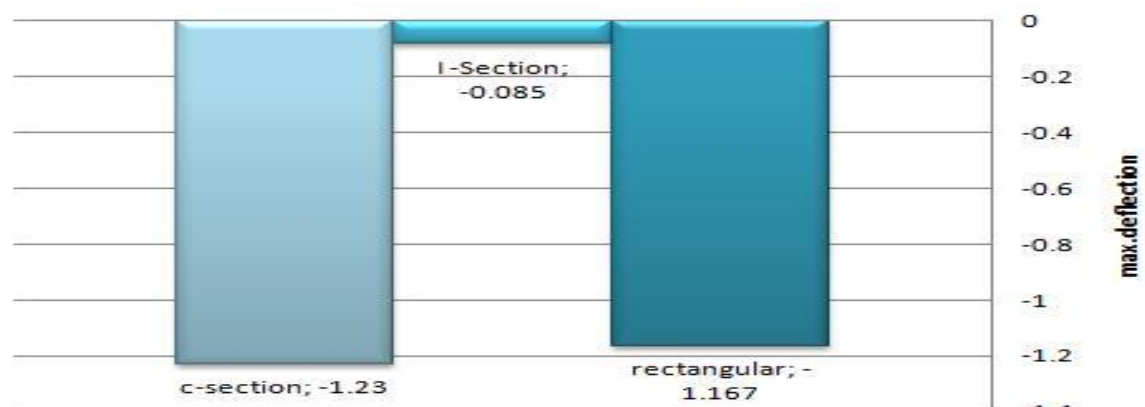


Figure 13 Compare between the value of deflection for cross section

This model is drawn on the solid-work as shown figure 14, it represents deflection which result from change cross section.

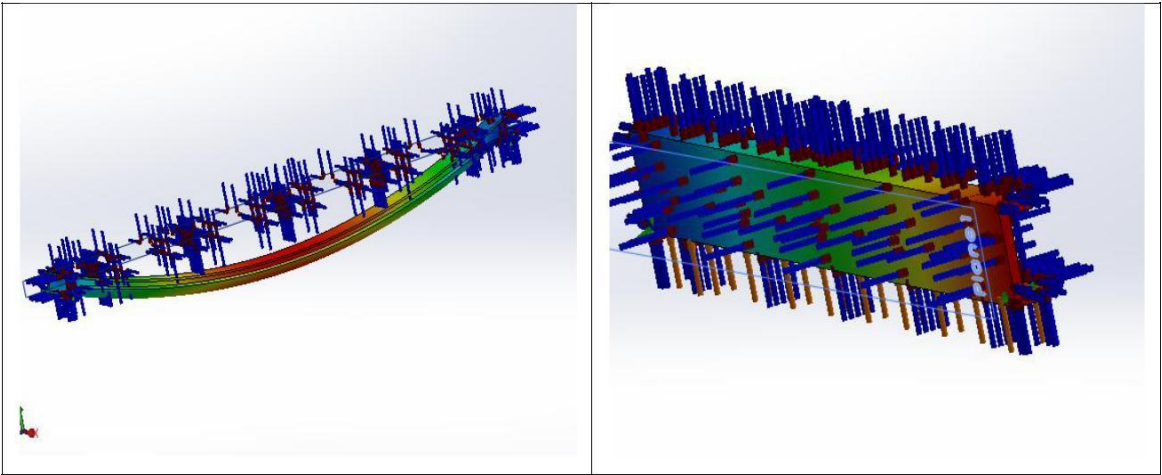


Figure 14 deflection profile which results from change cross section include I-section & c-section respectively.
Figure 15 shows the values of deflection resulting from changing the cross section for beam.

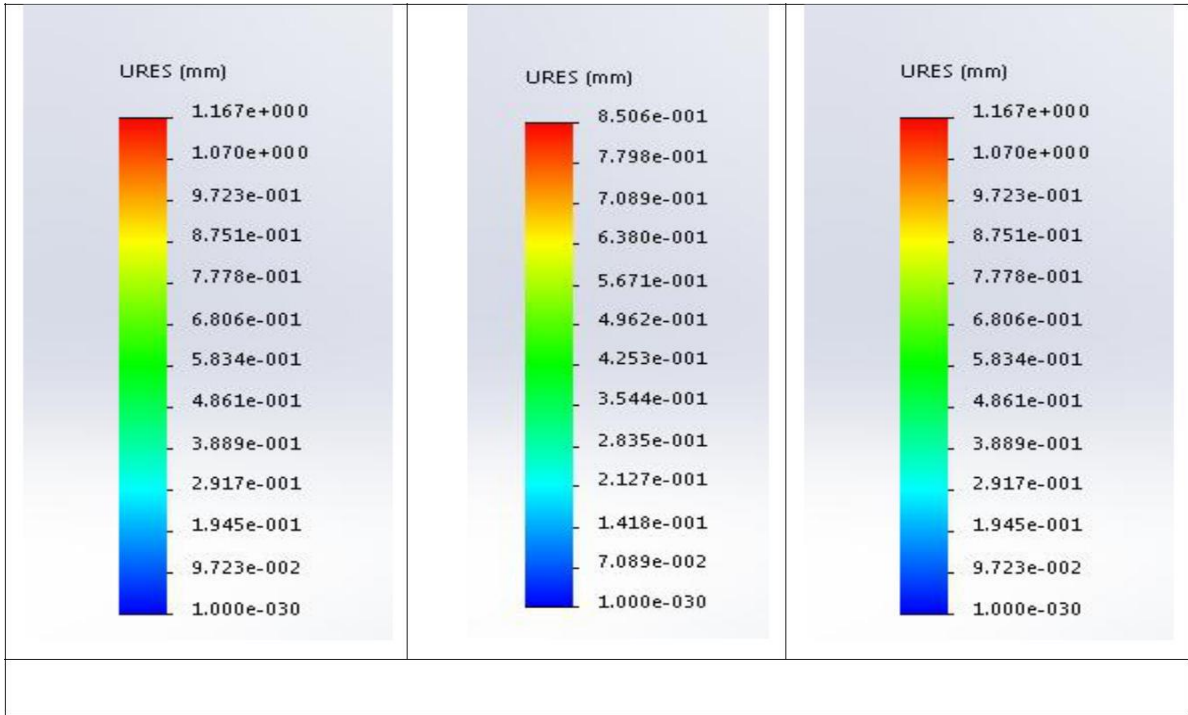


Figure 15 deflection with change section to rectangular, I, C-sections respectively.-Deflection under thermal load using solid work by change support.

The following assumptions will be used to draw the beam and help to put a load as quantity and direction as well as determine the type of material for simulation:

- Moment of inertia is $I=5.27 \times 10^{-6} \text{mm}^4$
- beam length is 3 m.
- Reference temperature 25 c°
- applied temperature at 60 c°
- AISI 1020 steel cold rolled

-total node for:	-mesh type: solid mesh
rectangular (81811)	-total element for:
I-section (69332)	rectangular (53007)
c-section (97965)	I-section (35923)
	c-section (58183)

Table 5 relationship between change support and deflection for rectangular section .

Support	Pine-pine	fixed-fixed	Roller-fixed	Roller-pine	cantilever
Maximum deflection(mm)	-1.167	-0.04194	-0.05058	-0.03788	-4.718

Compare between the percentage of deflection for types of support for rectangular section as shown figure 16.

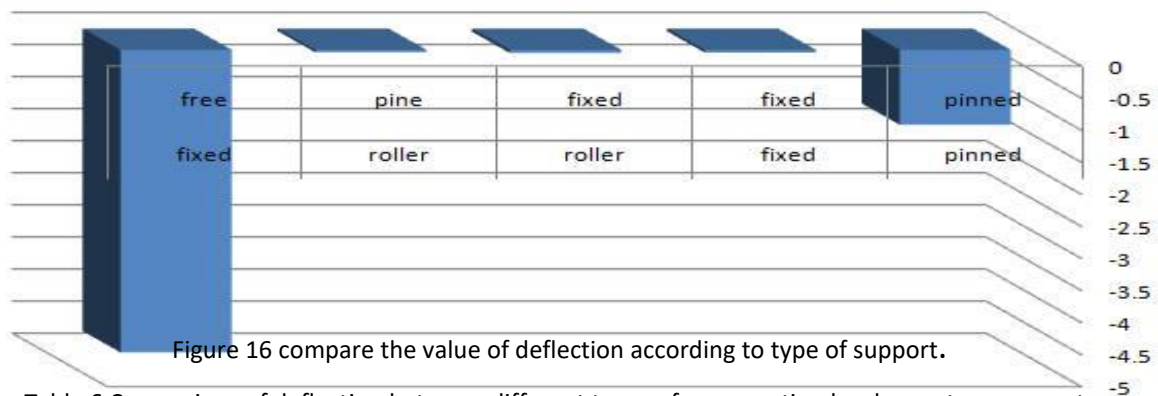


Figure 16 compare the value of deflection according to type of support.

Table 6 Comparison of deflection between different types of cross section by change type support.

■ deflection rectangular

Type of support	Pinned-pinned	Fixed-fixed	Roller-fixed	Roller-pine	cantilever
Deflection for rectangular section (mm)	-1.167	-0.04194	-0.05058	-0.03788	-4.718
Deflection for I-section (mm)	-0.85	-0.1558	-0.164	-0.539	-3.61
Deflection for c-section (mm)	-1.23	-0.1218	-0.369	-2.31	-2.26

For comparison the deflection by change type of support between rectangular section, I-section and c- section as shown figure 17.

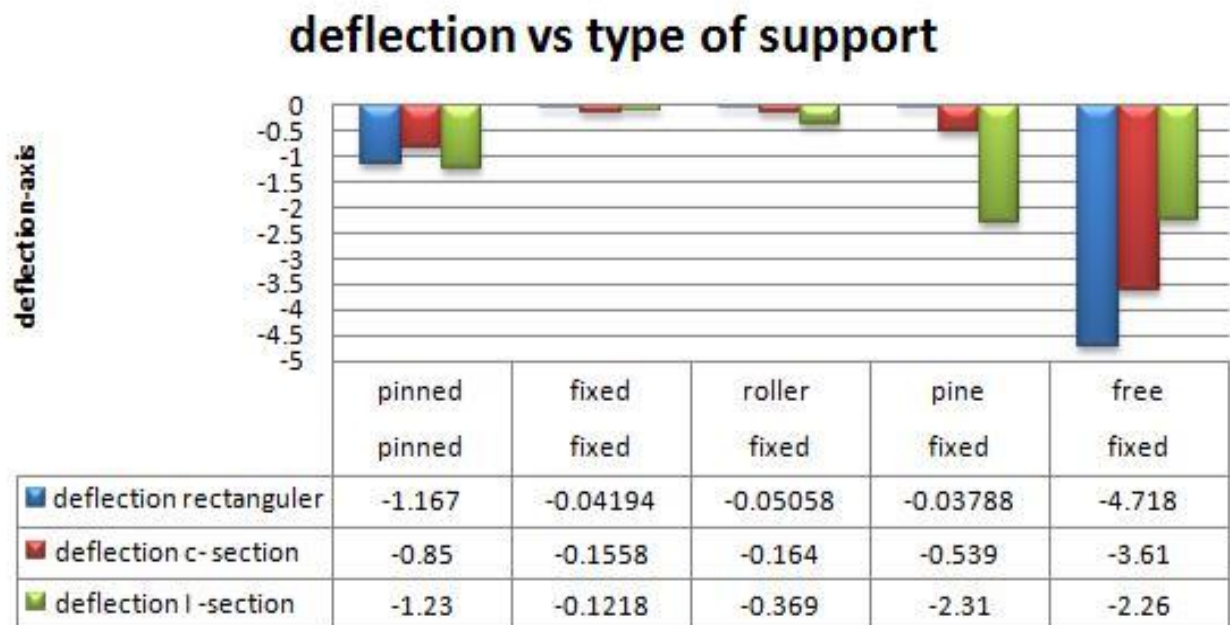


Figure 17 compare the value of deflection according to type of support.

This model is drawn on the solid-work as shown figure 18, it represents deflection which result from change support to cantilever for rectangular section

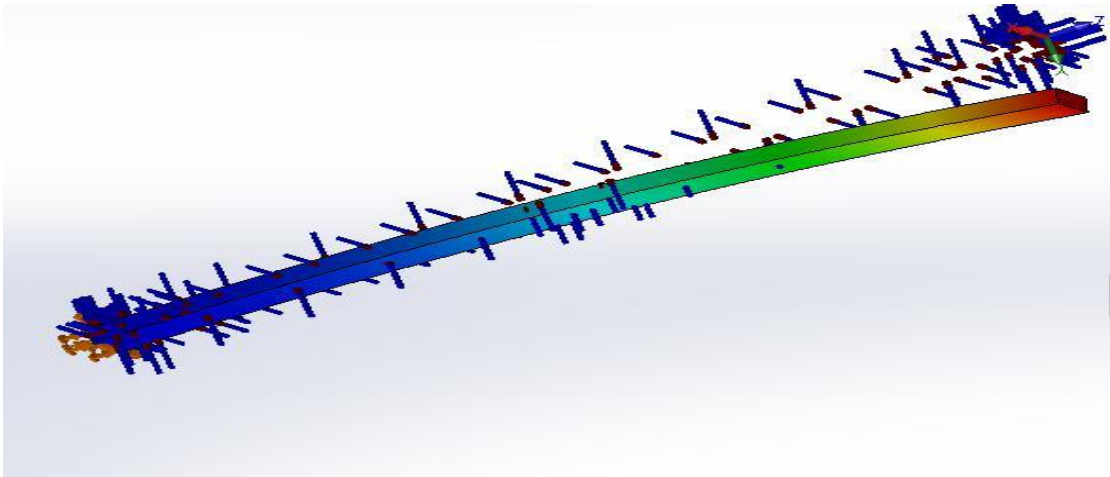
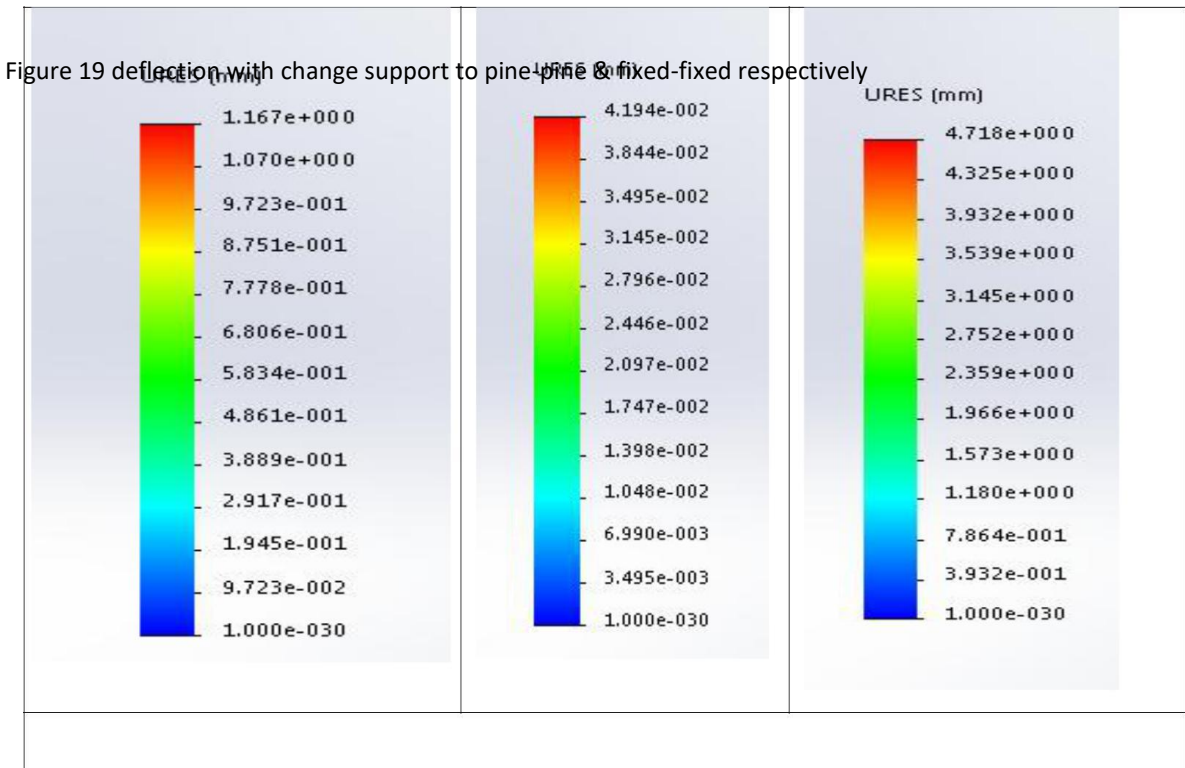


Figure 18 deflection profile which results from change support

Figure 19 & figure 20 show the values of deflection resulting from changing the type of support for beam for rectangular section.



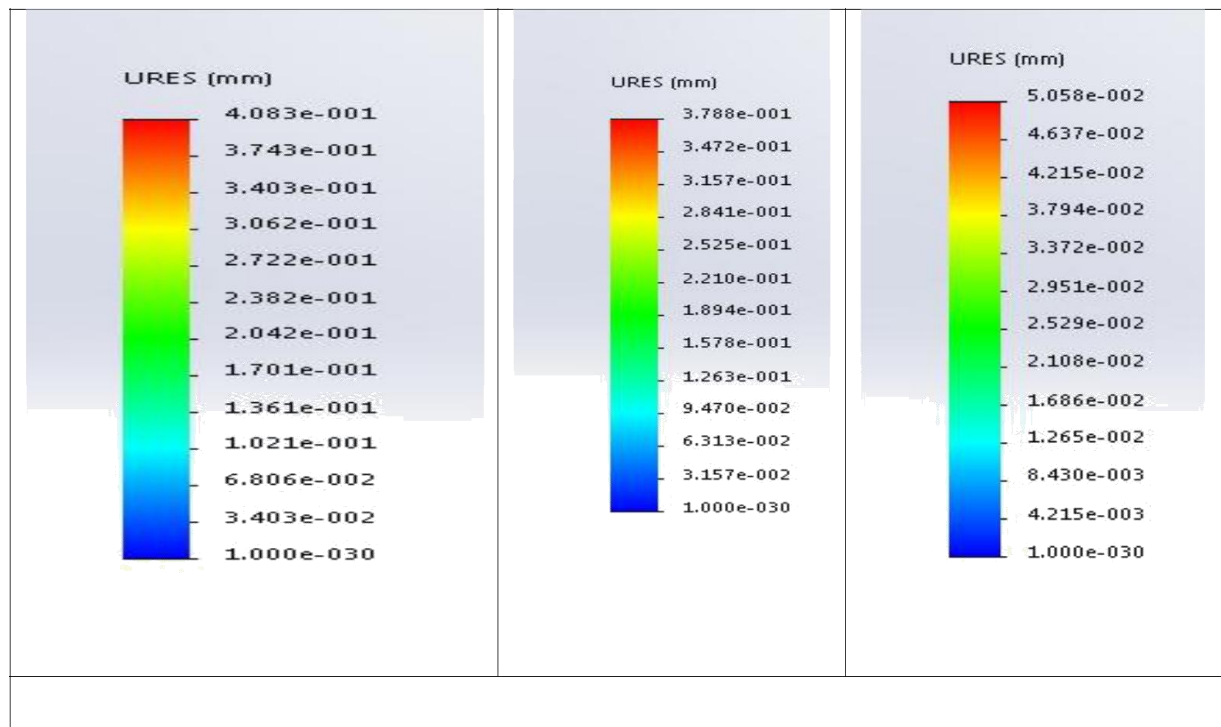


Figure 20 deflection with change support to pinned-roller, fixed-roller & cantilever respectively

Conclusions

Through the comparison process between sections turns out to be the weakest section was c-section and the best I-section by stabilizing of concentrated load in the middle of beam. The consolidation process is a collection process for deflection the result from mechanical load and thermal load with a slight difference of values. This process does not affect on value of deflection mechanical and thermal load both separately. In case of beam refraction due to load the program cannot solve the equation and find the value that broke the beam.

Reference

- [1] Jung-Hoon Kim, Seung-Jo Kim, and Seung-Jo Kim (2022). "Overlapping Finite Element Analysis for Structures Under Thermal Loads". *Journal of Mechanical Science and Technology*, Volume 36, Issue 9, 2022
- [2] J. M. R. S. Tavares and J. M. R. S. Tavares (1983). "Improved Finite Element Methodology for Integrated Thermal-Structural Analysis". *NASA Technical Reports*, 1983
- [3] J. M. R. S. Tavares and J. M. R. S. Tavares (2021). "Thermo-Mechanical Load Influence on Electronics Packages". *IEEE Transactions on Components, Packaging and Manufacturing Technology*, Volume 11, Issue 10, 2021.
- [4] M. Shamshirsaz, M. R. S. Tavares, and M. R. S. Tavares (2021). "A Thermal Stress Finite Element Analysis of Beam Structures by Hierarchical Modelling". *Computers, Materials & Continua*, Volume 66, Issue 3, 2021
- [5] M. Shamshirsaz, M. R. S. Tavares, and M. R. S. Tavares (2018). "Finite Element Analysis of the Effect of Thermo-Mechanical Loads on Buried Polyethylene Gas Pipes". *Energies*, Volume 11, Issue 10, 2018
- [6] P. O'Hara and V. Ucar (2013). "An Analysis of Thermal and Mechanical Loads on Piston Design". *Journal of Theoretical and Applied Information Technology*, Volume 48, Issue 2, 2013.

- [7] J. M. R. S. Tavares and J. M. R. S. Tavares (2008). "Methodology for Thermally Induced Loading in Aerospace Structures". *Proceedings of the 26th Congress of the International Council of the Aeronautical Sciences*, 2008
- [8] J. M. R. S. Tavares and J. M. R. S. Tavares (2011). "Finite Element Analysis on Mechanical Behaviors of a Steel-Concrete Hybrid Structure". *IEEE Transactions on Components, Packaging and Manufacturing Technology*, Volume 1, Issue 4, 2011
- [9] Nicolai Friedlich, Hanno Gottschalk, and Georg Vossen (2024). "Optimal Control of Thermal and Mechanical Loads in Activation Processes of Mechanical Components". *February 2024 Applied Mathematics in Science and Engineering* 32(1). DOI: 10.1080/27690911.2024.2313645.
- [10] Anne Le Pécheur, François Curtit, Michel Clavel, Jean-Michel Stephan, Colette Rey, and Philippe Bompard (2013). "Thermo-Mechanical Finite Element Model with Memory Effect for 304L Austenitic Stainless Steel Presenting Microstructure Gradient" *International Journal of Fatigue* 45. DOI: 10.1016/j.ijfatigue.2012.05.016
- [11] Ronald E. Barr -Davor Juricic-Thomas J. Krueger. (2015). *Engineering-Computer-Graphics-Workbook-Using SolidWorks*. SDC PUBLICATIONS
- [12] David Hutton (2004). *Fundamentals of finite element analysis*, McGraw Hill.
- [13] Randy H. Shih. *Learning SOLIDWORKS 2015*. SDC Publications.