Design and Analysis of Cantilever Beam

Sridhar Parshana, Manoj Kumar Dandu and Narendhar Kethavath

December 25, 2020

MallaReddy College of Engineering

Maisammaguda

Group 1

Group members

Parshana sridhar (17Q91A0334)

Dandu Manoj kumar (17Q91A0331)

Kethavath Narendhar (17Q91A0323)

Guide : V Ramya

MODELING AND FE ANALYSIS OF CANTILEVER BEAM

ABSTRACT

This study investigates the deflection and stress distribution in a long, slender cantilever beam of uniform rectangular cross section made of linear elastic material properties that are homogeneous and isotropic. The deflection of a cantilever beam is essentially a three dimensional problem. An elastic stretching in one direction is accompanied by a compression in perpendicular directions.

In this project ,static and Modal analysis is a process to determine the stress ,strain and deformation. vibration characteristics (natural frequencies and mode shapes) of a structure or a machine component while it is being designed. It has become a major alternative to provide a helpful contribution in understanding control of many vibration phenomena which encountered in practice.

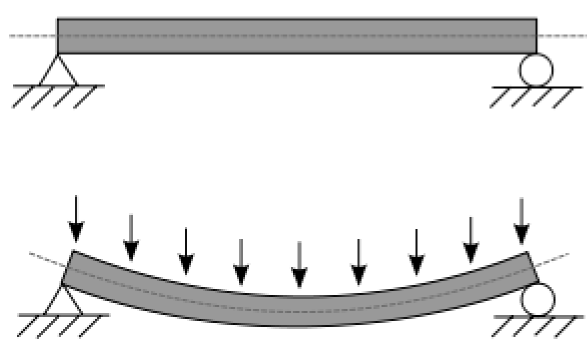
In this work we compared the stress and natural frequency for different material having same I, C and T cross- sectional beam. The cantilever beam is designed and analyzed in ANSYS. The cantilever beam which is fixed at one end is vibrated to obtain the natural frequency, mode shapes and deflection with different sections and materials.

INTRODUCTION

BEAM

A **beam** is a structural element that is capable of withstanding load primarily by resisting against bending. The bending force induced into the material of the beam as a result of the external loads, own weight, span and external reactions to these loads is called a bending moment. Beams are characterized by their profile (shape of cross-section), their length, and their material.

Beams are traditionally descriptions of building or civil engineering structural elements, but smaller structures such as truck or automobile frames, machine frames, and other mechanical or structural systems contain beam structures that are designed and analyzed in a similar fashion.



A statically determinate beam, bending (sagging) under a uniformly distributed load

Overview

Historically beams were squared timbers but are also metal, stone, or combinations of wood and metalsuch as a flitch beam. Beams generally carry vertical gravitational forces but can also be used to carry horizontal loads (e.g., loads due to an earthquake or wind or in tension to resist rafter thrust as a tie beam or (usually) compression as a collar beam). The loads carried by a beam are transferred to columns, walls, or girders, which then transfer the force to adjacent structural compression members. In light frame construction joists may rest on beams.

In carpentry a beam is called a plate as in a sill plate or wall plate, beam as in a summer beam or dragon beam.

CLASSIFICATION OF BEAMS BASED ON SUPPORTS

In engineering, beams are of several types:

1. Simply supported - a beam supported on the ends which are free to rotate and have no moment resistance.
2. Fixed - a beam supported on both ends and restrained from rotation.
3. Over hanging - a simple beam extending beyond its support on one end.
4. Double overhanging - a simple beam with both ends extending beyond its supports on both ends.
5. Continuous - a beam extending over more than two supports.
6. Cantilever - a projecting beam fixed only at one end.
7. Trussed - a beam strengthened by adding a cable or rod to form a truss.

Area moment of inertia

In the beam equation I is used to represent the second moment of area. It is commonly known as the moment of inertia, and is the sum, about the neutral axis, of dA*r^2, where r is the distance from the neutral axis, and dA is a small patch of area. Therefore, it encompasses not just how much area the beam section has overall, but how far each bit of area is from the axis, squared. The greater I is, the stiffer the beam in bending, for a given material.

Stress in beams

Internally, beams experience compressive, tensile and shear stresses as a result of the loads applied to them. Typically, under gravity loads, the original length of the beam is slightly reduced to enclose a smaller radius arc at the top of the beam, resulting in compression, while the same original beam length at the bottom of the beam is slightly stretched to enclose a larger radius arc, and so is under tension. The same original length of the middle of the beam, generally halfway between the top and bottom, is the same as the radial arc of bending, and so it is under neither compression nor tension, and defines the neutral axis (dotted line in the beam figure). Above the supports, the beam is exposed to shear stress. There are some reinforced concrete beams in which the concrete is entirely in compression with tensile forces taken by steel tendons. These beams are known as concrete beams, and are fabricated to produce a compression more than the expected tension under loading conditions. High strength steel tendons are stretched while the beam is cast over them. Then, when the concrete has cured, the tendons are slowly released and the beam is immediately under eccentric axial loads. This eccentric loading creates an internal moment, and, in turn, increases the moment carrying capacity of the beam. They are commonly used on highway bridges.

The primary tool for structural analysis of beams is the Euler–Bernoulli beam equation. Other mathematical methods for determining the deflection of beams include "method of virtual work" and the "slope deflection method". Engineers are interested in determining deflections because the beam may be in direct contact with a brittle material such as glass. Beam deflections are also minimized for aesthetic reasons. A visibly sagging beam, even if structurally safe, is unsightly and to be avoided. A stiffer beam (high modulus of elasticity and high second moment of area) produces less deflection.

Mathematical methods for determining the beam forces (internal forces of the beam and the forces that are imposed on the beam support) include the "moment distribution method", the force or flexibility method and the direct stiffness method.

General shapes

Most beams in reinforced concrete buildings have rectangular cross sections, but a more efficient cross section for a beam is an I or H section which is typically seen in steel construction. Because of the parallel axis theorem and the fact that most of the material is away from the neutral axis, the second moment of area of the beam increases, which in turn increases the stiffness.

An I-beam is only the most efficient shape in one direction of bending: up and down looking at the profile as an I. If the beam is bent side to side, it functions as an H where it is less efficient. The most efficient shape for both directions in 2D is a box (a square shell) however the most efficient shape for bending in any direction is a cylindrical shell or tube. But, for unidirectional bending, the I or wide flange beam is superior.

Efficiency means that for the same cross sectional area (volume of beam per length) subjected to the same loading conditions, the beam deflects less.

Other shapes, like L (angles), C (channels) or tubes, are also used in construction when there are special require ements

Thin walled beams

A thin walled beam is a very useful type of beam (structure). The cross section of *thin walled beams* is made up from thin panels connected among them to create closed or open cross sections of a beam (structure). Typical closed sections include round, square, and rectangular tubes. Open sections include I-beams, T-beams, L-beams, and so on. Thin walled beams exist because their bending stiffness per unit cross sectional area is much higher than that for solid cross sections such a rod or bar. In this way, stiff beams can be achieved with minimum weight. Thin walled beams are particularly useful when the material is a composite laminates. Pioneer work on composite laminates thin walled beams was done by Librescu.

CANTILEVER BEAM

A cantilever is a rigid structural element, such as a beam or a plate, anchored at only one end to a (usually vertical) support from which it is protruding. Cantilevers can also be constructed with trusses or slabs. When subjected to a structural load, the cantilever carries the load to the support where it is forced against by a moment and shear stress.

Cantilever construction allows for overhanging structures without external bracing, in contrast to constructions supported at both ends with loads applied between the supports, such as a simply supported beam found in a post and lintel system.

APPLICATIONS

In bridges, towers, and buildings

Cantilevers are widely found in construction, notably in cantilever bridges and balconies (see corbel). In cantilever bridges the cantilevers are usually built as pairs, with each cantilever used to support one end of a central section. The Forth Bridge in Scotland is an example of a cantilever truss bridge. A cantilever in a traditionally timber framed building is called a jetty or forebay. In the southern United States a historic barn type is the cantilever barn of log construction.

Temporary cantilevers are often used in construction. The partially constructed structure creates a cantilever, but the completed structure does not act as a cantilever. This is very helpful when temporary supports, or falsework, cannot be used to support the structure while it is being built (e.g., over a busy roadway or river, or in a deep valley). So some truss arch bridges (see Navajo Bridge) are built from each side as cantilevers until the spans reach each other and are then jacked apart to stress them in compression before final joining. Nearly all cable-stayed bridges are built using cantilevers as this is one of their chief advantages. Many box girder bridges are built segmentally, or in short pieces. This type of construction lends itself well to balanced cantilever construction where the bridge is built in both directions from a single support.

These structures are highly based on torque and rotational equilibrium.

In an architectural application, Frank Lloyd Wright's Fallingwater used cantilevers to project large balconies. The East Stand at Elland Road Stadium in Leeds was, when completed, the largest cantilever stand in the world[2] holding 17,000 spectators. The roof built over the stands at Old Trafford Football Ground uses a cantilever so that no supports will block views of the field. The old, now demolished Miami Stadium had a similar roof over the spectator area. The largest cantilever in Europe is located at St

James' Park in Newcastle-Upon-Tyne, the home stadium of Newcastle United F.C.

Less obvious examples of cantilevers are free-standing (vertical) radio towers without guy-wires, and chimneys, which resist being blown over by the wind through cantilever action at their base.

ADVANTAGES AND DISADVANTAGES

Advantages

- Does not require a support on the opposite side (probably the main reason you would ever have a cantilever beam).
- Creates a negative bending moment, which can help to counteract a positive bending moment created elsewhere. This is particular helpful in cantilevers with a backspanwhere a uniform load on the backspan creates positive bending, but a uniform load on the cantilever creates negative bending.

Disadvantages

- Large deflections
- Generally results in larger moments
- You either need to have a fixed support, or have a backspan and check for uplift of the far support.

INTRODUCTION TO CAD

Throughout the history of our industrial society, many inventions have been patented and whole new technologies have evolved. Perhaps the single development that has impacted manufacturing more quickly and significantly than any previous technology is the digital computer.

Computers are being used increasingly for both design and detailing of engineering components in the drawing office. Computer-aided design (CAD) is defined as the application of computers and graphics software to aid or enhance the product design from conceptualization to documentation. CAD is most commonly associated with the use of an interactive computer graphics system, referred to as a CAD system. Computer -aided design systems are powerful tools and in the mechanical design and geometric modeling of products and components.

There are several good reasons for using a CAD system to support the engineering design function:

- To increase the productivity
- To improve the quality of the design
- To uniform design standards
- To create a manufacturing data base
- To eliminate inaccuracies caused by hand-copying of drawings and inconsistency between
- Drawings

INTRODUCTION TO CREO

PTC CREO, formerly known as Pro/ENGINEER, is 3D modeling software used in mechanical engineering, design, manufacturing, and in CAD drafting service firms. It was one of the first 3D CAD modeling applications that used a rule-based parametric system. Using parameters, dimensions and features to capture the behavior of the product, it can optimize the development product as well as the design itself.

The name was changed in 2010 from Pro/ENGINEER Wildfire to CREO. It was announced by the company who developed it, Parametric Technology Company (PTC), during the launch of its suite of design products that includes applications such as assembly modeling, 2D orthographic views for technical drawing, finite element analysis and more.

PTC CREO says it can offer a more efficient design experience than other modeling software because of its unique features including the integration of parametric and direct modeling in one platform. The complete suite of applications spans the spectrum of product development, giving designers options to use in each step of the process. The software also has a more user friendly interface that provides a better experience for designers. It also has collaborative capacities that make it easy to share designs and make changes.

There are countless benefits to using PTC CREO. We'll take a look at them in this two-part series.

First up, the biggest advantage is increased productivity because of its efficient and flexible design capabilities. It was designed to be easier to use and have features that allow for design processes to move more quickly, making a designer's productivity level increase.

Part of the reason productivity can be increased is because the package offers tools for all phases of development, from the beginning stages to the hands-on creation and manufacturing. Late stage changes are common in the design process, but PTC CREO can handle it. Changes can be made that are reflected in other parts of the process.

The collaborative capability of the software also makes it easier and faster to use. One of the reasons it can process information more quickly is because of the interface between MCAD and ECAD designs. Designs can be altered and highlighted between the electrical and mechanical designers working on the project.

The time saved by using PTC CREO isn't the only advantage. It has many ways of saving costs. For instance, the cost of creating a new product can be lowered because the development process is shortened due to the automation of the generation of associative manufacturing and service deliverables.

PTC also offers comprehensive training on how to use the software. This can save businesses by eliminating the need to hire new employees. Their training program is available online and in-person, but materials are available to access anytime.

A unique feature is that the software is available in 10 languages. PTC knows they have people from all over the world using their software, so they offer it in multiple languages so nearly anyone who wants to use it is able to do so.

ADVANTAGES OF CREO PARAMETRIC SOFTWARE

1. Optimized for model-based enterprises
2. Increased engineer productivity
3. Better enabled concept design
4. Increased engineering capabilities
5. Increased manufacturing capabilities
6. Better simulation
7. Design capabilities for additive manufacturing

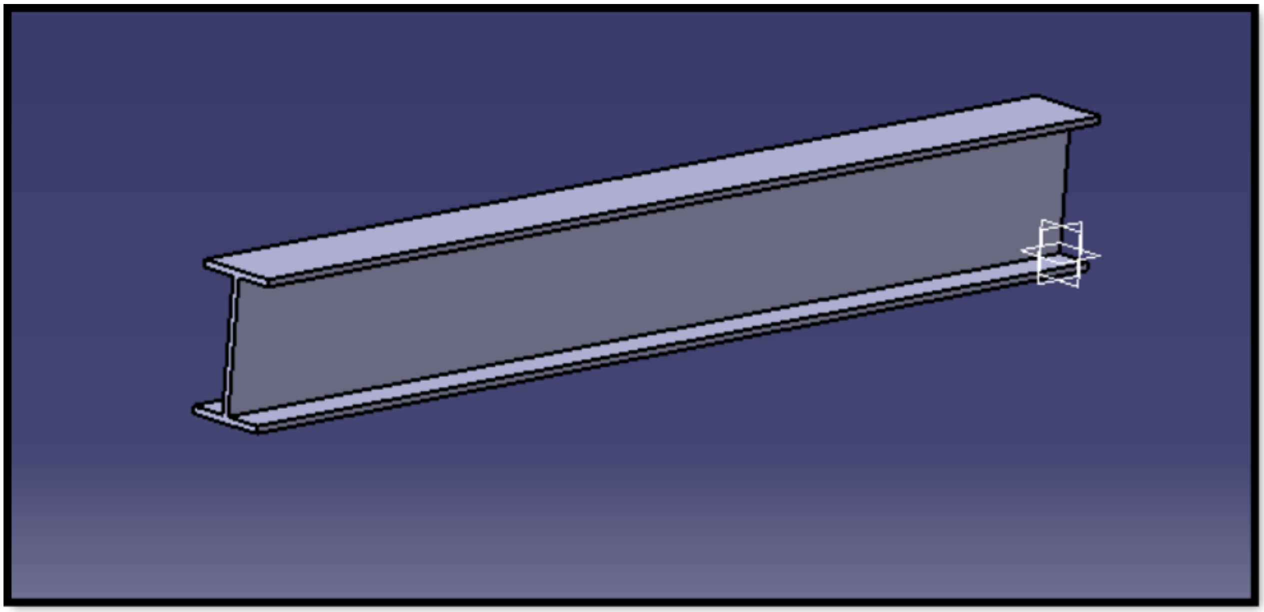
CREO parametric modules:

- Sketcher
- Part modeling
- Assembly

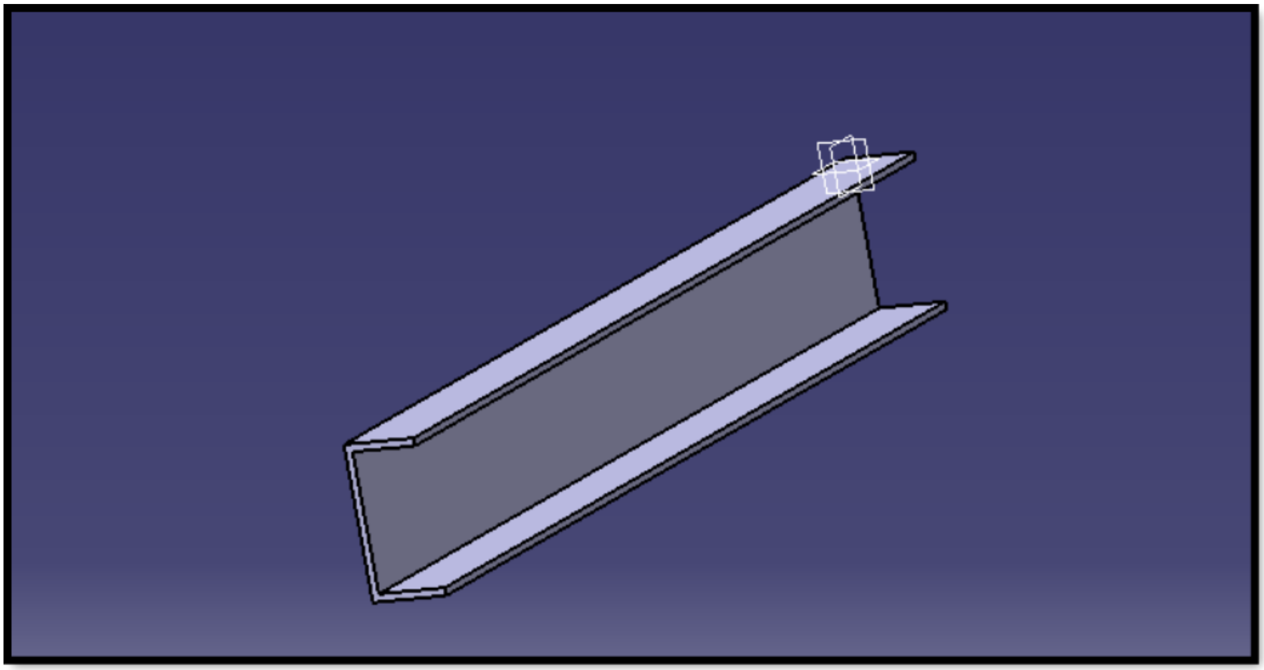
- Drafting

3d model

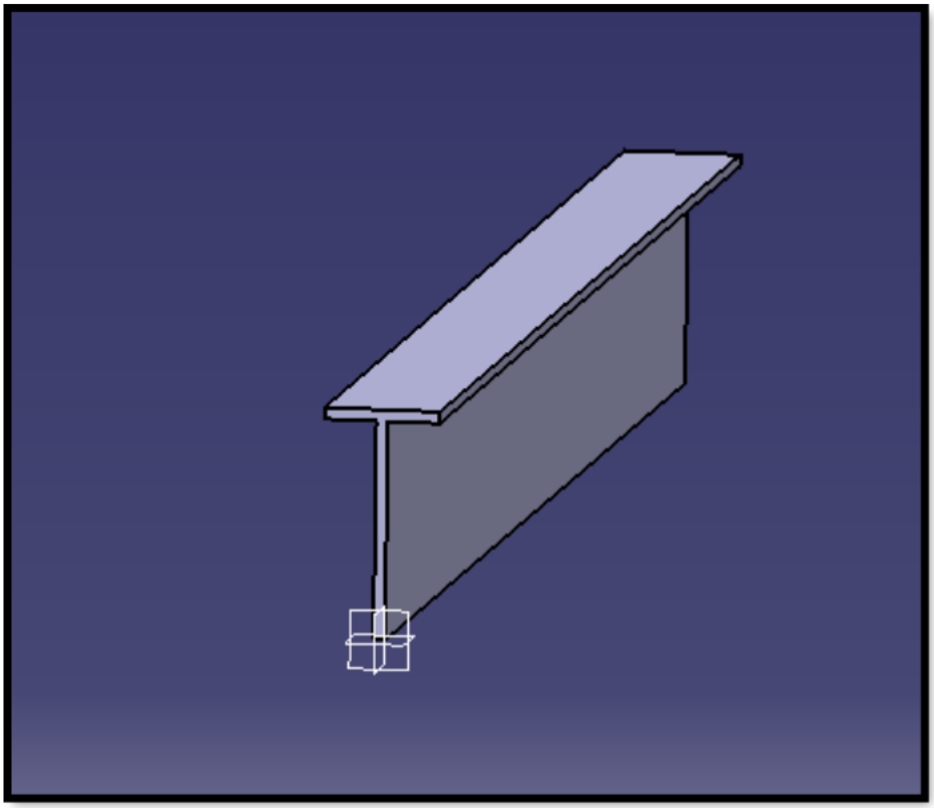
I-section



C –section



T –section



INTRODUCTION TO FEM

The Basic concept in FEA is that the body or structure may be divided into smaller elements of finite dimensions called “Finite Elements”. The original body or the structure is then considered as an assemblage of these elements connected at a finite number of joints called “Nodes” or “Nodal Points”. Simple functions are chosen to approximate the displacements over each finite element. Such assumed functions are called “shape functions”. This will represent the displacement with in the element in terms of the displacement at the nodes of the element.

The Finite Element Method is a mathematical tool for solving ordinary and partial differential equations. Because it is a numerical tool, it has the ability to solve the complex problems that can be represented in differential equations form. The applications of FEM are limitless as regards the solution of practical design problems.

Due to high cost of computing power of years gone by, FEA has a history of being used to solve complex and cost critical problems. Classical methods alone usually cannot provide adequate information to determine the safe working limits of a major civil engineering construction or an automobile or an aircraft. In the recent years, FEA has been universally used to solve structural engineering problems. The departments, which are heavily relied on this technology, are the automotive and aerospace industry. Due to the need to meet the extreme demands for faster, stronger, efficient and lightweight automobiles and aircraft, manufacturers have to rely on this technique to stay competitive.

FEA has been used routinely in high volume production and manufacturing industries for many years, as to get a product design wrong would be detrimental. For example, if a large manufacturer had to recall one model alone due to a hand brake design fault, they would end up having to replace up to few millions of hand brakes. This will cause a heavier loss to the company.

The finite element method is a very important tool for those involved in engineering design; it is now used routinely to solve problems in the following areas.

Structural analysis

- Thermal analysis

- Vibrations and Dynamics
- Buckling analysis
- Acoustics
- Fluid flow simulations
- Crash simulations
- Mold flow simulations

Nowadays, even the most simple of products rely on the finite element method for design evaluation. This is because contemporary design problems usually can not be solved as accurately & cheaply using any other method that is currently available. Physical testing was the norm in the years gone by, but now it is simply too expensive and time consuming also.

INTRODUCTION TO ANSYS

The ANSYS program is self contained general purpose finite element program developed and maintained by Swason Analysis Systems Inc. The program contain many routines, all inter related, and all for main purpose of achieving a solution to an an engineering problem by finite element method.

ANSYS finite element analysis software enables engineers to perform the following tasks:

- Build computer models or transfer CAD models of structures, products, components, or systems.
- Apply operating loads or other design performance conditions
- Study physical responses ,such as stress levels, temperature distributions, or electromagnetic fields
- Optimize a design early in the development process to reduce production costs.
- Do prototype testing in environments where it otherwise would be undesirable or impossible

The ANSYS program has a compressive graphical user interface (GUI) that gives users easy, interactive access to program functions, commands, documentation, and reference material. An intuitive menu system helps users navigate through the ANSYS Program. Users can input data using a mouse, a keyboard, or a combination of both. A graphical user interface is available throughout the program, to guide new users through the learning process and provide more experienced users with multiple windows, pull-down menus, dialog boxes, tool bar and online documentation.

STRUCTURAL ANALYSIS

Static analysis calculates the effects of steady loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time-varying loads. A static analysis, however, includes steady inertia loads (such as gravity and rotational velocity), and time-varying loads that can be approximated as static equivalent loads (such as the static equivalent wind and seismic loads commonly defined in many building codes).

LOADS IN A STRUCTURAL ANALYSIS

Static analysis is used to determine the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed; that is, the loads and the structure's response are assumed to vary slowly with respect to time. The kinds of loading that can be applied in a static analysis include:

- Externally applied forces and pressures
- Steady-state inertial forces (such as gravity or rotational velocity)
- Imposed (non-zero) displacements
- Temperatures (for thermal strain)
- Fluences (for nuclear swelling)

MODAL ANALYSIS

Any physical system can vibrate. The frequencies at which vibration naturally occurs, and the modal shapes which the vibrating system assumes are properties of the system, and can be determined analytically using Modal Analysis.

Modal analysis is the procedure of determining a structure's dynamic characteristics; namely, resonant frequencies, damping values, and the associated pattern of structural deformation called mode shapes. It also can be a starting point for another, more detailed, dynamic analysis, such as a transient dynamic analysis, a harmonic response analysis, or a spectrum analysis.

Modal analysis in the ANSYS family of products is a linear analysis. Any nonlinearities, such as plasticity and contact (gap) elements, are ignored even if they are defined. Modal analysis can be done through several mode extraction methods: subspace, Block Lanczos, Power Dynamics, Reduced, Unsymmetrical and Damped. The damped method allows you to include damping in the structure.

USES OF MODAL ANALYSIS

Modal analysis is used to determine the natural frequencies and mode shapes of a structure. The natural frequencies and mode shapes are important parameters in the design of a structure for dynamic loading conditions. They are also required to do a spectrum analysis or a mode superposition harmonic or transient analysis. Another useful feature is modal cyclic symmetry, which allows reviewing the mode shapes of a cyclically symmetric structure by modeling just a sector of it.

RANDOM VIBRATION ANALYSIS

A Random Vibration Analysis is a form of Spectrum Analysis.

•The spectrum is a graph of spectral value versus frequency that captures the intensity and frequency content of time-history loads.

•Random vibration analysis is probabilistic in nature, because both input and output quantities represent only the probability that they take on certain values

Random Vibration Analysis uses Power spectral density to quantify the loading.

• (PSD) is a statistical measure defined as the limiting mean-square value of a random variable. It is used in random vibration analyses in which the instantaneous magnitudes of the response can be specified only by probability distribution functions that show the probability of the magnitude taking a particular value.

STRUCTURAL ANALYSIS OF CANTILEVER BEAM

CONDITION 1- I-SECTION

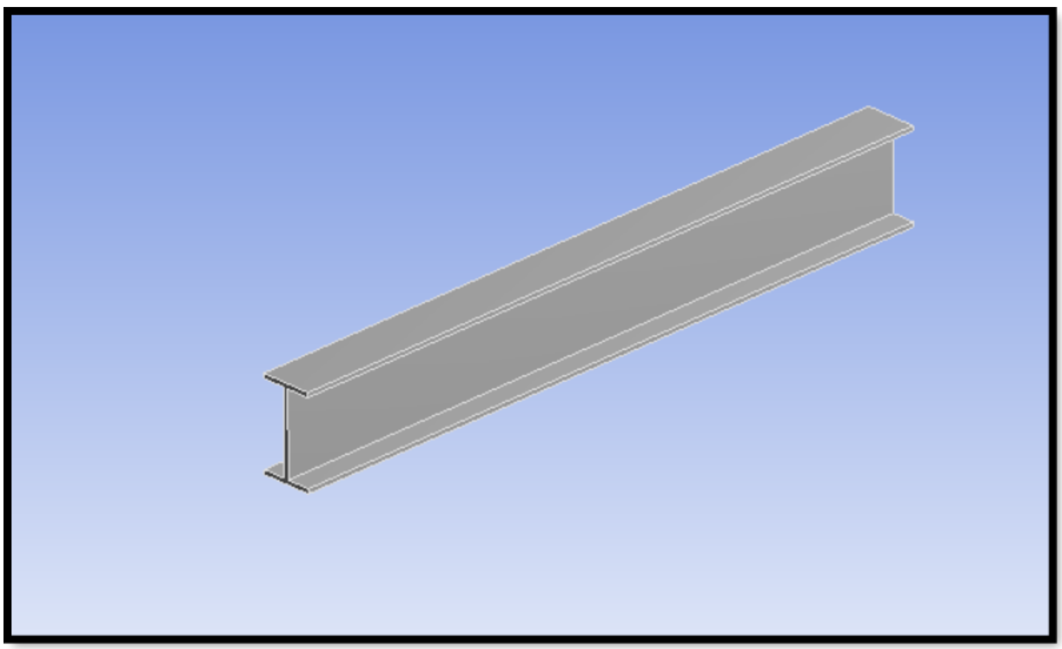
MATERIAL - STEEL

Save Creo Model as .iges format

→ → Ansys → Workbench → Select analysis system → static structural → double click

→ → Select geometry → right click → import geometry → select browse → open part → ok

→ → select mesh on work bench → right click → edit



Double click on geometry → select geometries → edit material

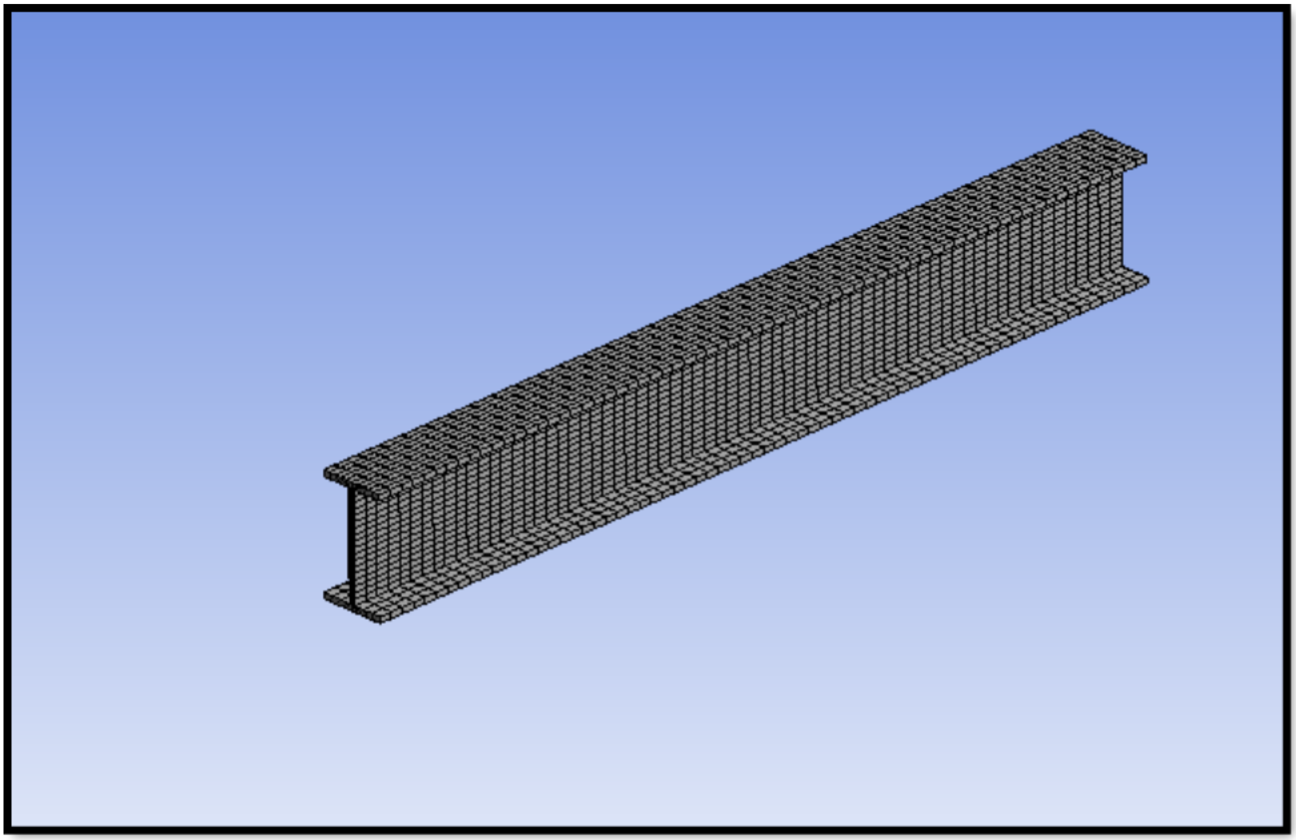
MATERIAL PROPERTIES OF STEEL

Density : 7850kg/m^3

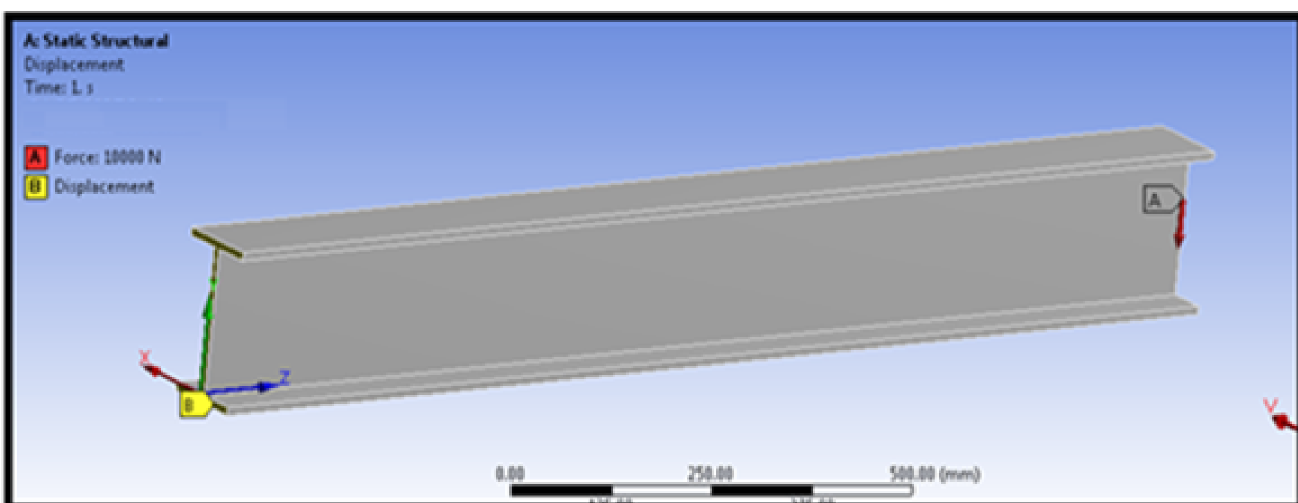
Young's modulus : 205000Mpa

Poisson's ratio : 0.3

Select mesh on left side part tree → right click → generate mesh →



Select static structural right click → insert → select pressure –0.096MPa



Select displacement → select required area → click on apply →

Select solution right click → solve →

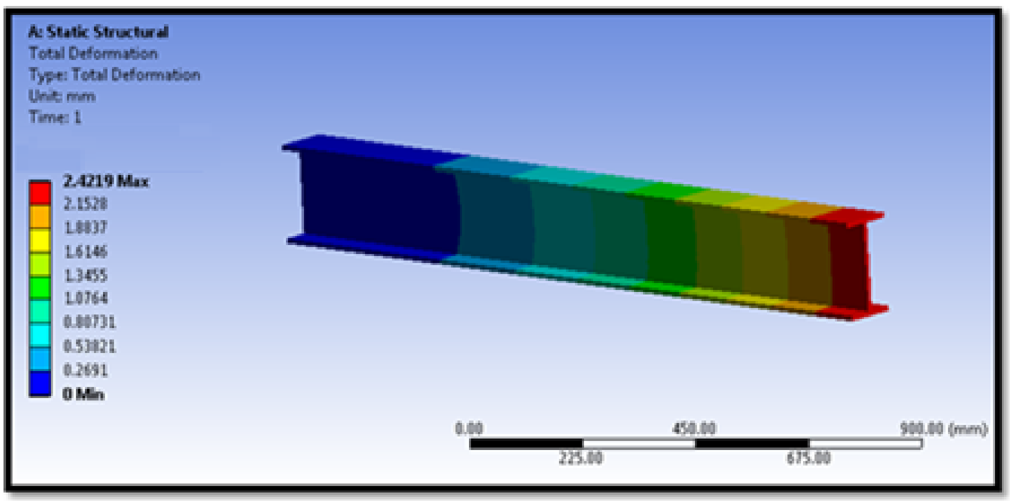
Solution- right click → insert → deformation → total

Solution right click → insert → strain → equivant (von-mises) →

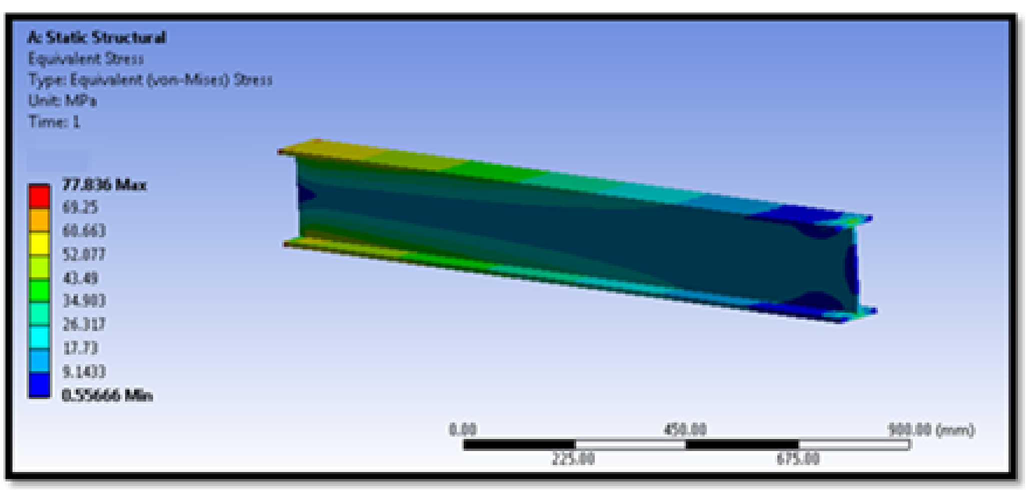
Solution right click → insert → stress → equivant (von-mises) →

Right click on deformation → evaluate all result

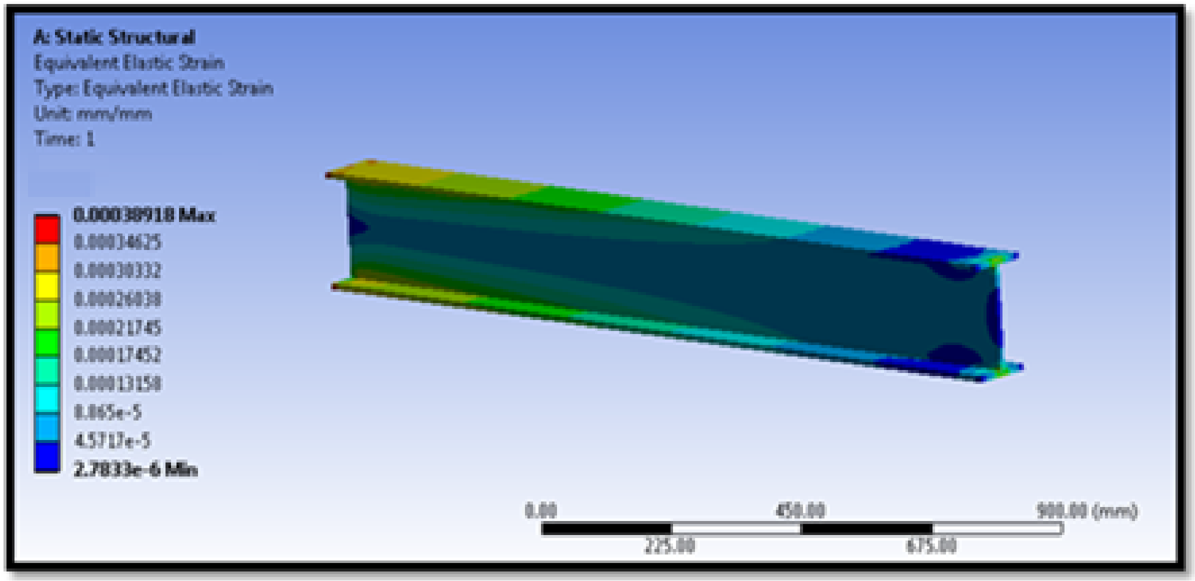
TOTAL DEFORMATION



VON-MISES STRESS

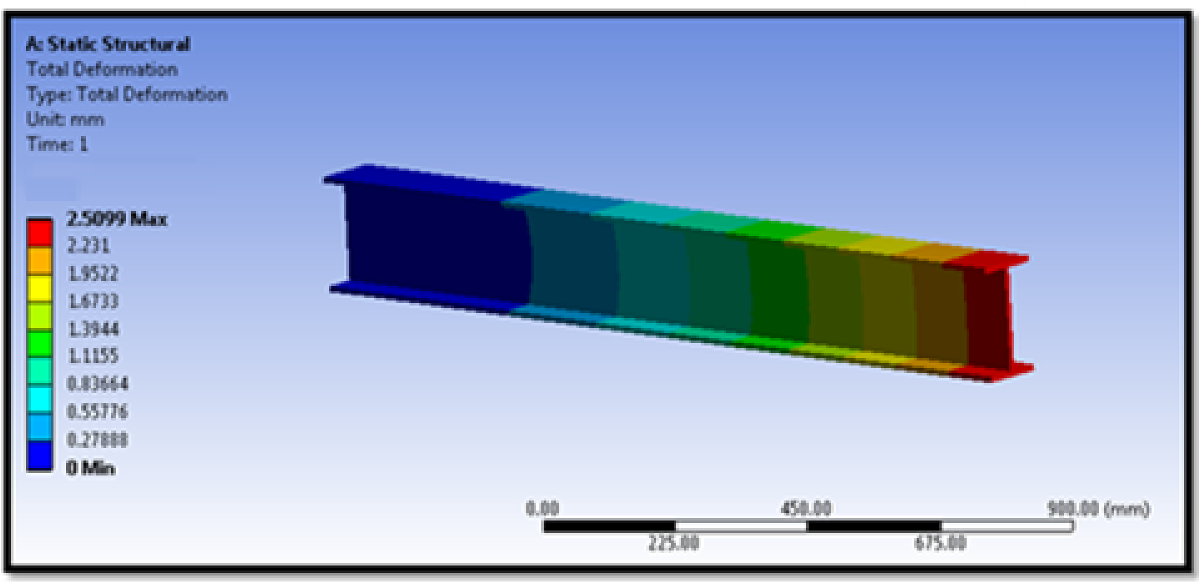


VON-MISES STRAIN

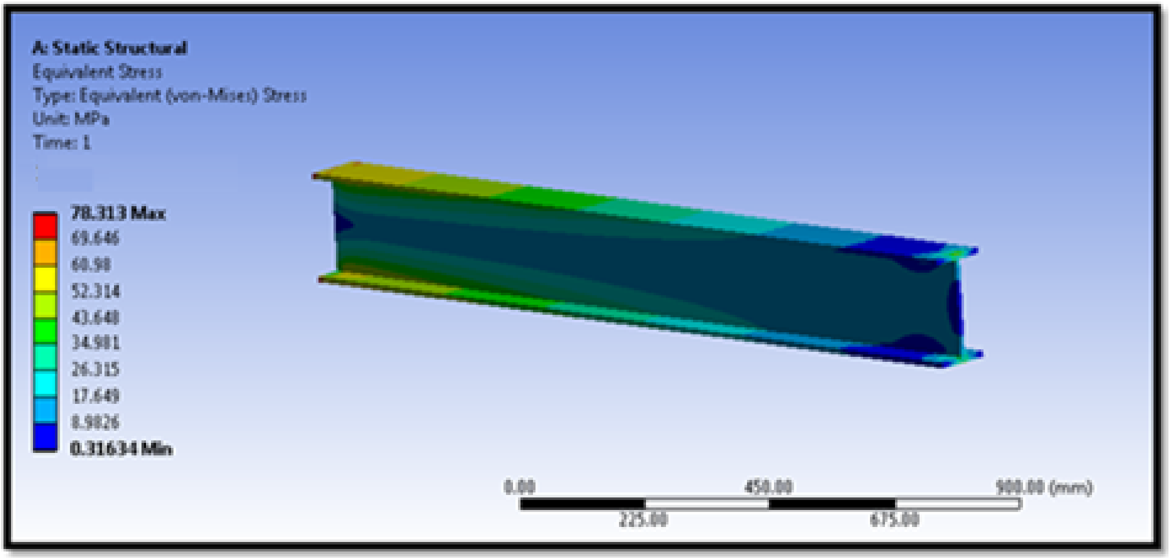


MATERIAL - STAINLESS STEEL

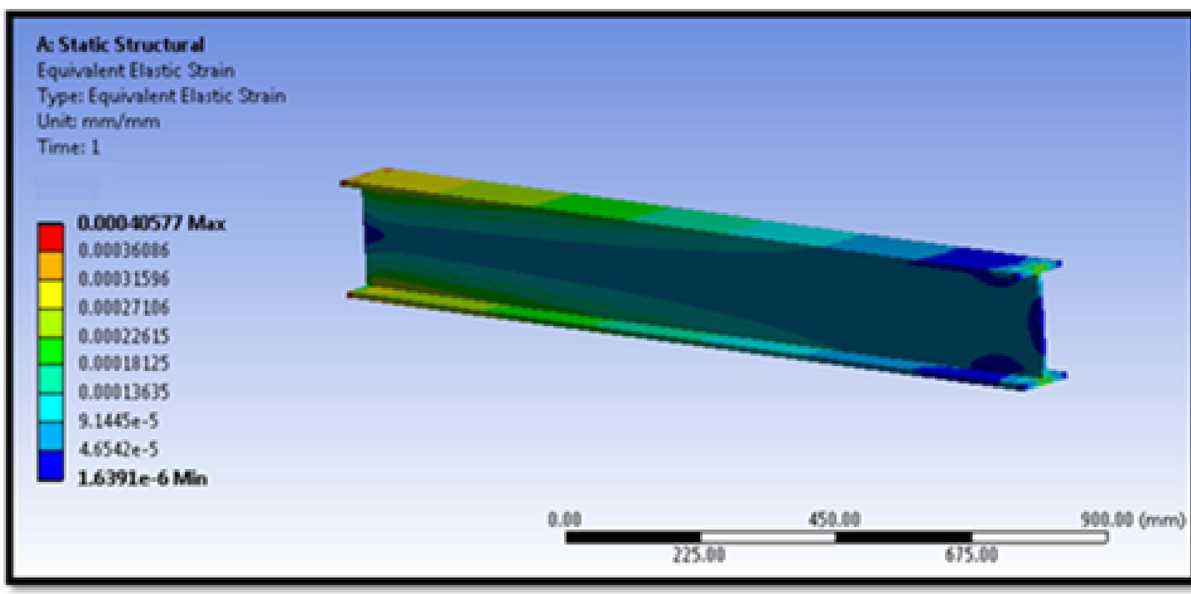
TOTAL DEFORMATION



VON-MISES STRESS



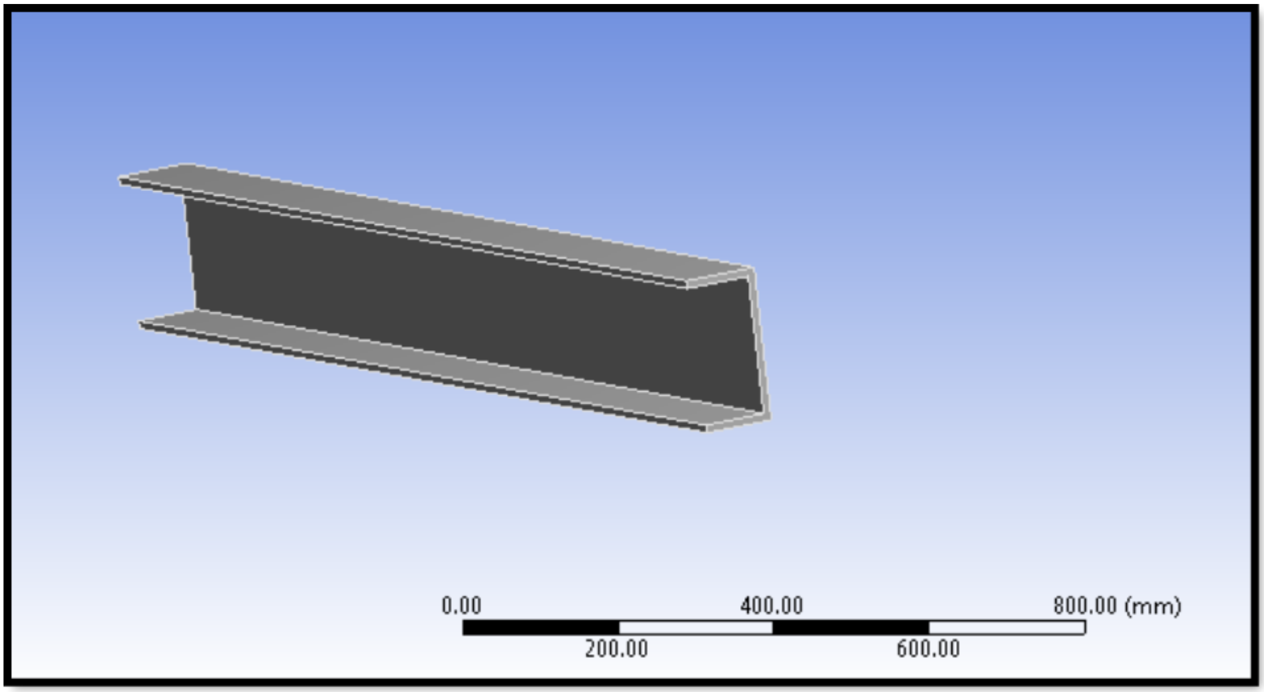
VON-MISES STRAIN



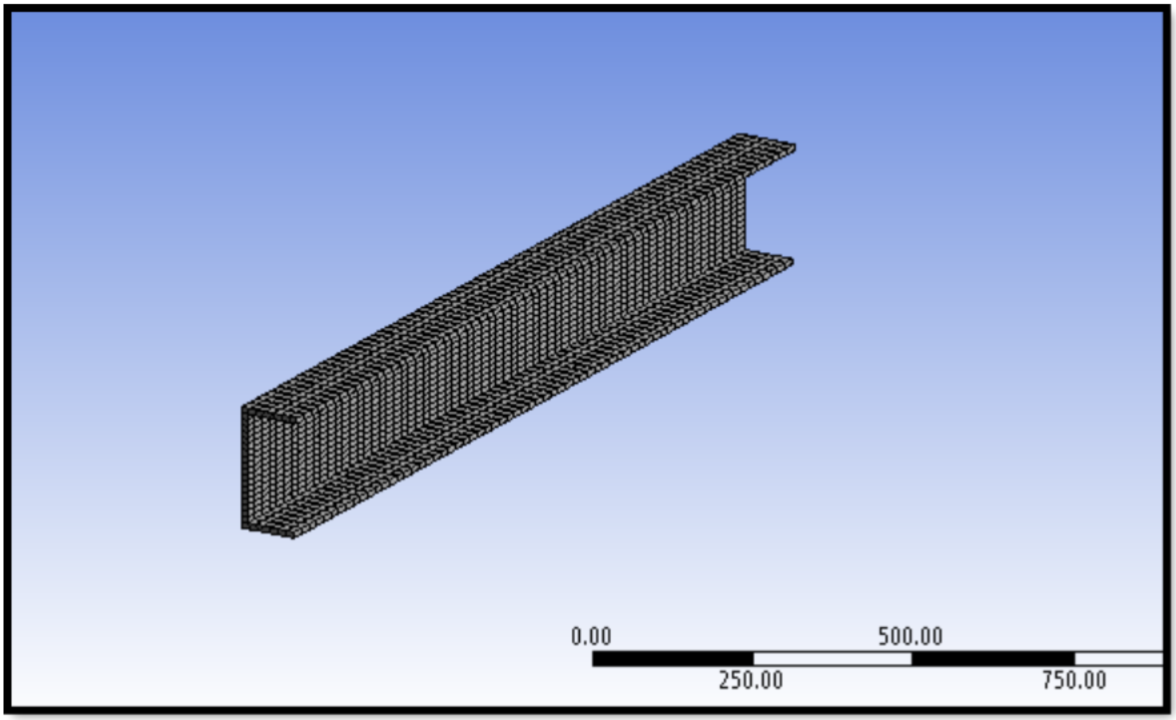
CONDITION 2- C-SECTION

MATERIAL - STEEL

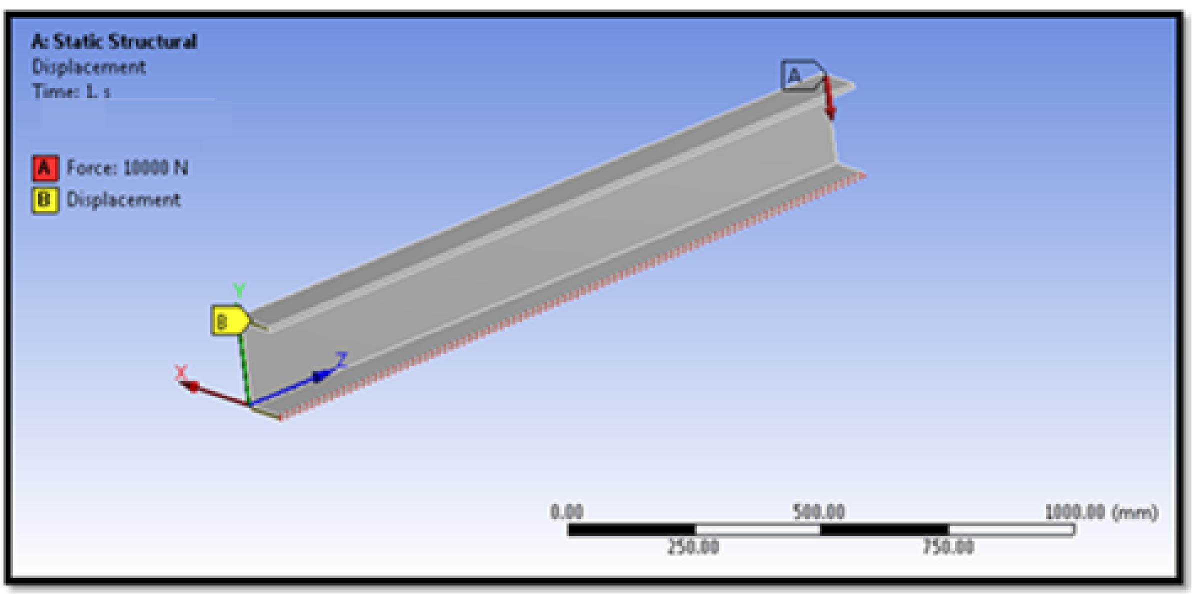
Imported model



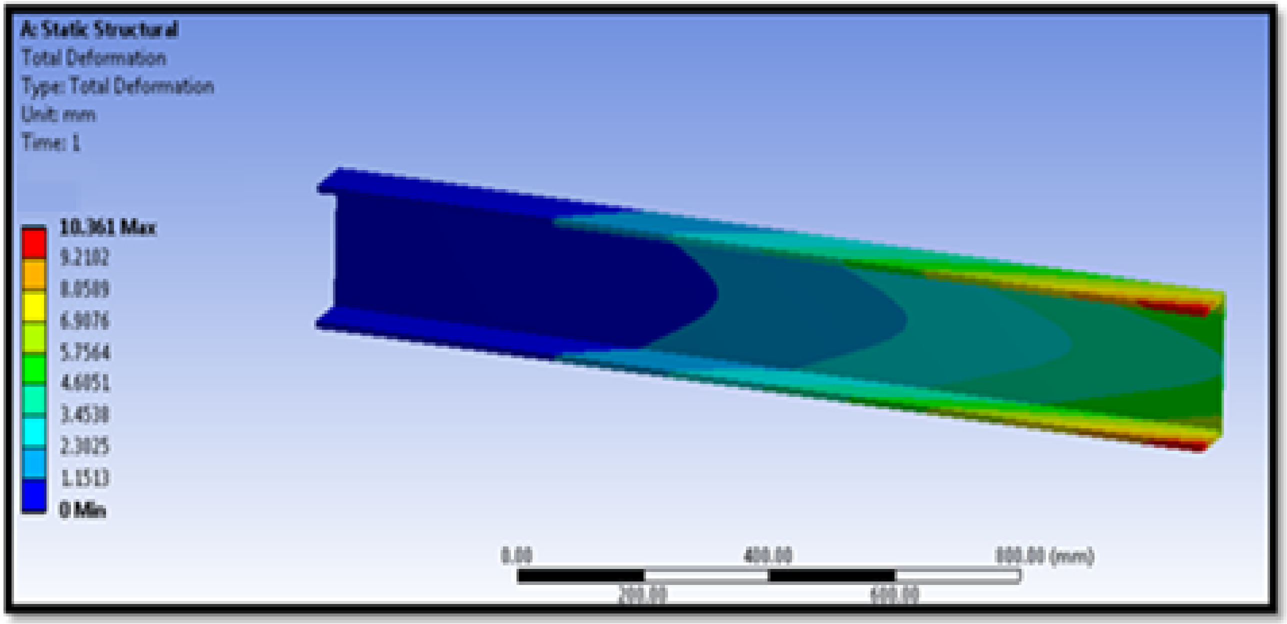
Meshed model



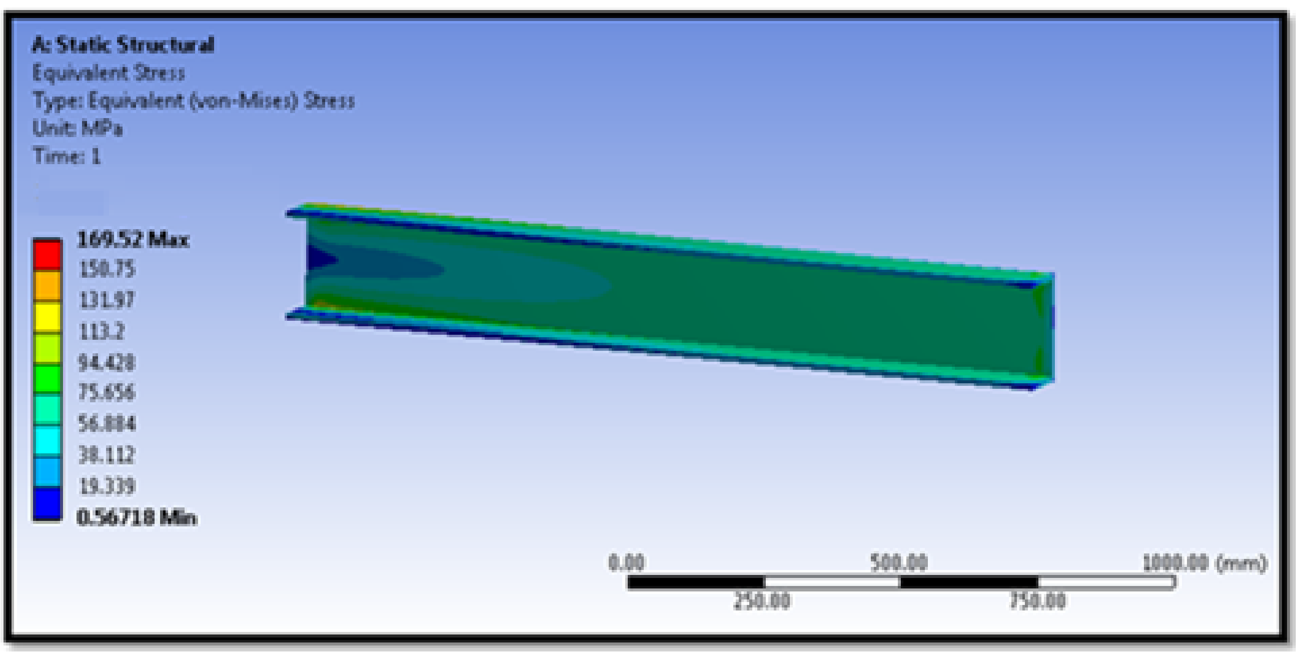
Force &displacement



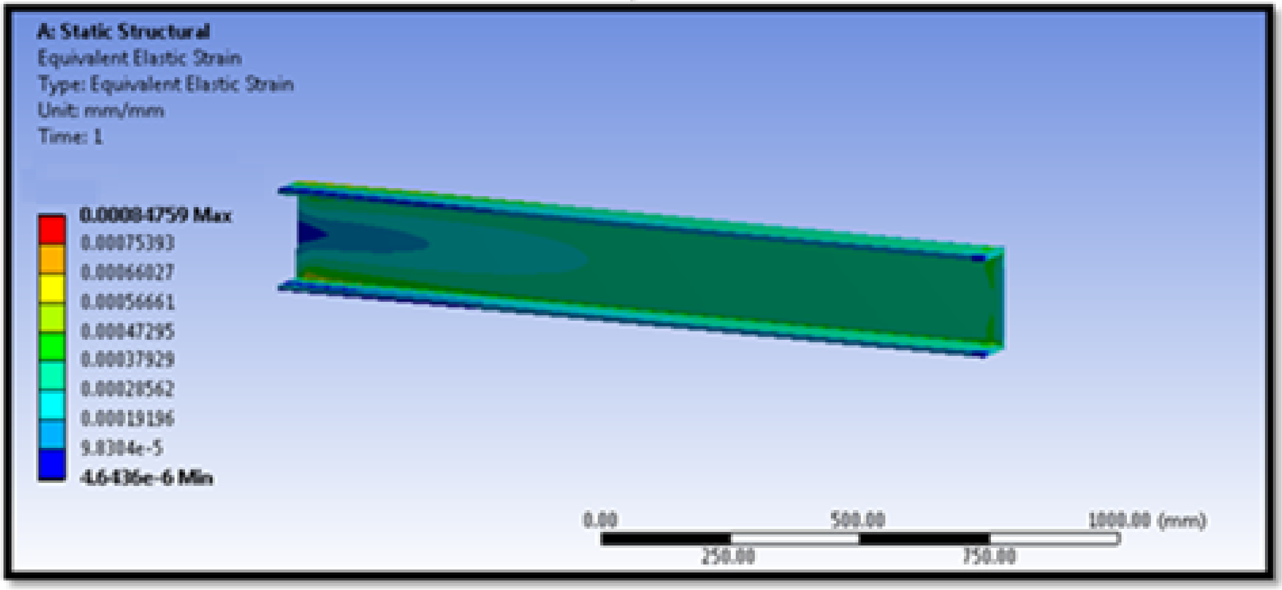
TOTAL DEFORMATION



VON-MISES STRESS



VON-MISES STRAIN



MODAL ANALYSIS OFCANTILEVER BEAM

CONDITION 1- I-SECTION

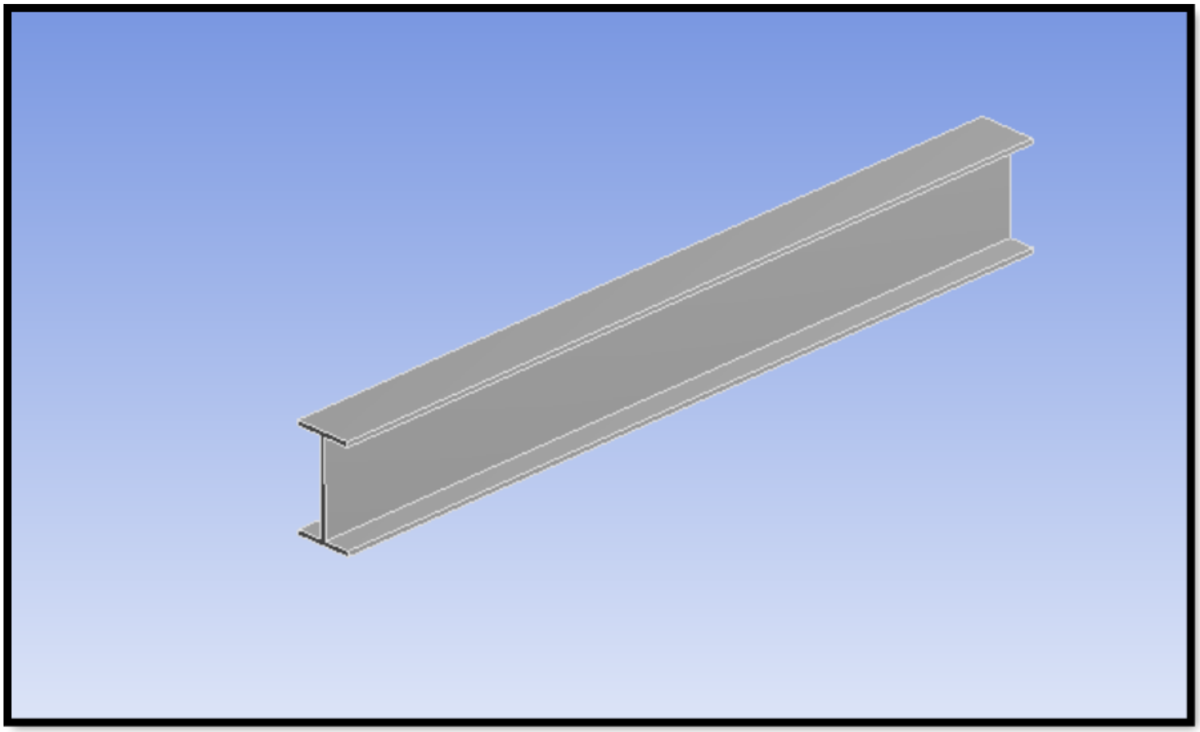
MATERIAL - STEEL

Save Creo Model as .iges format

→ →Ansys → Workbench→ Select analysis system → model → double click

→ →Select geometry → right click → import geometry → select browse →open part → ok

→ →Select modal → right click →select edit → another window will be open



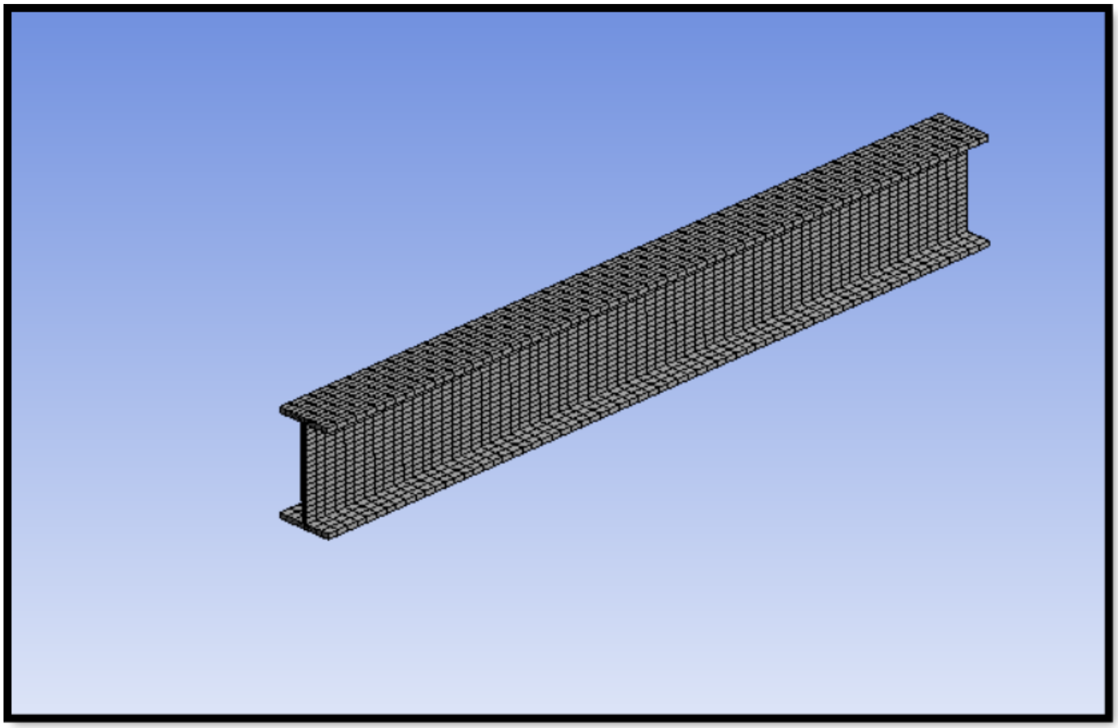
MATERIAL PROPERTIES OF STEEL

Density : 7850kg/m^3

Young’s modulus : 205000Mpa

Poisson’s ratio : 0.3

Select mesh on left side part tree → right click → generate mesh →

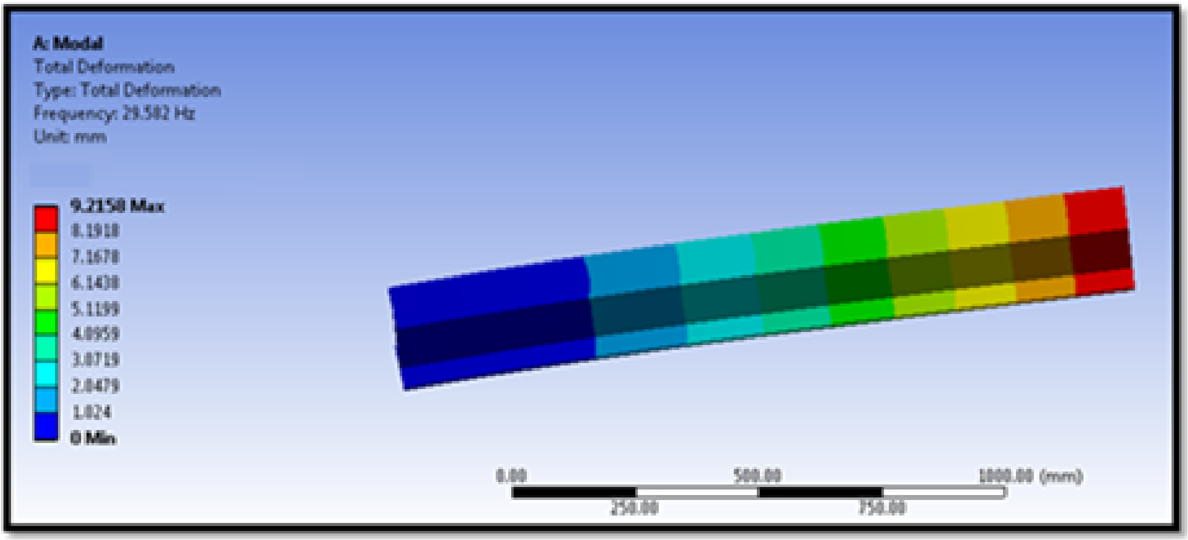


Select displacement → select required area → click on apply → Select solution right click →

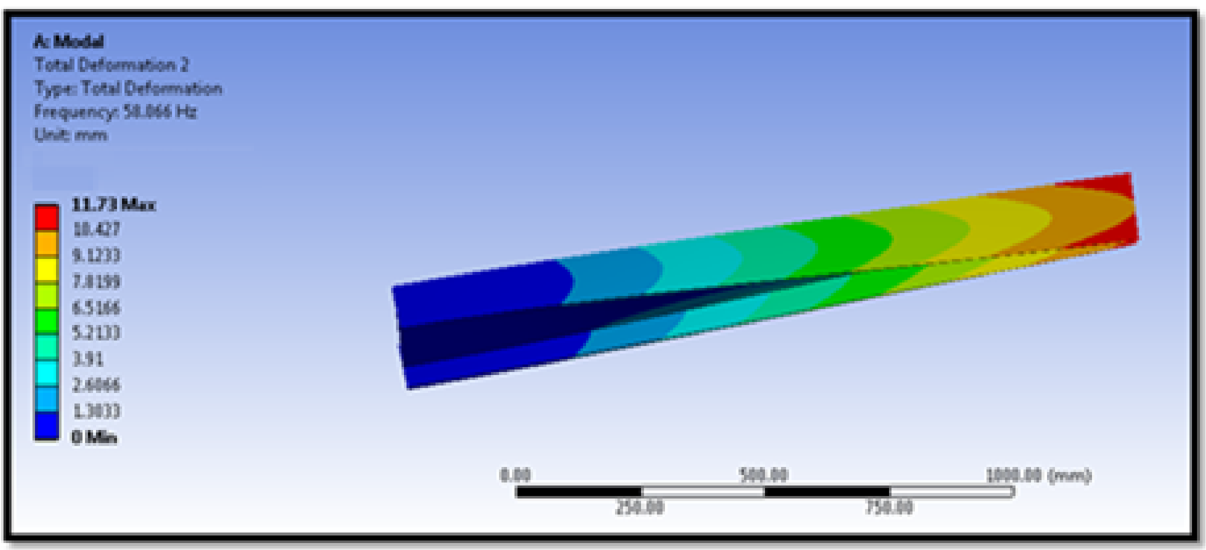
solve →

Solution right click → insert → deformation → total deformation → mode 1
Solution right click → insert → deformation → total deformation2 → mode 2
Solution right click → insert → deformation → total deformation 3 → mode 3
Right click on deformation → evaluate all result

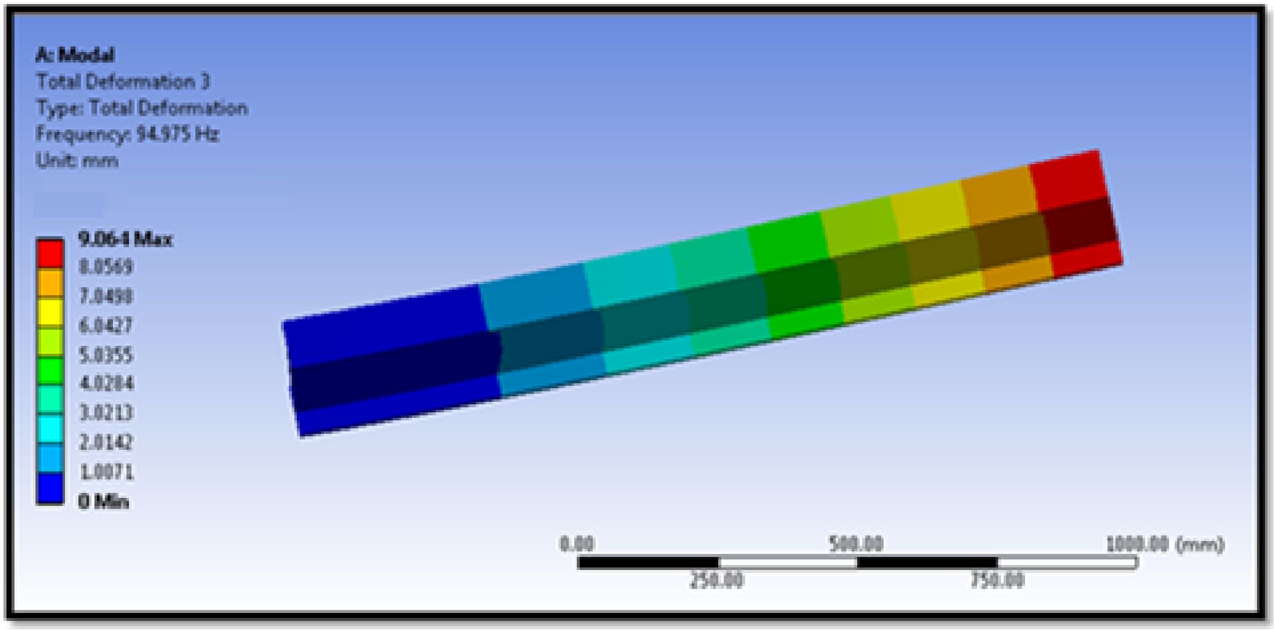
TOTAL DEFORMATION 1:



TOTAL DEFORMATION 2

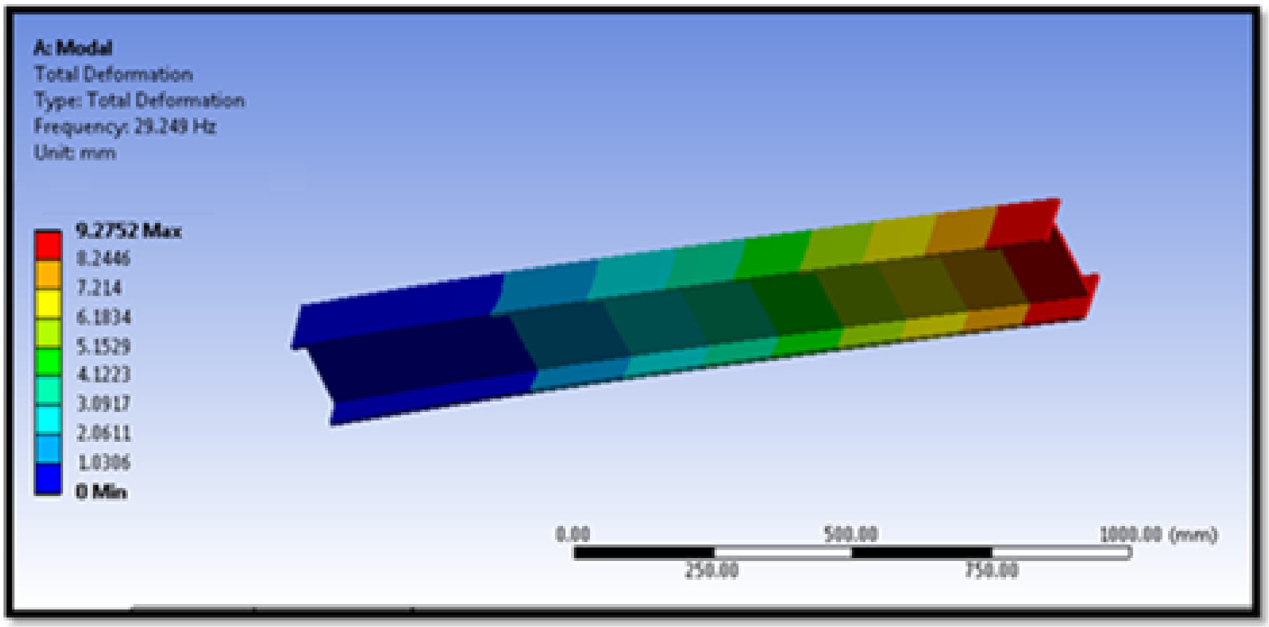


TOTAL DEFORMATION 3

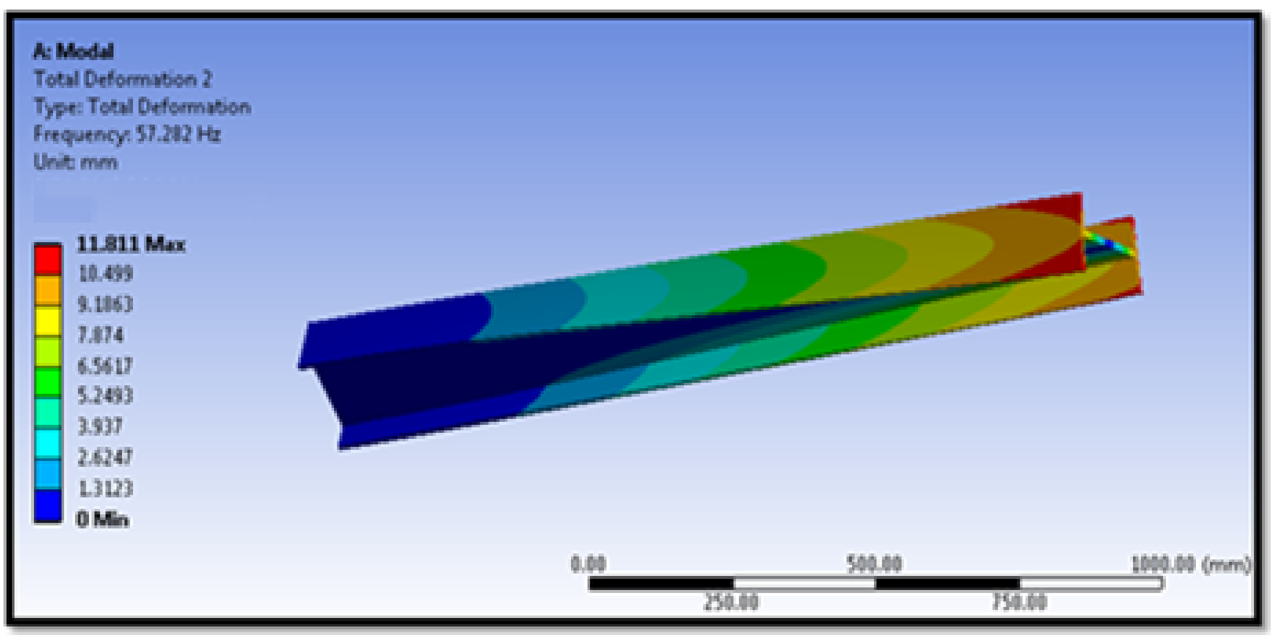


MATERIAL – STAINLESS STEEL

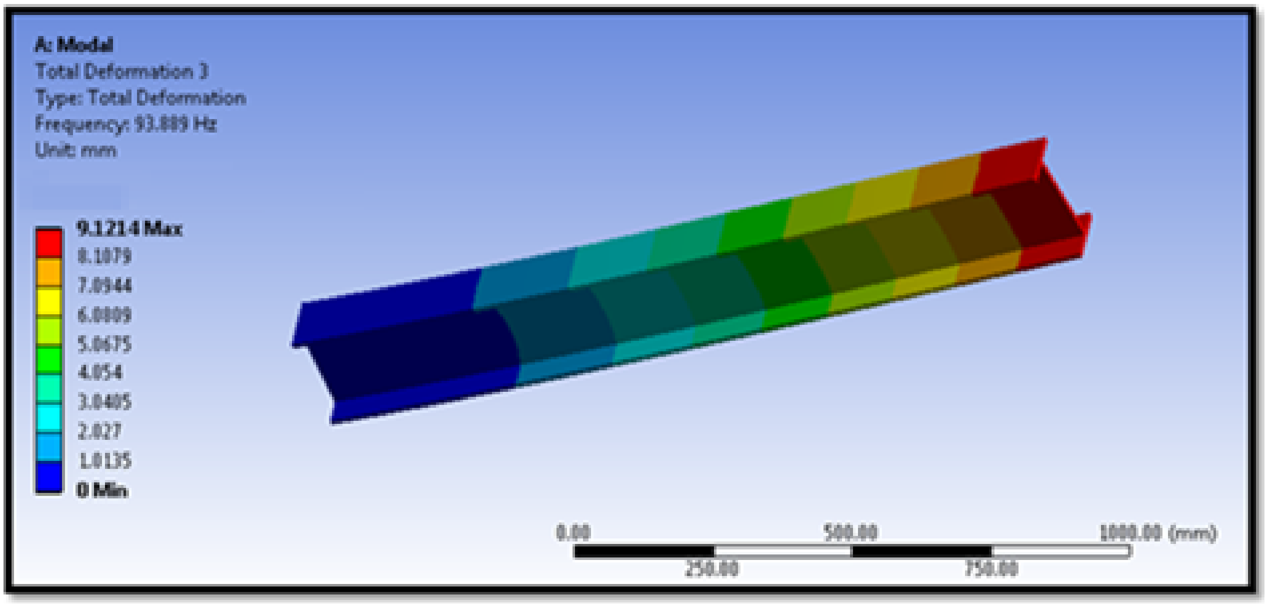
TOTAL DEFORMATION 1



TOTAL DEFORMATION 2



TOTAL DEFORMATION 3


A: Modal
Total Deformation 3
Type: Total Deformation
Frequency: 93.889 Hz
Unit: mm
9.1214 Max
8.1079
7.0944
6.0809
5.0675
4.054
3.0405
2.027
1.0135
0 Min
0.00
500.00
1000.00 (mm)
250.00
750.00

RESULT TABLES

Static analysis results

I-section

Material	Deformation(mm)	Stress (N/mm^2)	Strain
Steel	2.4219	77.836	0.00038918
Stainless steel	2.5099	78.313	0.00040577
Cast iron	4.4028	76.871	0.00069883

C-section

Material	Deformation(mm)	Stress (N/mm^2)	Strain
Steel	10.361	169.52	0.00084759
Stainless steel	10.778	170.35	0.00088267
Cast iron	18.696	167.77	0.0015251

T-section

Material	Deformation(mm)	Stress (N/mm^2)	Strain
Steel	4.7314	207.08	0.0010354
Stainless steel	4.9027	208.22	0.0010789
Cast iron	8.6034	204.75	0.0018614

Modal analysis results

Material	Total Deformation 1(mm)	Total Deformation 2(mm)	Total Deformation 3(mm)
Steel	9.2158	11.7	9.06
Stainless steel	9.2752	11.8	9.12
Cast iron	9.6225	12.23	9.466

C-section

Material	Total Deformation 1(mm)	Total Deformation 2(mm)	Total Deformation 3(mm)
Steel	9.9427	14.092	11.479
Stainless steel	10.012	14.192	11.553
Cast iron	10.37	14.691	11.987

T-section

Material	Total Deformation	Total Deformation	Total Deformation

	1(mm)	2(mm)	3(mm)
Steel	13.151	13.465	10.829
Stainless steel	13.053	13.37	10.76
Cast iron	13.599	13.941	11.236

CONCLUSION

In this work we compared the stress and natural frequency for different material having same I, C and T cross- sectional beam. The cantilever beam is designed and analyzed in ANSYS. The cantilever beam which is fixed at one end is vibrated to obtain the natural frequency, mode shapes and deflection with different sections and materials.

By observing the static analysis the deformation and stress values are less for I-section cantilever beam at cast iron material than steel and stainless steel.

By observing the modal analysis results the deformation and frequency values are less for I-section cantilever beam more for T-section.

So it can be conclude the cast iron material is better material for cantilever beam in this type I-section model.

REFERENCES

[1] Chandradeep Kumar, Anjani Kumar Singh, Nitesh Kumar, Ajit Kumar, "Model Analysis and Harmonic Analysis of Cantilever Beam by ANSYS" Global journal for research analysis, 2014, Volume-3, Issue-9, PP:51- 55.

[2] Yuan, F.G. and R.E. Miller. A higher order finite element for laminated composite beams. Computers & Structures, 14 (1990): 125-150.

[3] Dipak Kr. Maiti& P. K. Sinha. Bending and free vibration analysis of shear deformable laminated composite beams by finite element method. Composite Structures, 29 (1994): 421- 431

[4] Teboub Y, Hajela P. Free vibration of generally layered composite beams using symbolic computations. Composite Structures, 33 (1995): 123– 34.

[5] Banerjee, J.R. Free vibration of axially loaded composite Timoshenko beams using the dynamic stiffness matrix method. Computers & Structures, 69 (1998): 197-208 [6] Bassiouni AS, Gad-Elrab RM, Elmahdy TH. Dynamic analysis for laminated composite beams. Composite Structures, 44 (1999): 81–7.