

ANALYSIS OF CANTILEVER STEEL BEAM

SITI NURUL HUDA BINTI MOHD SHARIF

B. ENG (HONS.) CIVIL ENGINEERING

UNIVERSITI MALAYSIA PAHANG

UNIVERSITI MALAYSIA PAHANG

DECLARATION OF THESIS AND COPYRIGHT

Author's Full Name : **SITI NURUL HUDA BINTI MOHD SHARIF**

Date of Birth : **26 MAY 1995**

Title : **ANALYSIS OF CANTILEVER STEEL BEAM**

Academic Session : **2017/2018**

I declare that this thesis is classified as:

- ☐ CONFIDENTIAL (Contains confidential information under the Official Secret Act 1997)*
- ☐ RESTRICTED (Contains restricted information as specified by the organization where research was done)*
- ☒ OPEN ACCESS I agree that my thesis to be published as online open access (Full Text)

I acknowledge that Universiti Malaysia Pahang reserves the following rights:

1. The Thesis is the Property of Universiti Malaysia Pahang
2. The Library of Universiti Malaysia Pahang has the right to make copies of the thesis for the purpose of research only.
3. The Library has the right to make copies of the thesis for academic exchange.

Certified by:

(Student's Signature)

(Supervisor's Signature)

950526-05-5436
New IC/Passport Number
Date: **JUNE 2018**

DR GUL AHMED JOKHIO
Name of Supervisor
Date: **JUNE 2018**

NOTE : * If the thesis is CONFIDENTIAL or RESTRICTED, please attach a thesis declaration letter.

THESIS DECLARATION LETTER (OPTIONAL)

Librarian,
Perpustakaan Universiti Malaysia Pahang,
Universiti Malaysia Pahang,
Lebuhraya Tun Razak,
26300, Gambang, Kuantan.

Dear Sir,

CLASSIFICATION OF THESIS AS RESTRICTED

Please be informed that the following thesis is classified as RESTRICTED for a period of three (3) years from the date of this letter. The reasons for this classification are as listed below.

Author's Name
Thesis Title

Reasons (i)

(ii)

(iii)

Thank you.

Yours faithfully,

(Supervisor's Signature)

Date:

Stamp:

Note: This letter should be written by the supervisor, addressed to the Librarian, *Perpustakaan Universiti Malaysia Pahang* with its copy attached to the thesis.

(Please take out if not related)



SUPERVISOR'S DECLARATION

I/We* hereby declare that I/We* have checked this thesis/project* and in my/our* opinion, this thesis/project* is adequate in terms of scope and quality for the award of the Bachelor Degree of Civil Engineering

(Supervisor's Signature)

Full Name : DR GUL AHMED JOKHIO

Position : LECTURER

Date : 11 JUNE 2018



STUDENT'S DECLARATION

I hereby declare that the work in this thesis is based on my original work except for quotations and citations which have been duly acknowledged. I also declare that it has not been previously or concurrently submitted for any other degree at Universiti Malaysia Pahang or any other institutions.

(Student's Signature)

Full Name : SITI NURUL HUDA BINTI MOHD SHARIF

ID Number : AA14157

Date : 11 JUNE 2018

ANALYSIS OF CANTILEVER STEEL BEAM

SITI NURUL HUDA BINTI MOHD SHARIF

Thesis submitted in fulfillment of the requirements
for the award of the
Bachelor Degree in Civil Engineering

Faculty of Civil Engineering and Earth Resources
UNIVERSITI MALAYSIA PAHANG

JUNE 2018

ACKNOWLEDGEMENTS

First and foremost, thanks to God for giving the strength and spirit which allows me to accomplished this final year project as a requirement to graduate and acquire in a Bachelor of Civil Engineering from University Malaysia of Pahang (UMP).

I would like to take this golden opportunity to express my deep and sincere gratitude to my supervisor, Dr Gul Ahmed Jokhio. His wide knowledge and his logical way of thinking have been of great value for me. His understanding, encouraging and guidance have provided a good basis for me during my research. I am grateful to have Dr Gul Ahmed Jokhio as my supervisor for these detailed and constructive comments, and for his important support throughout this research.

In addition, I would also like to thanks all my beloved friends especially my team members. Last but not least, millions of thanks and gratitude appreciations to my parents and siblings. Without their encouragement and understanding, it would have been impossible for me to finish this thesis. Thanks again for their undying love and care.

ABSTRAK

Penggunaan rasuk keluli dalam industri pembinaan telah bermula sejak beberapa tahun dahulu. Aplikasi penggunaan rasuk keluli berjenis kantilivar menjadi pilihan utama di dalam industri pembinaan. Penggunaanya menjadi pilihan utama kerana rasuk keluli berjenis kantilivar ini digunakan secara meluas dalam rekaan keluli pada bangunan perumahan awam. Objektif projek ini adalah untuk menganalisis nilai tekanan di dalam rasuk keluli bersaiz tertentu. Selain itu, projek ini bertejuan untuk mencari nilai tertinggi tekanan pada paksi y rasuk keluli tersebut. Keluli yang digunakan adalah berjenis “Hot-Rolled”, berbentuk IPE 120, dan diperbuat daripada keluli “Fe430”, Projek ini menggunakan analisis “Linear” untuk rasuk keluli dengan bantuan perisian ANSYS 12.0, “Finite Element Software” untuk menganalisis analisis “Linear Startic” tekanan dan kuasa 2D rasuk keluli berjenis kantilivar. Geometri yang digunakan untuk menghasilkan model-model adalah berasaskan beberapa elemen dan nod. Analisis ini juga menyediakan pengiraan berdasarkan tekanan dan kuasa berdasarkan teori yang sebenar. Keputusan analisis yang dihasilkan oleh perisian ANSYS telah dibandingkan dengan pengiraan yang berasaskan teori yang sebenar. Tingkah laku unsur terhingga dari rasuk keluli berjenis kantilivar didapati sedikit berbeza dengan pengiraan dan tingkah laku teoritis.

ABSTRACT

The usage of steel beam has been run for a long time in a construction industry. The application of cantilever steel beam become popular choice of Industry, by extending the steelwork design into residential housing. The objectives of this project are to analyse the forces and stresses in a selective size steel cantilever beam and finding the maximum vertical force (Newton) for beam which is hot rolled shape IPE 120 and is made of Fe430 steel. The thesis presents the linear analysis of cantilever steel beam by using ANSYS 12.0, Finite Element Software was use to model the cantilever steel beam with I beam section IPE 120. ANSYS software is used to analyse linear static of the forces and stress for the elastic 2D cantilever steel beam. The model geometry is defined with elements and nodes. This thesis also shows the theoretical calculation of stress. The results from ANSYS software were compared with the theoretical calculation results. The finite element behaviour of cantilever steel beam was found to be slightly different with the theoretical calculation and behaviour.

TABLE OF CONTENT

CHAPTER	TITLE	PAGE
	TITLE PAGES	i
	ACKNOWLEDGEMENT	ii
	ABSTRAK	iii
	ABSTRACT	iv
	TABLE OF CONTENTS	v
	LIST OF TABLES	x
	LIST OF FIGURES	xi
	LIST OF SYMBOLS	xv
	LIST OF ABBREVIATIONS	xvi
1	INTRODUCTION	1
	1.1 Introduction	1
	1.2 Problem Statement	2
	1.3 Objective of Study	2
	1.4 Scope of Study	3
	1.5 Significant of Study	3
2	LITERATURE REVIEW	5
	2.1 Introduction	5
	2.2 Steel Beam	5
	2.2.1 Steel I Beam	7
	2.2.2 Cantilever Beam - Single Load at the End	11

	2.2.3 Cantilever Types	12
	2.2.4 Advantages and Disadvantages of Using the Steel Beam	13
	2.3 Hot Rolled Steel	14
	2.3.1 Benefits of hot rolled steel	14
	2.4 Cold Formed Steel Structure	15
	2.4.1 Benefits of Cold-formed Steel Section	16
3	METHODOLOGY	17
	3.1 Introduction	17
	3.2 ANSYS Concepts	18
	3.3 ANSYS Modeller	18
	3.4 Approach and Assumption	21
	3.5 Parameters of Cantilever Steel Beam	21
	3.6 Stress Analysis Steps	22
	3.6.1 Preprocessing	22
	3.6.2 Solution	22
	3.6.3 Postprocessing	23
	3.7 Interactive Step-by-Step Solution	23
	3.7.1 Preprocessing	23
	3.7.2 Solution	34
	3.7.3 Solve	38
	3.7.4 Post processing	39
4	RESULT AND ANALYSIS	45
	4.1 Introduction	45
	4.2 Boundary Condition	47
	4.3 Forces and Moment	48

4.3.1	Graph Result of Force and Moment	48
4.3.1.1	Bending Moment for Initial Load (2500N)	48
4.3.1.2	Bending Moment for forces of 3000N	49
4.3.1.3	Bending Moment for forces of 3500N	49
4.3.1.4	Bending Moment for forces of 4000N	50
4.3.1.5	Bending Moment for forces of 4500N	50
4.3.1.6	Bending Moment for forces of 5000N	51
4.3.1.7	Bending Moment for forces of 5500N	52
4.3.1.8	Bending Moment for forces of 6000N	52
4.3.1.9	Bending Moment for forces of 6500N	53
4.3.1.10	Bending Moment for forces Of 7000N	53
4.4	Table of Stresses Result From Ansys	54
4.5	Value of Stress From Theoretical Calculation	57
4.5.1	Calculation of Stress	58
4.5.2	Theoretical Value of Stress	59
4.6	Graph Result of Stress and Strain	60
4.6.1	Graph Result of Stress and Strain	60

	For 2500N of Load	
4.6.2	Graph Result of Stress and Strain	61
	For 3000N of Load	
4.6.3	Graph Result of Stress and Strain	61
	For 3500N of Load	
4.6.4	Graph Result of Stress and Strain	62
	For 4000N of Load	
4.6.5	Graph Result of Stress and Strain	62
	For 4500N of Load	
4.6.6	Graph Result of Stress and Strain	63
	For 5000N of Load	
4.6.7	Graph Result of Stress and Strain	63
	For 5500N of Load	
4.6.8	Graph Result of Stress and Strain	64
	For 6000N of Load	
4.6.9	Graph Result of Stress and Strain	64
	For 6500N of Load	
4.6.10	Graph Result of Stress and Strain	65
	For 7000N of Load	
4.7	Graph Relationship of Load and Maximum Stress	65
5	CONCLUSION AND RECOMMENDATIONS	67
5.1	Introduction	67
5.2	Conclusion	67
5.3	Recommendations	68
	REFERENCES	70

LIST OF TABLES

TABLE	TITLE	PAGE
4.1	Load To Be Analyzed	45

LIST OF FIGURES

FIGURES	TITLE	PAGE
2.1	Steel I Beam	7
2.2	I Beam Section	9
2.3	Wide-flange I-beam	10
2.4	Force and Moment Diagram of Overhanging Cantilever Steel Beam	11
2.5	Schematic Image of Three Types of Cantilever	12
3.1	Project Flow Chart	20
3.2	A-A Section Cantilever Steel Beam	21
3.3	Change Title	23
3.4	Setting Civil Fem	24
3.5	Setting The Code	24
3.6	Setting The Unit	25
3.7	Define Materials	26
3.8	Select Steel Material	27
3.9	Confirm The Selected Material	27
3.10	Select Beam Element Types for Code Checking	28
3.11	Select Cross Section	29
3.12	Select Beam and Shell properties	29
3.13	Create Nodes In Actives Coordinate System	30
3.14	Create Nodes	30
3.15	Pick nodes	31
3.16	Create Nodes Between Two Nodes	31

3.17	Fill Between Node	32
3.18	Pick All Nodes	33
3.19	Copy Elements by Automatically Numbered	34
3.20	Copy Element	35
3.21	Select Element to Be Constrained	36
3.22	Numbered Line	36
3.23	Pick Node To Put Load	37
3.24	Select The Direction of Forces	37
3.25	Location of Load	38
3.26	Solve Current Load Step	38
3.27	Read Results by Load Step Number	39
3.28	Graph Force and Moment Results	40
3.29	Check Boundary View and View Point.	40
3.30	Cross Section Explorer	41
3.31	Graph Stress and Strain Results	41
3.32	List Stress and Strain Results	42
3.33	List Stress and Strain Results	43
3.34	Graph Sections Results	44
4.1	Boundary Condition For Cantilever Steel Beam Hot Rolled IPE 120	47
4.2	Bending Moment Z for Hot Rolled IPE 120 (2500N)	48
4.3	Bending Moment Z for Hot Rolled IPE 120 (3000N)	49

4.4	Bending Moment Z for Hot Rolled IPE 120 (3500N)	49
4.5	Bending Moment Z for Hot Rolled IPE 120 (4000N)	50
4.6	Bending Moment Z for Hot Rolled IPE 120 (4500N)	50
4.7	Bending Moment Z for Hot Rolled IPE 120 (5000N)	51
4.8	Bending Moment Z for Hot Rolled IPE 120 (5500N)	52
4.9	Bending Moment Z for Hot Rolled IPE 120 (6000N)	52
4.10	Bending Moment Z for Hot Rolled IPE 120 (6500N)	53
4.11	Bending Moment Z for Hot Rolled IPE 120 (7000N)	53
4.12(a)	Table of Stress of Cantiliver Steel beam, Hot Rolled IPE 120	54
4.12(b)	Table of Stress of Cantilever Steel beam, Hot Rolled IPE 120	55
4.13(a)	Table of Stress of Cantilever Steel beam, Hot Rolled IPE 120	56
4,13(b)	Table of Stress of Cantilever Steel beam, Hot Rolled IPE 120	57
4.14	Over hanging Cantilever Steel Beam	57
4.15	Value of Stress and Strain From Theoretical	59

Calculation Hot Rolled IPE 120

4.16	Graph of Stress SX at Point 50 (2500N)	60
4.17	Graph of Stress SX at Point 50 (3000N)	61
4.18	Graph of Stress SX at Point 50 (3500N)	61
4.19	Graph of Stress SX at Point 50 (4000N)	62
4.20	Graph of Stress SX at Point 50 (4500N)	62
4.21	Graph of Stress SX at Point 50 (5000N)	63
4.22	Graph of Stress SX at Point 50 (5500N)	63
4.23	Graph of Stress SX at Point 50 (6000N)	64
4.24	Graph of Stress SX at Point 50 (6500N)	64
4.25	Graph of Stress SX at Point 50 (7000N)	65
4.26	Maximum Stress vs Load for Cantilever Steel I	65

Beam IPE 120

LIST OF SYMBOL

δ = Deflection

LIST OF ABBREVIATIONS

FEM	=	Finite Element Method
AISC	=	America Institute of Steel Construction
AISI	=	America Iron and Steel Institute Specification
ASD	=	Australia Steel Structure Code
LRFD	=	Load and Resistance Factor Design
BS	=	British Standard
CAN/CSA	=	Canadian Standard
SABS	=	South Africa Standard
3D	=	Three-Dimensional
IWF	=	International Weightlifting Federation
1D	=	One-Dimensional
CAE	=	Computer Aided Engineering
FE	=	Finite Element
2D	=	Two-Dimensional
CAD	=	Computer Aid Design
UC	=	Universal Column

UB	=	Universal Beam
RSJ	=	Rolled Steel Joist
DIN	=	Deutshes Institut fur Normung
IS	=	International Standard
FEA	=	Finite Element Analysis
CS	=	Coordinate System
LS	=	Load Step
UDL	=	Distribution Load
AS/NZS	=	Australia / New Zealand Standard

CHAPTER 1

INTRODUCTION

1.1 Introduction

Beams are one of structural element which hold bending load and shear loads. It more dominant than axial load. The supports of beam are also very diverse and one of it is cantilever beam. Cantilever beam is a beam that has fixed support at one end and free at the other end. Cantilever beam can be made from many of materials, one of which is steel section. Its light weight and have a large enough strength, and made by fabrication so it has an isotropic material making easier to be applied as a beam structure. But cantilevered structure that has bigger number of internal forces toward its fixed supports, causing usage of uniform steel section becomes less optimal. Therefore, it will be investigated stress and deformation I section steel beam with uniform cross-section with half IWF steel section with diagonal cut so it has non-prismatic section when used as a cantilever beam.

A new set of effective-length factors is presented for designing doubly symmetric I-shaped built-in cantilever beams against lateral-torsional buckling when subject to different loading and restraint conditions at the cantilever tip. This refined approach rectifies inherent problems of current solutions caused by overlooking the restraint conditions as well as the limitations that existed in their original derivation. Because of these problems, the effective-length factors currently available may result in either overly conservative or unconservative designs, depending on the type of problem involved. Also, an interaction buckling design model is suggested for overhanging beams, in which the load is applied only at the cantilever tip. This design model takes into account the ratio of the length of the cantilever span to that of the back span, a significant parameter that has not generally been considered. Finally, a design procedure is given for determining the elastic critical moments of crane-trolley beams.

These analyses are based on a finite-element model that will be performing in finite element software.

1.2 Problem Statement

Across every industry today, companies are facing tough challenges—such as increased product complexity, new materials and manufacturing processes—and new opportunities from the availability of sensor-based data. Engineering today's complex systems requires new approaches, combining 1D simulation, 3D CAE, test and analytics. There are many other systems can give output at short time. But most of the result will be never accurate at time. To get an accurate result of analysis, the geometry and loads of the cantilever beam also need to be accurate. To get an accurate result of stress analysis I need to perform 3 or more analysis with different mesh densities (h Refinements. If the difference between the last and the second last analysis is below 10% then you know you are close to the maximum stress.

Repeating the steps will theoretically increase the accuracy of the next iteration of results. This means that a simulation requires much more time to compute the results, but the result may change by an insignificant value. The results of a Stress Analysis may change and you want to understand the accuracy of your stress results. Part level FE analysis falls short of natural boundary conditions. Moreover, the stress levels are higher and distribution is significantly different in compare the repeating steps will help to get an accurate results.

1.3 Objectives of Study

The aims of this study are:

- i.** To analyse the forces and stresses in a selective size steel cantilever beam.
- ii.** To find the maximum vertical force (Newton) for beam which is hot rolled shape IPE 120 and is made of Fe430 steel.

1.4 Scope of Study

The scope of study for this project is to determine the maximum load that can be applied to selective cross section of cantilever steel beam. We will discretize the beam with elastic 2D beam elements. Model geometry is defined with elements and nodes. All models were modelled and analysed by using finite element analysis software ANSYS. In this study, several samples with same dimension but different loading apply to I section cantilever steel beam, so it had a comparable loading. Analysis is held by using finite element method in a computer program named ANSYS as calculation program. The result showed that different loading will give different result of analysis. The analysis will be repeated until it reach the limit of loading that selective cross section of cantilever steel beam can withstand.

1.5 Significant of Study

The reasons in conducting this topic are to identify the maximum loading that selective cross section of cantilever steel beam can withstand. This significantly reduced the number of trial calculations required after knowing the actual limit of loading for any cantilever steel beam work. Finding the maximum loading in several material is very important in Engineering Field because all calculation need to be very accurate and possible. Several complex cases showing the validity and efficiency of the algorithm.

This research will give benefit to architectural art and technology which are responsible for coming up with initial concepts and designs, architectural technologists are more concerned with the technical side of construction. They work closely with architects and other building professionals to resolve any potential design problems before construction starts. These modification will improve in selecting the best materials and processes for the project surveying sites and identifying location benefits carrying out feasibility studies and risk assessments, increase ability in analyses architectural plans and drawings, highlighting any possible risks or problems and making amendments using 2D applications (2D), increase ability in specifying the appropriate technology and tools to be used in the project and advising where this

requires deviation from the initial design plans, liaising with the architect, surveyors and other construction professionals.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

Stress analysis describes analyses where the result quantities include stresses and strains. In other word it also calls structural analysis. Stress analysis is very close to ANSYS software, which allows several types of stress analyses. In static, ANSYS can analyses transient dynamic. ANSYS also can modal a spectrum and harmonic for Explicit Dynamic. Commonly, we will use a linear static analysis to describe the steps involved in an analysis. By learn these steps, we can quickly learn how to do other analyses. (G Lackshmi, 2009).

According to (A.M Jamadar, 2014), cantilever is a rigid structural element, such as a beam or a plate, anchored at one end to a (usually vertical) support from which it protrudes, this connection could also be perpendicular to a flat, vertical surface such as a wall. 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.

2.2 Steel Beam

Steel beam design is about selecting the lightest steel beam that will support the load without exceeding the bending strength or shear strength of the material, and without exceeding the maximum allowable deflection for the beam. We want the

lightest beam because it is generally the cheapest. Steel design is broken up into a variety of steps. You have to check to make sure the beam is braced often enough, you have to check to make sure the web does not buckle, you have to check to make sure there isn't too much shear going through the beam. (Tan., 2009).

All of these checks are normally done automatically with whatever structural design program you use, but if analysing a beam by hand (which with today's Steel Codes have become quite the tedious process), it can take quite some time to find the actual capacity of a steel beam. Structural steel comprises from components such as beams, girders, columns, trusses, floor plates, purlins and girts. They are joined at connections made using plate, structural shapes, welding, and fasteners. Some examples of the fasteners are bolts, nuts, washers and stud shear connectors. (Bayan A. S., 2011)

Here are some common steel beam shapes listed below:

I-Beam:

These steel beams are shaped like the capital letter 'I', just as the name implies. They are commonly used in industrial applications and have tapered flanges. I-shaped cross-section steel beams come in two different styles, one has a parallel flange surface (wide), whereas the other one has a slope on the inner flange surfaces. (Bayan A. S., 2010).

W-Beam:

These are similar to I-beams and they are also called as 'wide flange' beams. They have a straight flange and are most often used in residential construction.

H-Beam:

These are generally heavier and longer than I-beams. Appearing like the capital letter 'H', they have longer flanges. Often, the term is interchangeably used with I-beams, so this can be confusing at times. H-beams have webs and flanges that have the same thickness in many cases. (Xiaoyue Zhang, 2017).

Channels:

Like one half of the I-beam, these have a C-shaped cross-section. The top and bottom part of the beam have sloped inner flange surfaces that come out only on one side of the beam. They are used in supporting lighter loads since they are not as strong as the other beam shapes. (Mitra, 2014).

2.2.1 Steel I Beam



Figure 2.1 : Steel I Beam

An I-beam, also known as H-beam (for universal column, UC), w-beam (for "wide flange"), universal beam (UB), rolled steel joist (RSJ), or double-T (especially in Polish, Bulgarian, Spanish, Italian and German), is a beam with I or H-shaped cross-section. The horizontal elements of the "I" are known as flanges, while the vertical element is termed the "web". I-beams are usually made of structural steel and are used in construction and civil engineering.

The web resists shear forces, while the flanges resist most of the bending moment experienced by the beam. Beam theory shows that the I-shaped section is a very efficient form for carrying both bending and shears loads in the plane of the web.

On the other hand, the cross-section has a reduced capacity in the transverse direction, and is also inefficient in carrying torsion, for which hollow structural sections are often preferred. (Ross, 2014)

The method of producing an I-beam, as rolled from a single piece of steel, was patented by Alphonse Hallbour of the company Forges de la Providence in 1849. Bethlehem Steel was a leading supplier of rolled structural steel of various cross-sections in American bridge and skyscraper work of the mid-twentieth century. Today, rolled cross-sections have been partially displaced in such work by fabricated cross-sections. I-beams are commonly made of structural steel but may also be formed from alloy or other materials. A common type of I-beam is the rolled steel joist (RSJ)—sometimes incorrectly rendered as reinforced steel joist. British and European standards also specify Universal Beams (UBs) and Universal Columns (UCs). These sections have parallel flanges, as opposed to the varying thickness of RSJ flanges which are seldom now rolled in the UK. (Elsayed, 2012).

Parallel flanges are easier to connect to and do away with the need for tapering washers. UCs have equal or near-equal width and depth and are more suited to being oriented vertically to carry axial load such as columns in multi-stories construction, while UBs are significantly deeper than they are wide are more suited to carrying bending load such as beam elements in floors. I-beams are widely used in the construction industry and are available in a variety of standard sizes. Tables are available to allow easy selection of a suitable steel I-beam size for a given applied load. I-beams may be used both as beams and as columns. (Davidson, 2012)

I-beams may be used both on their own, or acting compositely with another material, typically concrete. Design may be governed by any of the following criteria:

- i. Deflection: the stiffness of the I-beam will be chosen to minimize deformation.
- ii. Vibration: the stiffness and mass are chosen to prevent unacceptable vibrations, particularly in settings sensitive to vibrations, such as offices and libraries.
- iii. Bending failure by yielding: where the stress in the cross section exceeds the yield stress.
- iv. Bending failure by lateral torsional buckling: where a flange in compression tends to buckle sideways or the entire cross-section buckles torsional.

- v. Bending failure by local buckling: where the flange or web is so slender as to buckle locally.
- vi. Local yield: caused by concentrated loads, such as at the beam's point of support.
- vii. Shear failure: where the web fails. Slender webs will fail by buckling, rippling in a phenomenon termed tension field action, but shear failure is also resisted by the stiffness of the flanges.

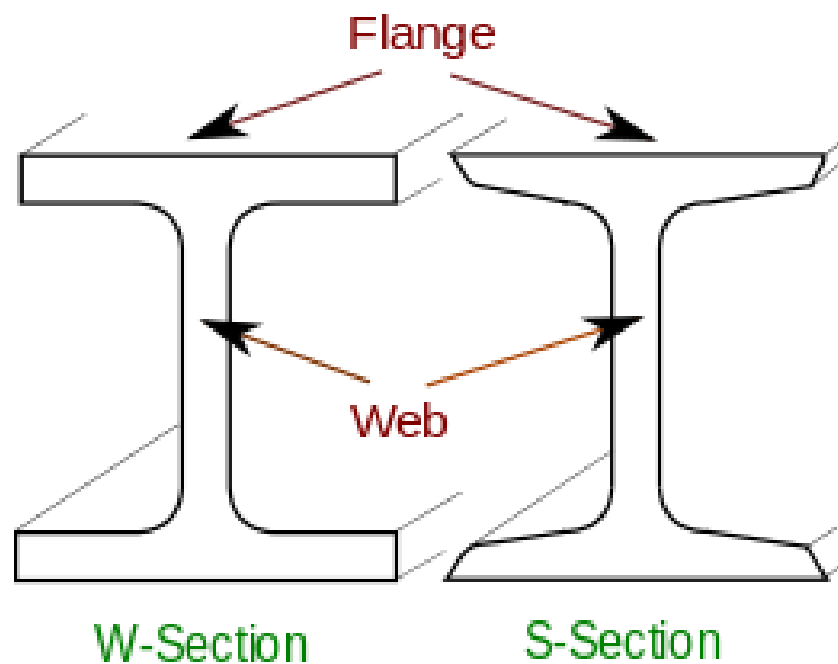


Figure 2.2: I Beam Section

The following standards define the shape and tolerances of I-beam steel sections:

- i. EN 10024, Hot rolled taper flange I sections – Tolerances on shape and dimensions.
- ii. EN 10034, Structural steel I and H sections – Tolerances on shape and dimensions.
- iii. EN 10162, Cold rolled steel sections – Technical delivery conditions – Dimensional and cross-sectional tolerances
- iv. DIN 1025-5
- v. ASTM A6, American Standard Beams
- vi. BS 4-1

- vii. IS 808 – Dimensions hot rolled steel beam, column, channel and angle sections
AS/NZS 3679.1 – Australia and New Zealand standard

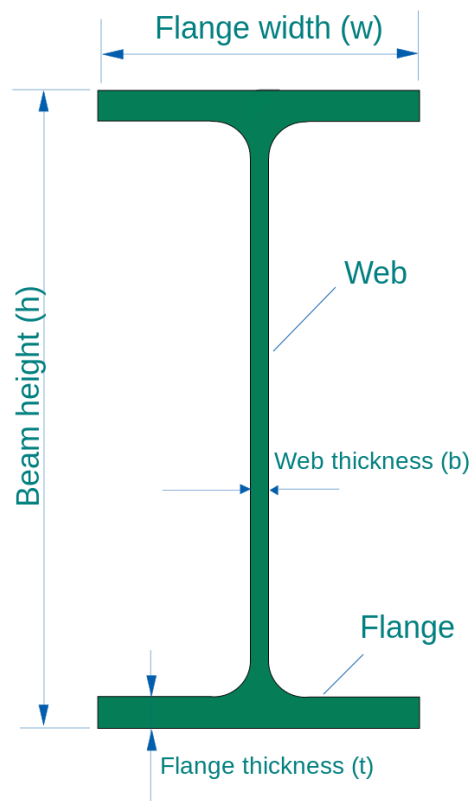


Figure 2.3: Wide-flange I-beam

2.2.2 Cantilever Beam - Single Load at the End

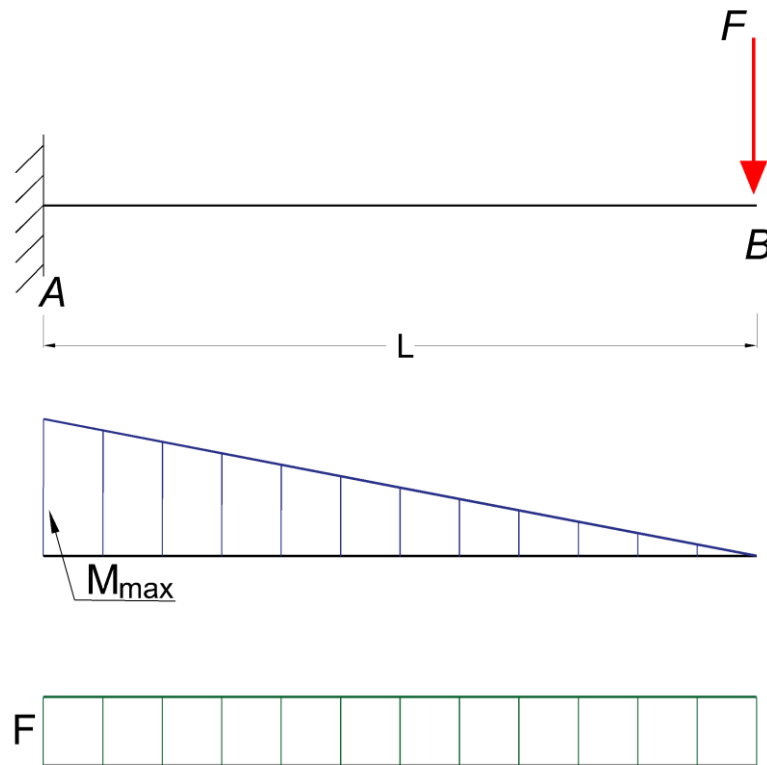


Figure 2.4: Force and Moment Diagram of Overhanging Cantilever Steel Beam

a) Maximum Reaction Force at the fixed end can be expressed as:

$$R_A = F$$

Where,

R_A = reaction force in A (N, lb)

F = single acting force in B (N, lb)

a) Maximum Moment at the fixed end can be expressed as

$$M_{\max} = M_A = -F L$$

Where,

M_A = maximum moment in A (N. m, N.mm, lb.in)

L = length of beam (m, mm, in)

b) Maximum Deflection at the end of the cantilever beam can be expressed as

$$\delta_B = F L^3 / (3 E I)$$

where,

δ_B = maximum deflection in B (m, mm, in)

E = modulus of elasticity (N/m² (Pa), N/mm², lb/in² (psi))

I = moment of Inertia (m⁴, mm⁴, in⁴)

b = length between B and C (m, mm, in)

2.2.3 Cantilever Types

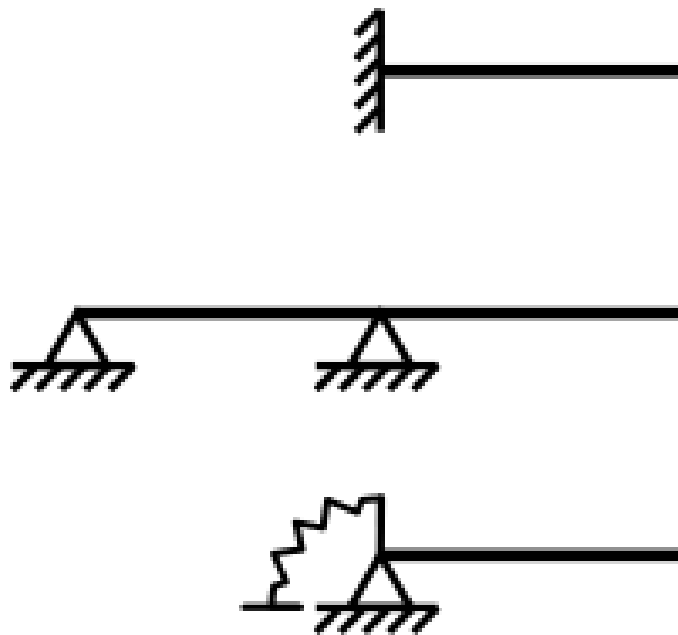


Figure 2.5: Schematic Image of Three Types of Cantilever

A schematic image of three types of cantilever. The top example has a full moment connection (like a horizontal flagpole bolted to the side of a building). The middle example is created by an extension of a simple supported beam (such as the way a diving board is anchored and extends over the edge of a swimming pool). The bottom example is created by adding a Robin boundary condition to the beam element, which

essentially adds an elastic spring to the end board. The middle and bottom example may be considered structurally equivalent, depending on the effective stiffness of the spring and beam element, (Varma, 2015).

The Cantilever method is an approximate method for calculating shear forces and moments developed in beams and columns of a frame or structure due to lateral loads. The applied lateral loads typically include wind loads and earthquake loads, which must be taken into consideration while designing buildings. The assumptions used in this method are that the points of contra flexure (or points of inflection of the moment diagram) in both the vertical and horizontal members are located at the midpoint of the member, and that the direct stresses in the columns are proportional to their distances from the centroid axis of the frame. The frame is analysed in step-wise (iterative) fashion, and the results can then be described by force diagrams drawn up at the end of the process. The method is quite versatile and can be used to analyse frames of any number of stories or floors. (Chri, 2013).

The position of the centroid axis (the center of gravity line for the frame) is determined by using the areas of the end columns and interior columns. The cantilever method is considered as one of the two primary approximate methods (the other being the portal method) for indeterminate structural analysis of frames for lateral loads. Its use is recommended for frames that are taller than they are wide, and therefore behave similar to a beam cantilevered up from the ground. (Zienkiewicz, 2014).

2.2.4 Advantages and Disadvantages of Using Steel Beam

According to (H, 2008) the benefits of steel beam:

- a) Steel beams are always more compact than engineered lumber or typical lumber headers.
- b) The deflection (how much the beam will bend under stress) is easier to dictate.
- c) Steel can offer larger openings that other beams cannot.

2.3 Hot Rolled Steel

Hot rolled steel is steel that has been roll-pressed at very high temperatures over 1,700°F, which is above the re-crystallization temperature for most steels. This makes the steel easier to form, and resulting in products that are easier to work with. (Yu C., 2018).

To process hot rolled steel, manufacturers first start with a large, rectangular length of metal, called a billet. The billet is heated and then sent for pre-processing, where it is flattened into a large roll. From there, it is kept at a high temperature and run through a series of rollers to achieve its finished dimensions. The white-hot strands of steel are pushed through the rollers at high speeds. For sheet metal, rolled steel is spun into coils and left to cool. For other forms, such as bars or plates, materials are sectioned and packaged. (Crisan A, 2012)

Steel shrinks slightly as it cools. Since hot rolled steel is cooled after processing, there is less control over its final shape, making it less suitable for precision applications. Hot rolled steel is often used in applications where minutely specific dimensions aren't crucial. Railroad tracks and construction projects often use hot rolled steel. (Xiang., 2017).

Hot rolled steel can often be identified by the following characteristics:

- i. A scaled surface, a remnant of cooling from extreme temperatures.
- ii. Slightly rounded edges and corners for bar and plate products (due to shrinkage and less precise finishing).
- iii. Slight distortions, where cooling may result in slightly trapezoidal forms, as opposed to perfectly squared angles.

2.3.1 Benefits of hot rolled steel

Hot rolled steel typically requires much less processing than cold rolled steel, which makes it a lot cheaper. Because hot rolled steel is allowed to cool at room

temperature, it's essentially normalized meaning it's free from internal stresses that can arise from quenching or work-hardening processes.

Hot rolled steel is ideal where dimensional tolerances aren't as important as overall material strength, and where surface finish isn't a key concern. Where surface finish is a concern, scaling can be removed by grinding, sand blasting, or acid-bath pickling. Once scaling has been removed, various brush or mirror finishes can also be applied. Descaled steel also offers a better surface for painting and other surface coatings.

2.4 Cold Formed Steel Structure

Over the past two decades, cold-formed steel has been seen increased usage as the structural frame for residential and multi-storey commercial buildings due to inherent features that downsidess of conventional products. Their strength, light, weight, versatility, non-combustibility, and ease of production have encouraged architects, engineers, and contractors to use cold-formed steel products which can improve structural function and building performances, and provide aesthetic appeal at lower cost. At the same time, cold formed steel members have a unique structural stability issues primarily due to the large width to thickness comparison element ratios, which is not commonly the use with in sections of hot-rolled. (Crisan A, 2012).

Cold formed steel can often be identified by the following characteristics:

- a) More finished surfaces with closer tolerances.
- b) Smooth surfaces that are often oily to the touch.
- c) Bars are true and square, and often have well-defined edges and corners.
- d) Tubes have better concentric uniformity and straightness.

2.4.1 Benefits of Cold-formed Steel Section

Based on (Mohamed, 2009) cold-formed steel has several advantages over hot-rolled steel section and these are shown below:

- a) Variety of materials which can be formed.
- b) Variety of connection and jointing methods.
- c) High strength to weight ratio
- d) Pregalvanised or recoated materials can be formed
- e) Versatility of profile shape
- f) Consistency and accuracy of profile
- g) Increase in yield strength due to cold forming

Besides, compared to timber and concrete, cold-formed steel members can also produce the following qualities.

- a) Fast and easy erection and installation
- b) Lightness
- c) High strength and stiffness
- d) Ease of prefabrication and mass production
- e) Substantial elimination of delays due to weather
- f) Economy in transportation and handling
- g) More accurate detailing
- h) Terminate-proof and rot proof
- i) Formwork unneeded
- j) Non-shrinking and non-creeping at ambient temperatures
- k) Uniform quality
- l) Non-combustibility

CHAPTER 3

METHODOLOGY

3.1 Introduction

This is an analysis of stress for a cantilever steel beam. I have selected a most suitable size, type and material to be used in my analysis. The results of these models were obtained by using the finite element software ANSYS. ANSYS structural analysis software is used widely in the industry to enable engineers to optimize their product designs and at the same time it can reduce the costs of physical testing. The intuitive interface of ANSYS Mechanical enables engineers at all levels to get answers fast.

The intuitive interface of ANSYS Mechanical enables engineers at all levels to get answers fast. ANSYS has intelligent meshing technology which is enabling us to rapidly obtain optimal meshing of the model. High quality meshes are generated, and it's easy to add controls for fine-tuning. A complete range of analysis tools is used to analyse single load cases, vibration or transient analysis, we can also examine linear and nonlinear behaviour of materials, joints and geometry.

3.2 ANSYS Concepts

Generally for basic concepts in using ANSYS:

- a) Building a model structure
- b) Entering the attribute into model
- c) Conduct analysis

In ANSYS, geometry is function that is used to build models in interface. There are four types of geometric properties which are point, lines, surface and volume. Each of these geometric properties must be processed in various ways to build structural models.

Entering the attribute is an important step in using ANSYS. The types of attribute in this software are mesh, the nature of geometry, type of materials, support and load, shear, density. Mesh is the process to discrete the model into smaller elements, the nature of geometry also include geometry information that cannot be done during modelling, type of material is used to determine whether the model is made of concrete, steel or others. Support shows how structure is held while the shear and density show how the structure is resist the load.

When all the attributes are include, the analysis can be done by running the software, if there is no error in building the model and assigning the attribute, statements about this can be issued.

3.3 ANSYS Modeller

ANSYS structural analysis software enables us to solve complex structural engineering problem. With the finite element analysis (FEA) tools available in the suite, we can customize and automate solutions for our structural mechanics problems and also can parameterize them to analyse multiple design condition. In addition, we can connect easily to other physics analysis tools for even greater fidelity. ANSYS structural analysis software is used widely in the industry to enable engineers in optimize their product designs and at the same time it's can reduce the costs of physical testing. The intuitive interface of ANSYS Mechanical enables engineers at all levels to get answers fast.

ANSYS has intelligent meshing technology which is enabling us to rapidly obtain optimal meshing the model. High quality meshes are generated, and it's easy to add controls for fine-tuning. A complete range of analysis tools is use to analyse single load cases, vibration or transient analysis, we can also examine linear and nonlinear behaviour of materials, joints and geometry. Advanced solver technology with Autodyne and LS-DYNA enables us to carry out drop, impact and explosion simulations. AQWA, along with the offshore simulation capabilities in ANSYS Mechanical, which is provide industry specific capabilities for engineers in designing for marine environments.

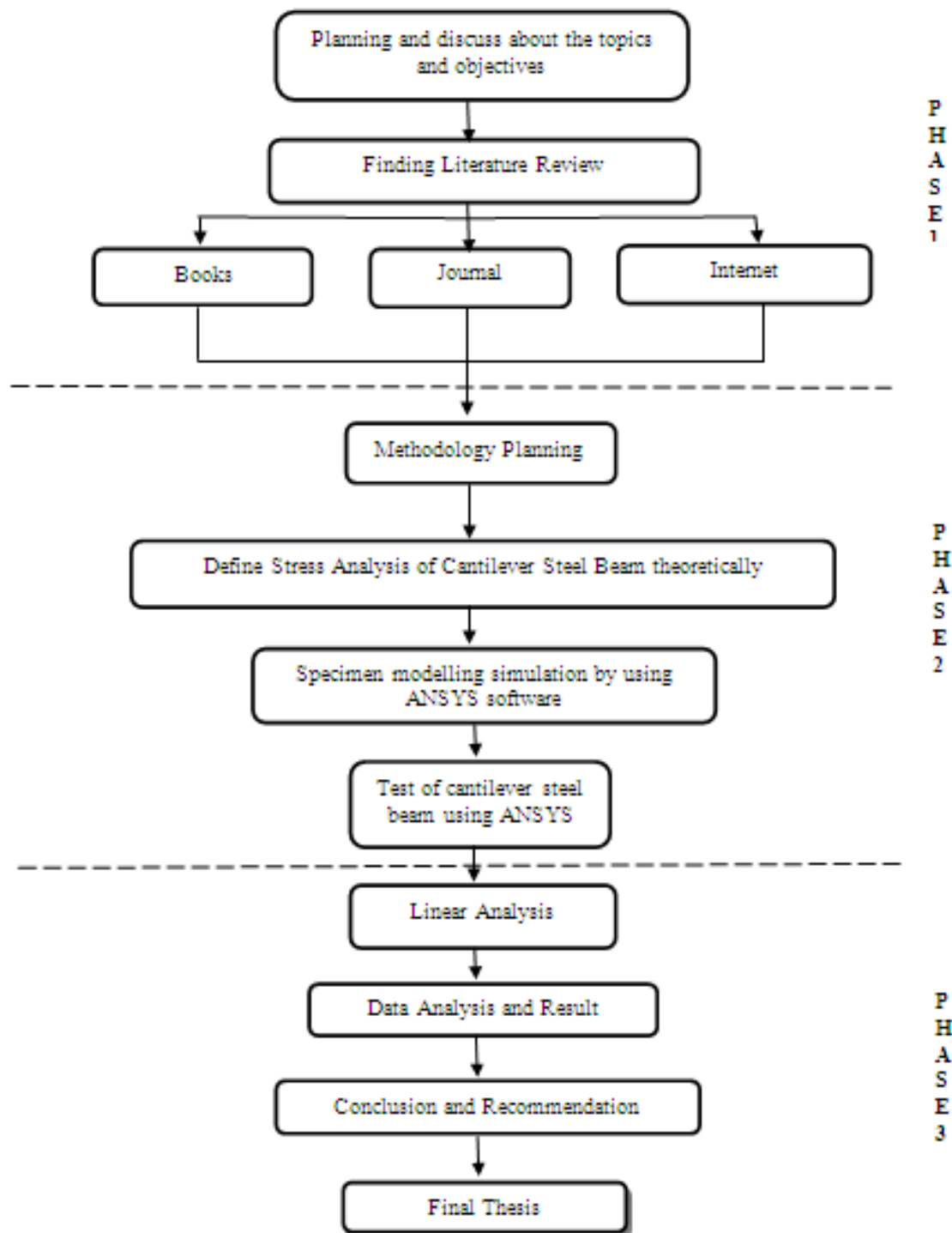


Figure 3.1: Project Flow Chart

This research is involving three main stages. In the first stage, where is beginning of the project. The propose title is “Analysis Stress of Cantilever Steel Beam”. The proposal of finite element analysis is better than experimental work

because of the time and cost saving in long term. Literature review is done by collecting information from journal, books, and other which is related.

In the beginning of stage two, modelling of form finding as testing item was modelled. The stage of modelling spacemen for form finding structure.

In stage three, all the models are analysing using the method of finite element. The analysis will be done in this project are linear analysis and linear analysis.

3.4 Approach and Assumption

We will discretize the beam with elastic 2D beam elements. Model geometry is defined with elements and nodes.

3.5 Parameters of Cantilever Steel Beam

This problem analyses the forces and stresses in a steel cantilever beam. Such a beam is subjected to a vertical force of 2500 newton at its free end. No plasticity effects are taken into accounts.

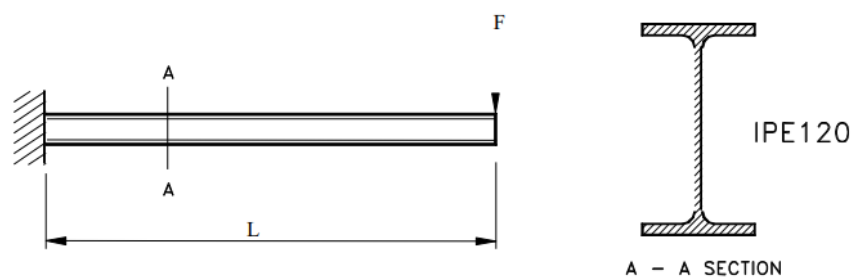


Figure 3.2 A-A Section Cantilever Steel Beam

The geometry and loads of the cantilever beam are shown in the previous figure. The beam is a hot rolled shape IPE 120 and is made of Fe430 steel. The following is a list of all the input parameters

- a) Material Fe 430
- b) Section type Hot Rolled IPE 120
- c) Length $L = 2$ m
- d) Load $F = 2500$ N

3.6 Stress Analysis Steps

3.6.1 Preprocessing

- a) Specify title
- b) Set code
- c) Set units
- d) Define material
- e) Define element type
- f) Define section
- g) Define Beam properties
- h) Define Nodes and Elements
- i) Save the database

3.6.2 Solution

- j) Apply displacement constraints
- k) Apply force load
- l) Solve

3.6.3 Post processing

- a) Enter the postprocessor and read results
- b) Plot bending moment
- c) Plot the bending stress in Y top
- d) List bending extreme stresses
- e) Plot bending stress distribution inside the cross-section
- f) . Exit the ANSYS program

3.7 Interactive Step-by-Step Solution

3.7.1 Preprocessing

A typical CivilFEM analysis begins providing data such as the units system, active code, materials, element types, and section and model geometry definition.

- a) Specify title

Although this step is not required for a CivilFEM analysis, we recommend that you make it part of all your analyses. Select Utility Menu: File, and Change title

- i) Enter the title: “Cantilever Steel Beam”
- ii) OK to define the title and close the dialog box.

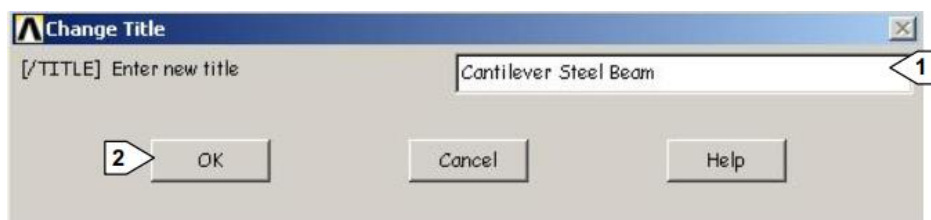


Figure 3.3: Change Title

b) Set code

In CivilFEM you can choose between different codes for checking and designing. CivilFEM allows you to uphold different active codes simultaneously, one for concrete calculations another one for steel calculations and a third one for seismic design. In this example the active code is Euro code 3, which is the,

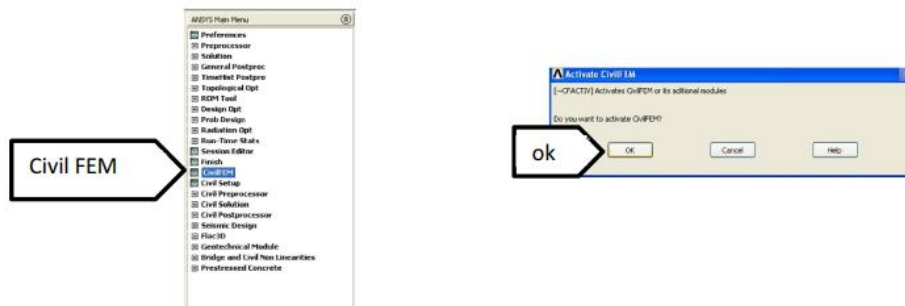


Figure 3.4: Setting Civil Fem

c) Select Civil Set Up

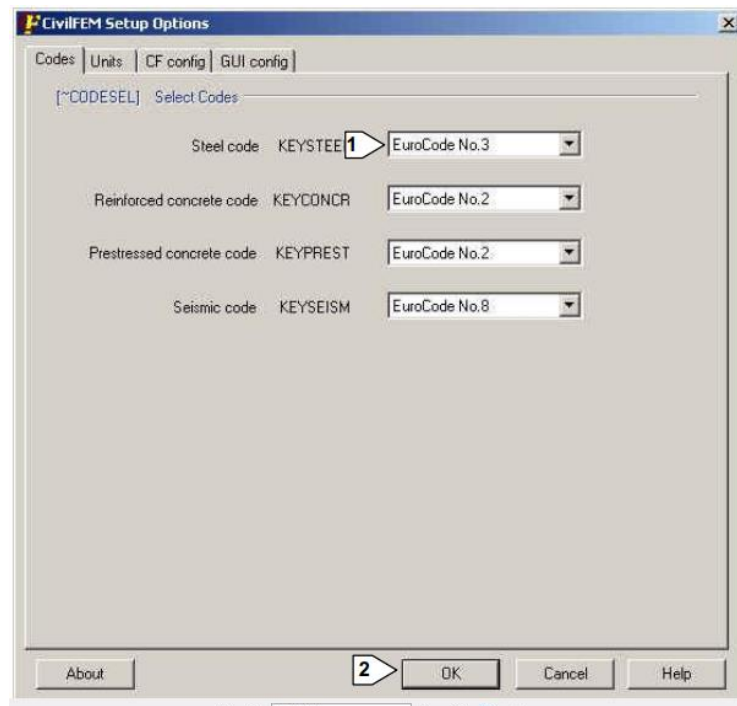


Figure 3.5: Setting The Code

d) Set units

In CivilFEM you must define a unit system. CivilFEM will need such system to perform calculations according to Code. You should maintain it during the entire design. In this analysis, we will select SI units, that is, meters, seconds and newtons

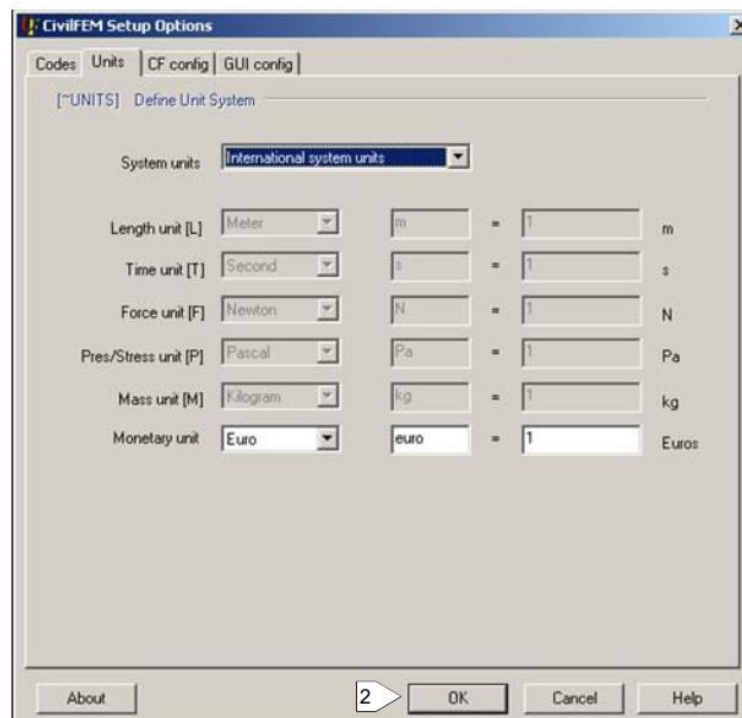


Figure 3.6: Setting The Unit

e) Define material

Main Menu, than select , Select Civil Preprocess Choose Materials. Pick new to define a new material. Pick on the Steel icon for structural steel

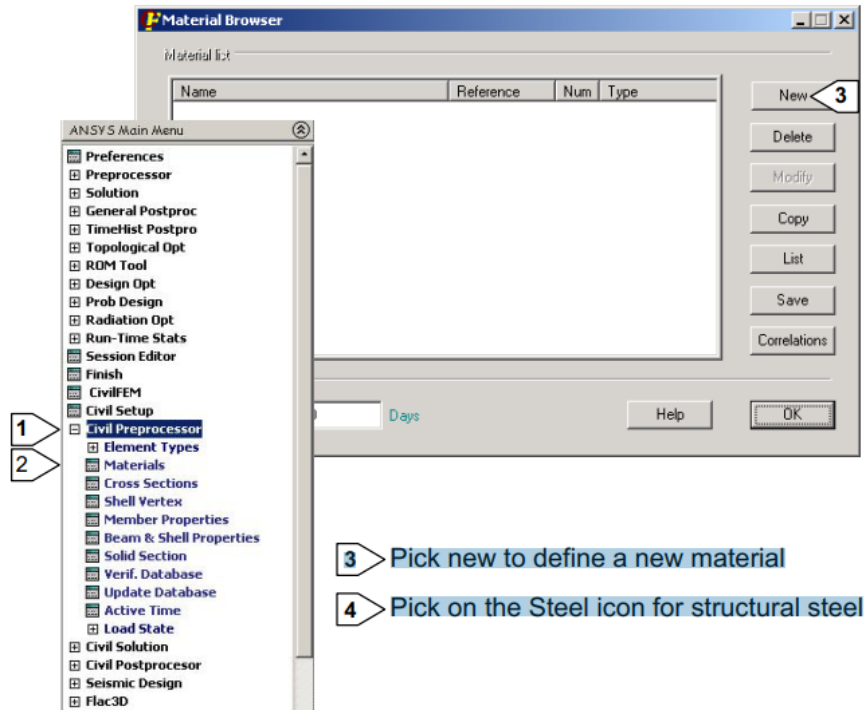


Figure 3.7: Define Materials

Pick on the EC3 icon to choose steel from EC3 code, Choose Fe430 Steel and all the material properties corresponding to Fe430 steel are automatically calculated according to Eurocode 3 (active code). Add to define the material properties set. Exit to close the dialog box, press OK.

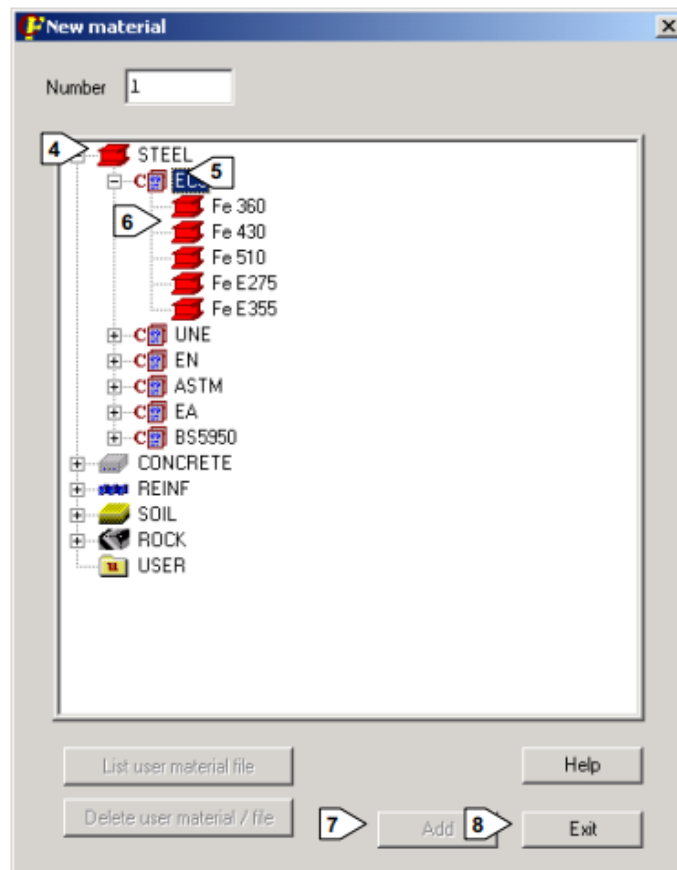


Figure 3.8: Select Steel Material

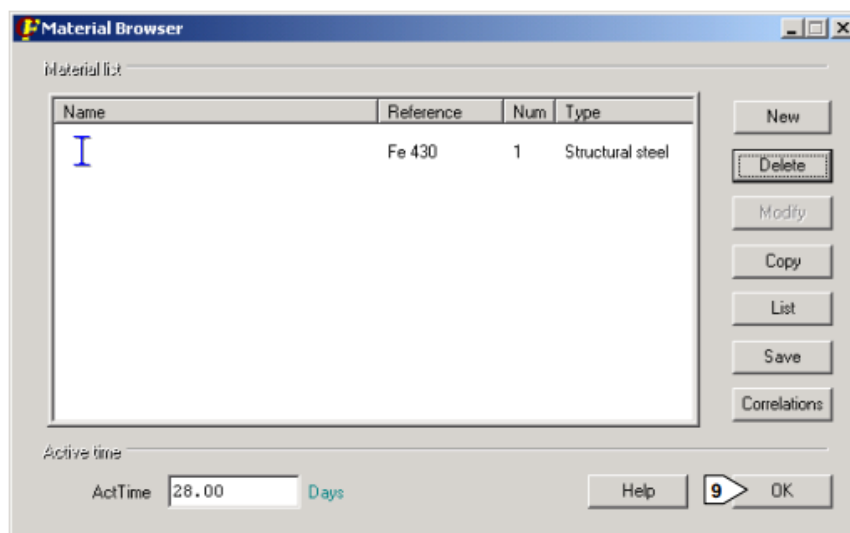


Figure 3.9: Confirm The Selected Marterial

f) Define element type

We will use 2D Elastic Beam 3 for this analysis. Select Main Menu, CivilFEM ,Civil Preprocessor, Element Types, Civil Beams.Then select 2D Elastic Beam 3 and OK to define element type.

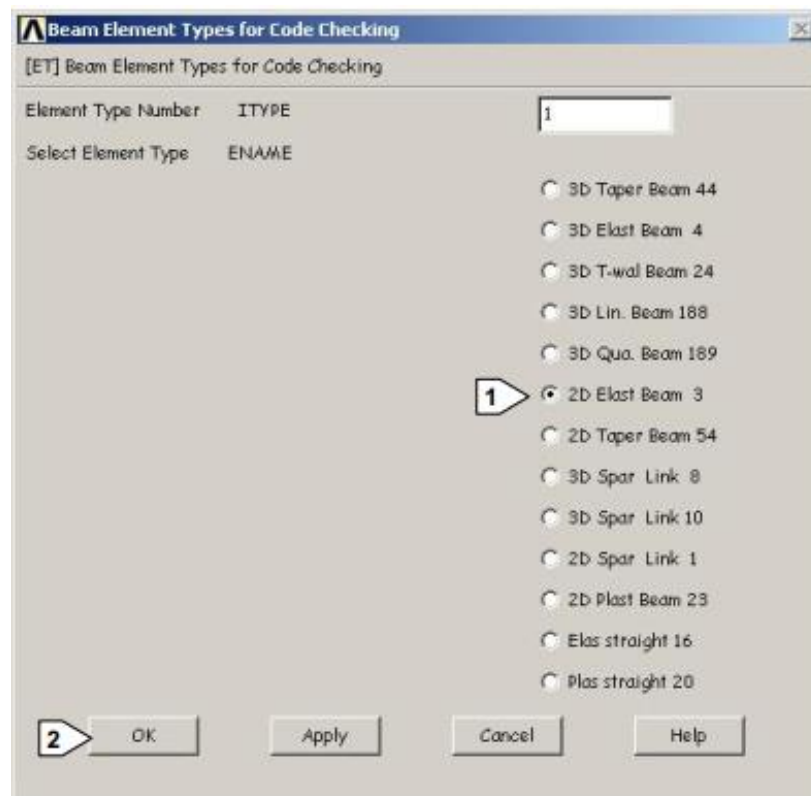


Figure 3.10: Select Beam Element Types for Code Checking

g) Define section

We will use IPE 120 and it will be selected from the Library using the Cross sections Explorer. Select Main Menu, CivilFEM, Civil Preprocessor, Cross Sections Click on the Hot Rolled Button, select I Beams group, select IPE 120 Shape, select OK to define cross section 1, than Exit to close cross section explorer

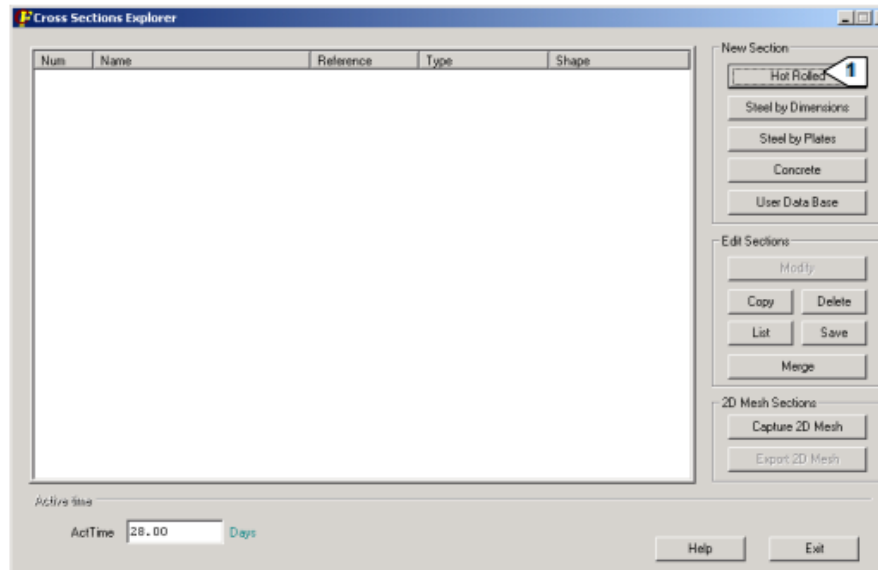


Figure 3.11: Select Cross Section

h) Define Beam and Shell Propertise

Pick on the New Beam button, select cross section number, enter a Name for the Beam property, select element type BEAM 3.

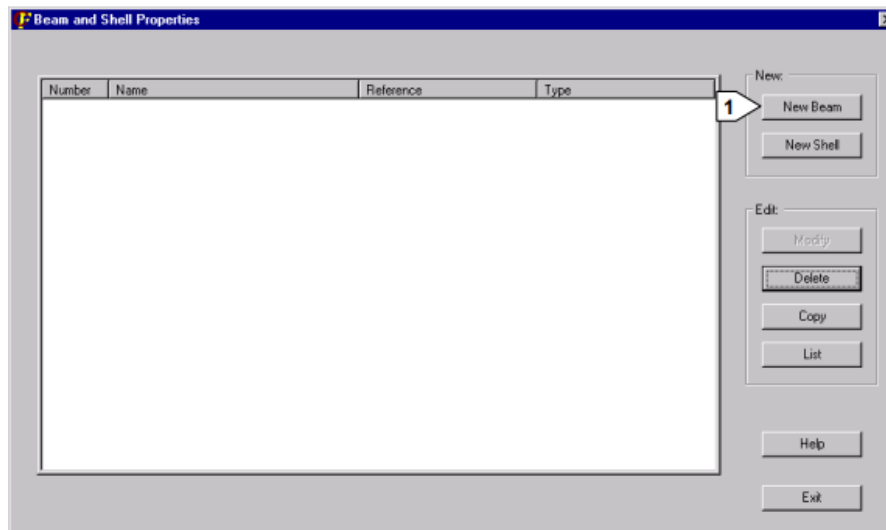


Figure 3.12: Select Beam and Shell properties

i) Define node and element

Main Menu, Preprocessor, Modeling, Create Nodes, In Active de, enter x=0, y=0, z=0 for node 1 coordinates, than Apply to create the first node 1

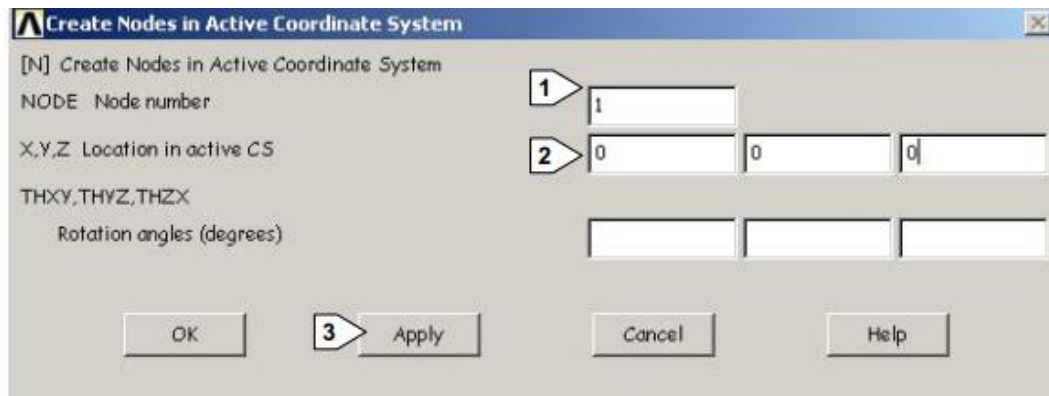


Figure 3.13: Creat Nodes In Actives Coordinate System

Enter 11 for last node, Enter x=2, y=0, z=0 for node 11 coordinates, than select OK to create the last node and close the dialog box

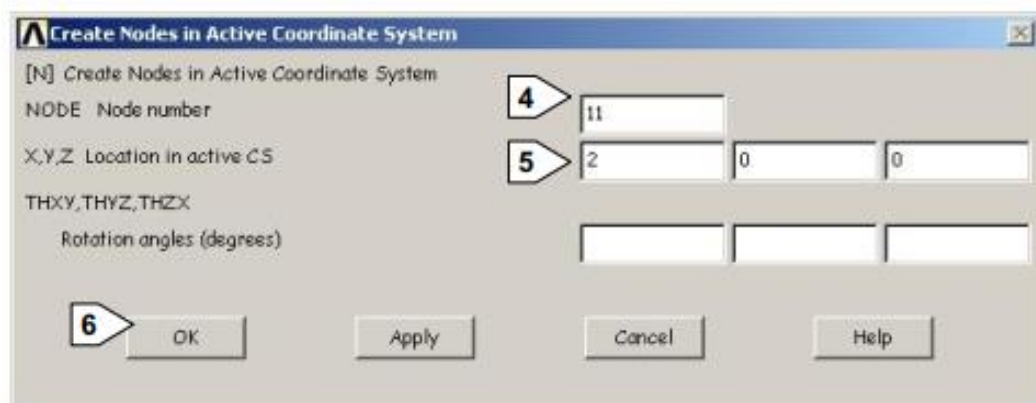


Figure 3.14: Creat Nodes

Main Menu, Preprocessor, Modeling, Create, Nodes, Fill between Nds , pick node 1 (on the coordinate system origin) and node 11, than press OK to finish picking nodes, and press OK to fill between nodes

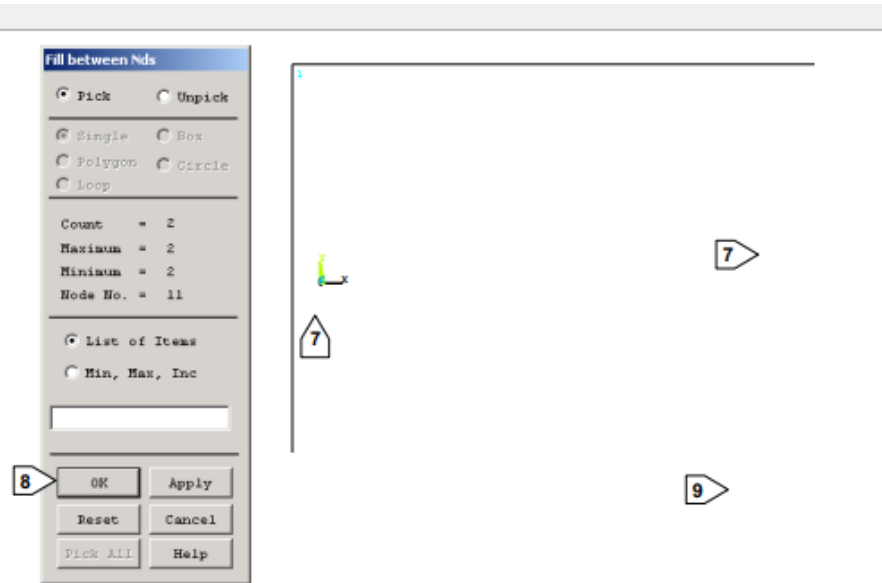


Figure 3.15: Pick nodes

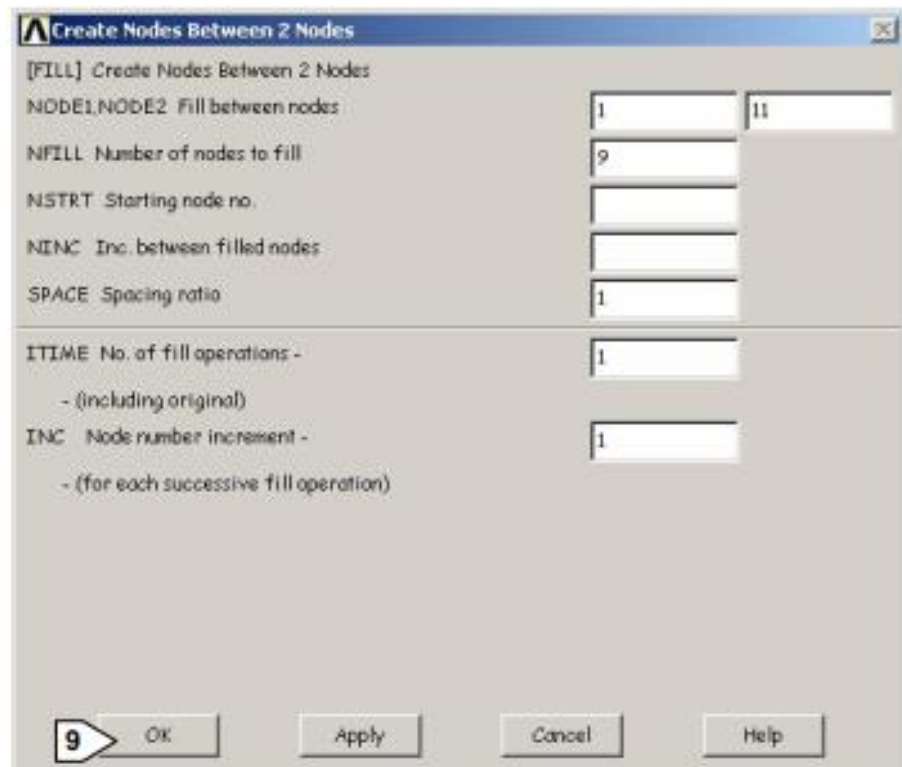


Figure 3.16: Creat Nodes Between Two Nodes

Main Menu: Preprocessor, Modeling , Create, Elements, Auto numbered
Through nodes, than pick node 1 (on the coordinate system origin) and node 2, press
OK to finish picking nodes

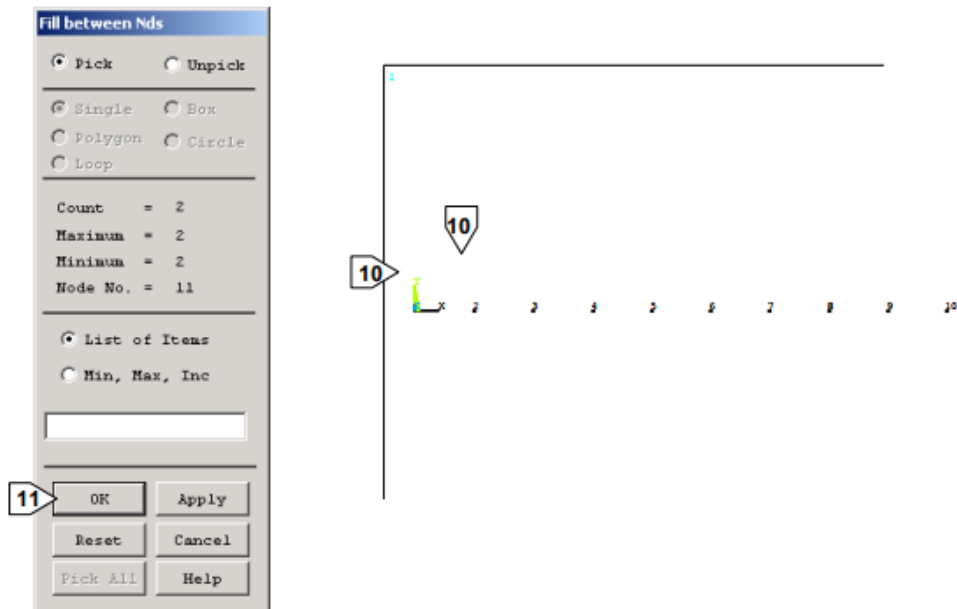


Figure 3.17: Fill Between Node

Main Menu: Preprocessor, Modeling , Copy, Elements, Auto Numbered Pick
All (to select the element), Enter 10 (number of copies including original), Enter 1
(node number increment), press OK to generate elements.

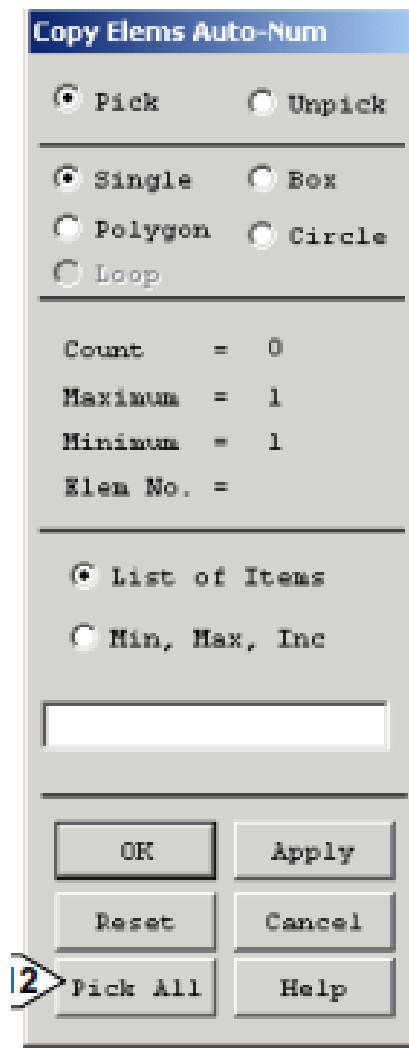


Figure 3.18: Pick All Nodes

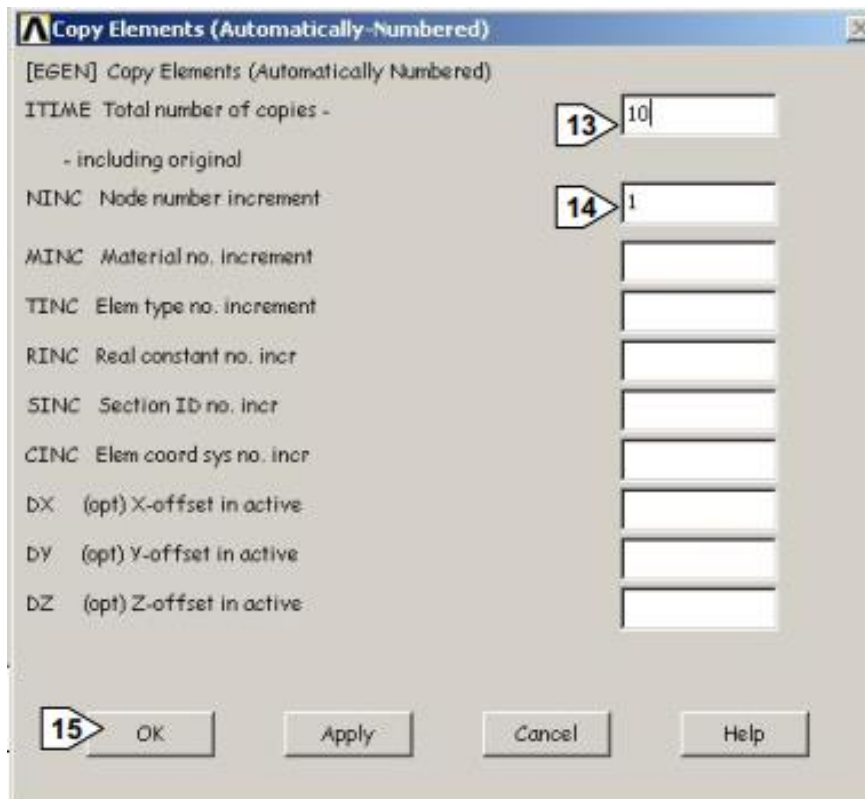


Figure 3.19: Copy Elements by Automatically Numbered

j) Save the database

Before moving to the next step, we will save all we have done so far. The save operation will save the database to file.db and file.cfdb.

3.7.2 Solution

In this step we will define the analysis type and its options, apply loads and initiate the finite element solution. A new, static analysis is the default option, so we will not need to specify the analysis type for this problem. Moreover, there are no analysis options for this problem.

- i. Apply displacement constraints

The model is a cantilever beam so you have to constrain all degrees of freedom at node 1. Than go to the Main Menu, select Solution, Define Loads, Apply, Structural, Displacement, On Nodes. Pick node 1, than press OK to finish picking nodes. Select All DOF 4 and press OK to apply constraints (zero displacement for all DOF) and close dialog box.

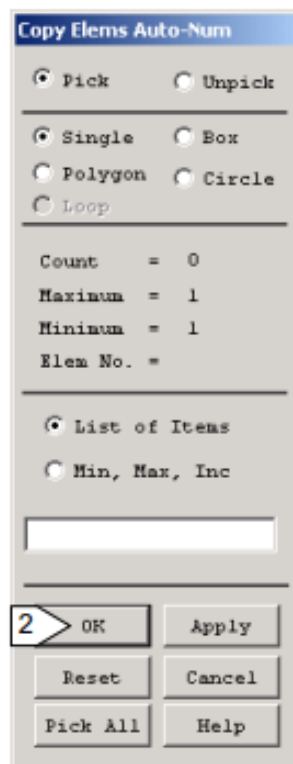


Figure 3.20: Copy Element

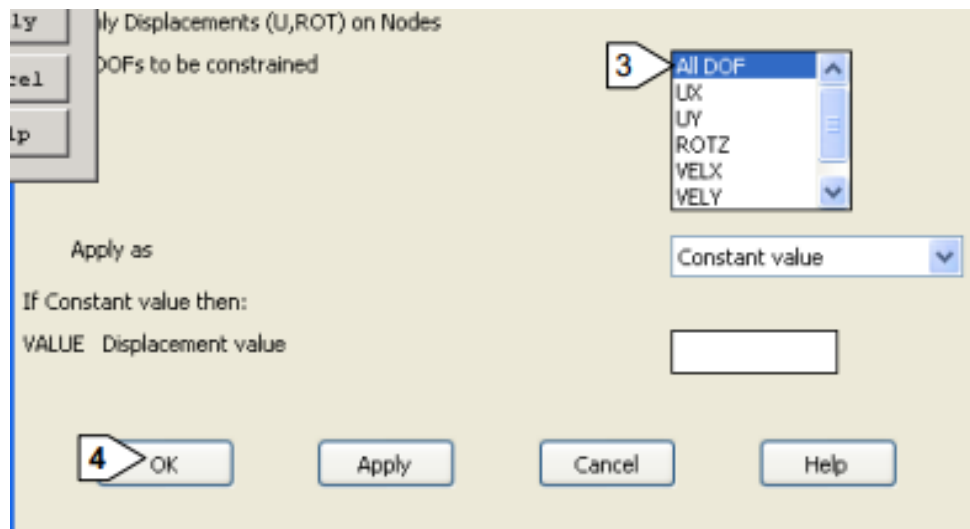


Figure 3.21: Select Element to Be Constrained

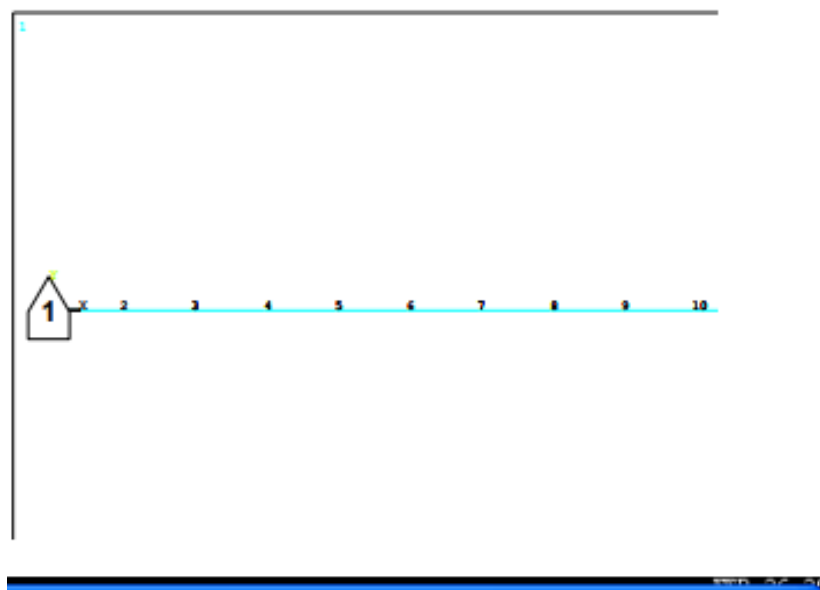


Figure 3.22: Numbered Line

a) Apply force load

You have to apply a concentrated load at node 11 with a value of 2500 N. Select Main Menu: Solution, Define Loads, Apply, Structural, Force/Moment On Nodes. Then select Pick node 11, press OK to finish picking nodes. Select FY as the force in the Y direction and enter -2500, then press OK to apply force and close dialog box.

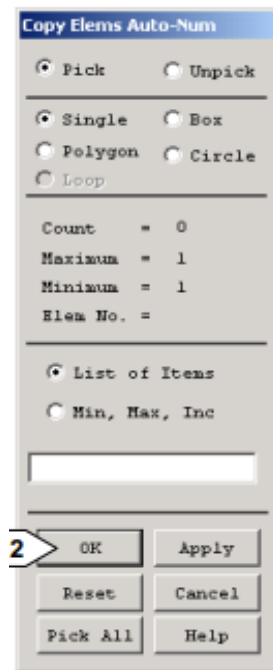


Figure 3.23: Pick Node To Put Load

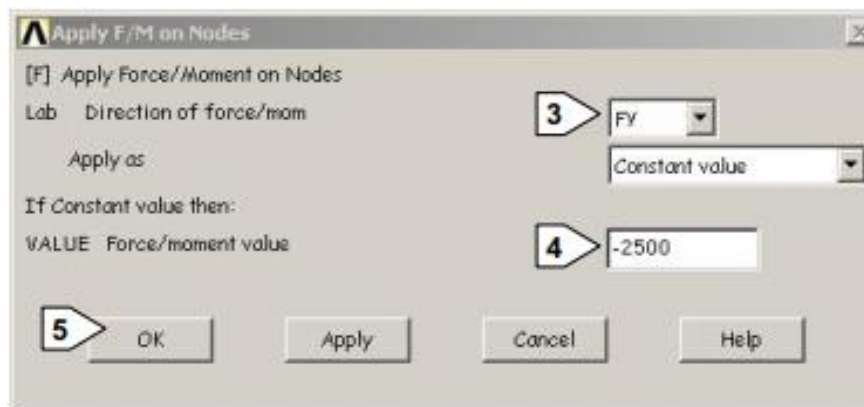


Figure 3.24: Select The Direction of Forces

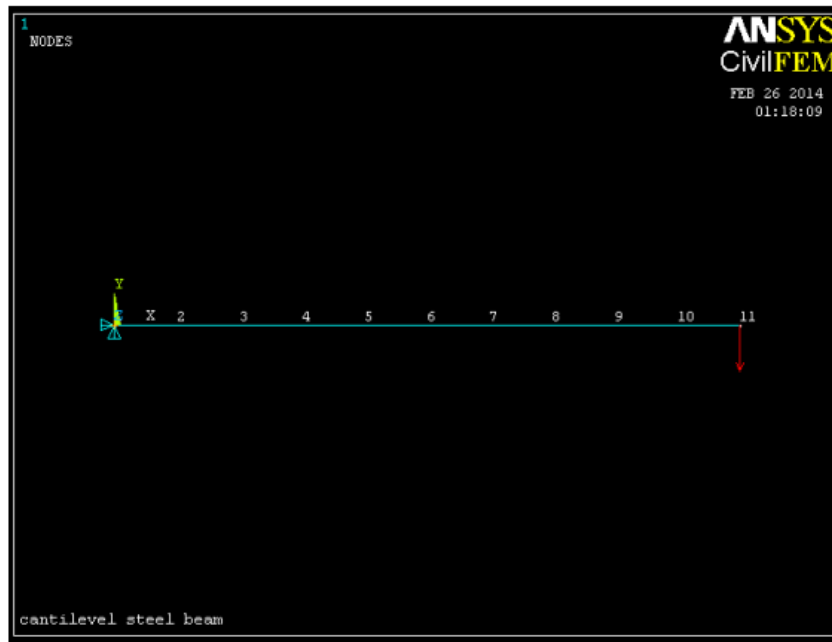


Figure 3.25: Location of Load

3.7.3 Solve

Main Menu: Solution, Solve, Current LS. Review information in the status window, and then pick File, then select Close to close the window and press OK to begin the solution. Close the information window when solution is done.



Figure 3.26: Solve Current Load Step

3.7.4 Post processing

Post processing is where you review the analysis results through graphic displays and tabular listings. CivilFEM saves all this data in its own results file called file RCV. To review its results you must define the dataset to be read from this file using the CivilFEM command ~CFSET. This command points to both ANSYS and CivilFEM data.

- a) Enter the postprocessor and read results

Main Menu: CivilFEM, Civil Postprocessor, Read Results, By Load Step. Enter 1 in the Load Step number box and press OK to read load step 1.

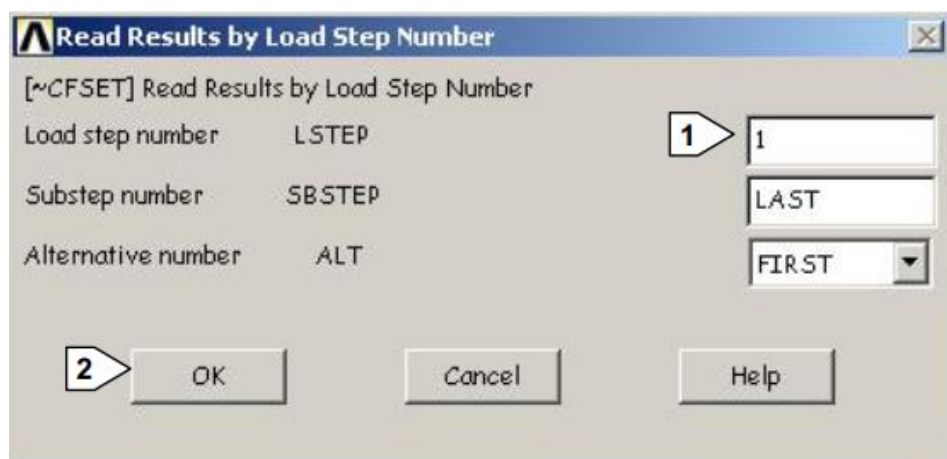


Figure 3.27: Read Results by Load Step Number

- b) Plot bending moment

Main Menu: CivilFEM, Civil Post process, Beam Utilities, GRAPH, RESULTS and select Forces & Moments. Choose bending moment Z, than press OK.

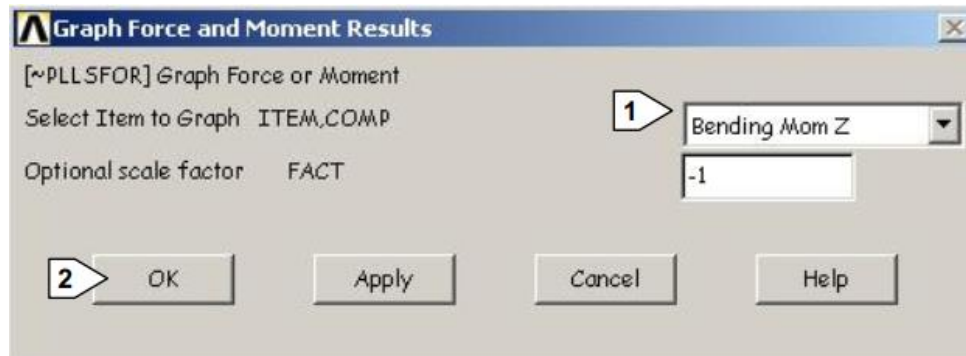


Figure 3.28: Graph Force and Moment Results

c) Plot the bending stress in Y top fiber

Main Menu: CivilFEM, Civil Preprocessor, Cross Sections. Select the IPE section and press Modify button.

Check boundry view and check view point.

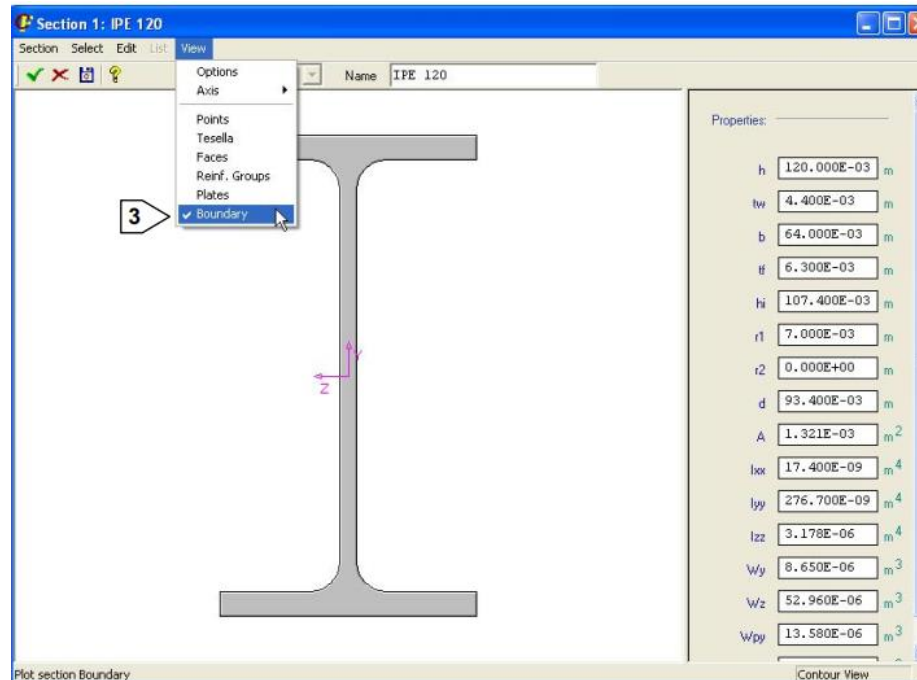


Figure 3.29: Check Boundry View and View Point.

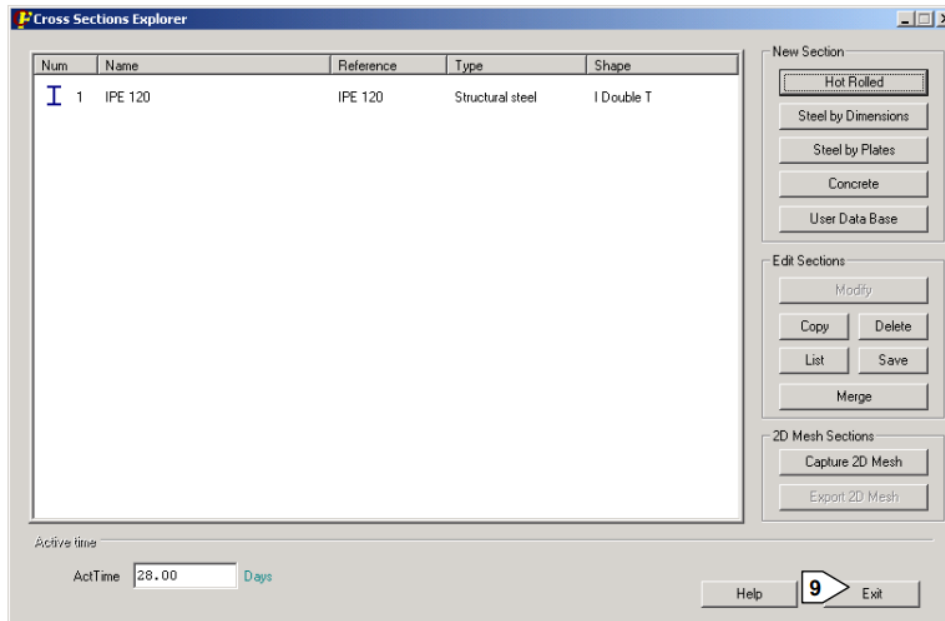


Figure 3.30: Cross Section Explorer

Select Main Menu, CivilFEM, Civil Postprocess, Beam Utilities, GRAPH RESULTS and choose Stress & Strain. Select Point to plot stresses on a section point then enter point number and press OK to plot stress results.

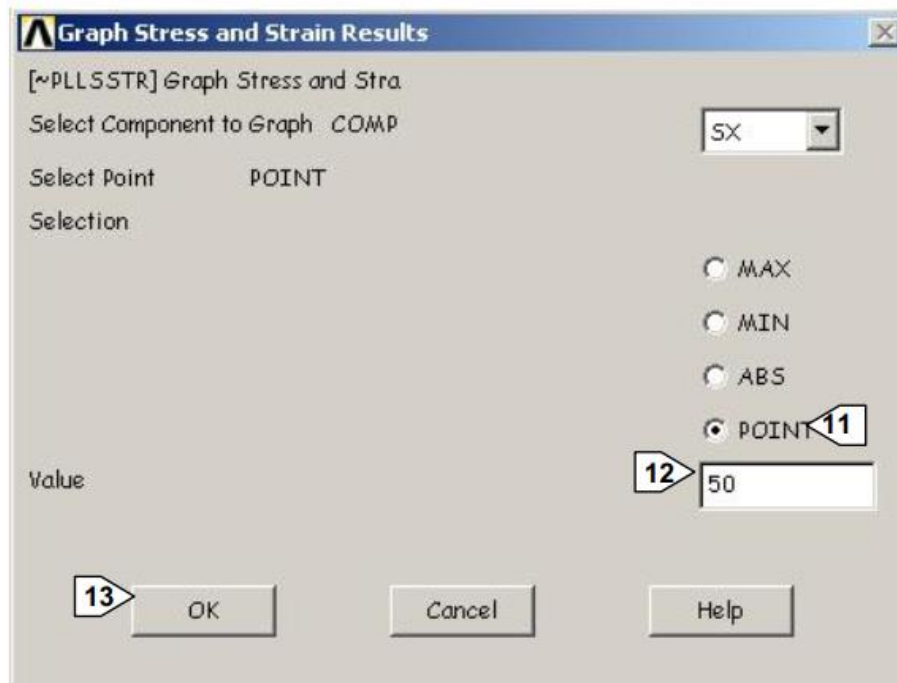


Figure 3.31: Graph Stress and Strain Results

d) List bending extreme stresses

Select Main Menu, CivilFEM, Civil Postprocess, Beam Utilities, LIST RESULTS and choose Stress & Strain.

CivilFEM allows to obtain the values of the different stress types at several section points. In this case we'll see the extreme values of bending stresses.

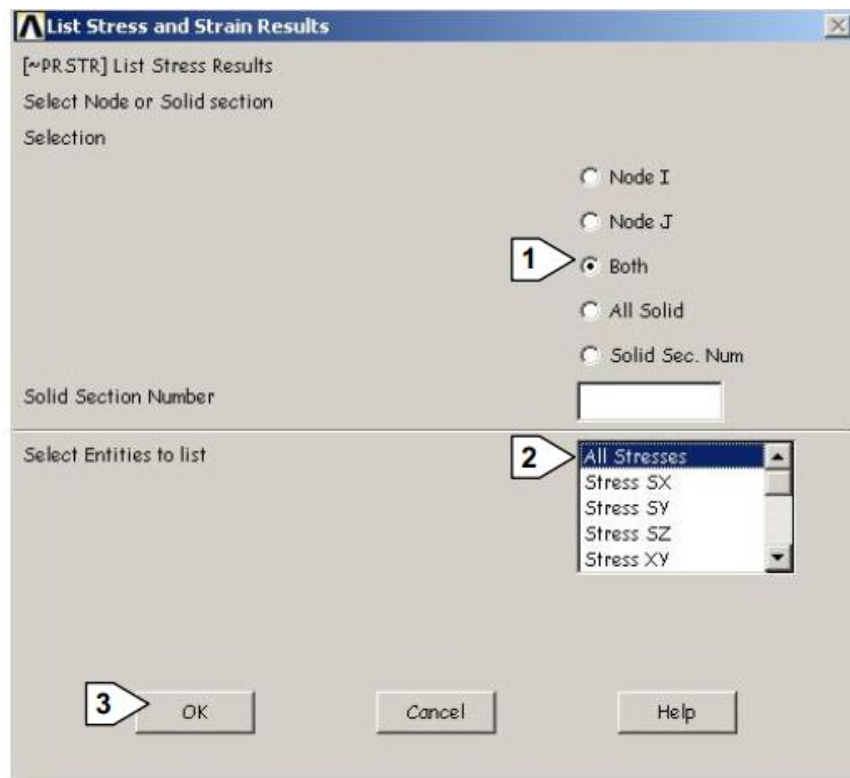


Figure 3.32: List Stress and Strain Results

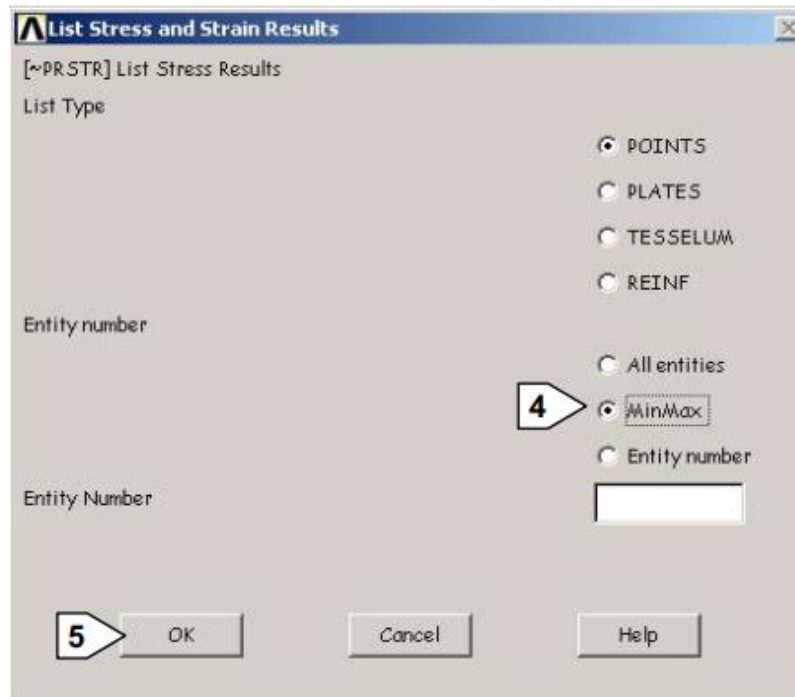


Figure 3.33 : List Stress and Strain Results

Select both nodes, I and J by select All Stresses in Entities to list and press OK. Select MinMax to list extreme values of bending stresses and press OK to list values. Then the htm file will open.

e) Plot bending stress distribution inside the cross-section

Finally we are going to plot the stress distribution inside the cross-section in the element where the bending stress is maximum. The ~ PLCSSTR command, plots the cross-section of any desired element. Since the first element has the maximum stress, it will be plotted by default:

Select Main Menu, CivilFEM, Civil Postprocess, Beam Utilities, GRAPH RESULTS. Then press Section Results, Enter Element number and press OK.

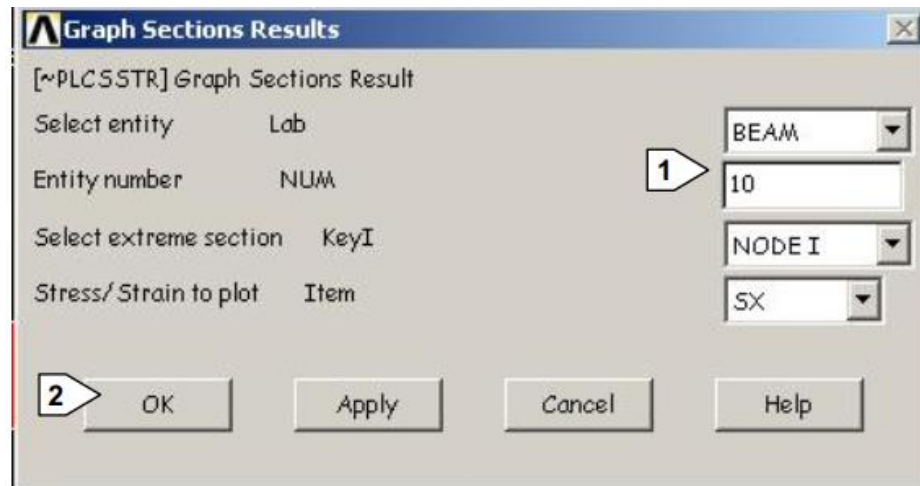


Figure 3.34 : Graph Sections Results

f) Exit the ANSYS program

Choose Toolbar Quit and select save everything than press OK.

CHAPTER 4

RESULTS AND DISCUSSION

4.1 Introduction

These are the results from the problem of analyzing the forces and stresses in the steel cantilever beam. Such a beam is initially subjected to a vertical force of 2500 Newton. The analyze process is repeated using different vertical forces which are 3000 Newton, 3500 Newton, 4000 Newton and 4500 Newton.

The geometry and loads of cantilever beam are shown in the previous chapter. The beam is a hot rolled shaped IPE 120 and is made of Fe430 steel. We will discretize the beam with elastic 2D beam elements, model geometry is defined with elements and nodes.

Table 4.1 Load to be analysed

Test	Load(N)
Test 1	2500
Test 2	3000
Test 3	3500
Test 4	4000
Test 5	4500
Test 6	5000
Test 7	5500
Test 8	6000
Test 9	6500
Test 10	7000

For my simulation work, I will conduct tens differences load with same condition boundaries. First trial with 2500N amount of load, followed by 3000N,

3500N, 4000N, 4500N, 5000N, 5500N, 6000N, 6500N, and 7000N. To get an accurate result, I repeat the simulation for each load. Based on previous studies on simulation work we can conclude that the higher number of repetition of some test or simulation, the highest probability to meet the actual amount.

4.2 Boundary Condition

Boundary condition is the most important data that need to be clearly decide to get the best result of simulation analysis. For a cantiliver steel bea, some boundary condition need to be choose wisely to produce linear static analysis of stress and strain. Therefor, the stress and strain behavior of the cantiliver steel beam is important to be dicussed. (Lee, Finite Element Simulations with ANSYS Workbench 15, 2011).

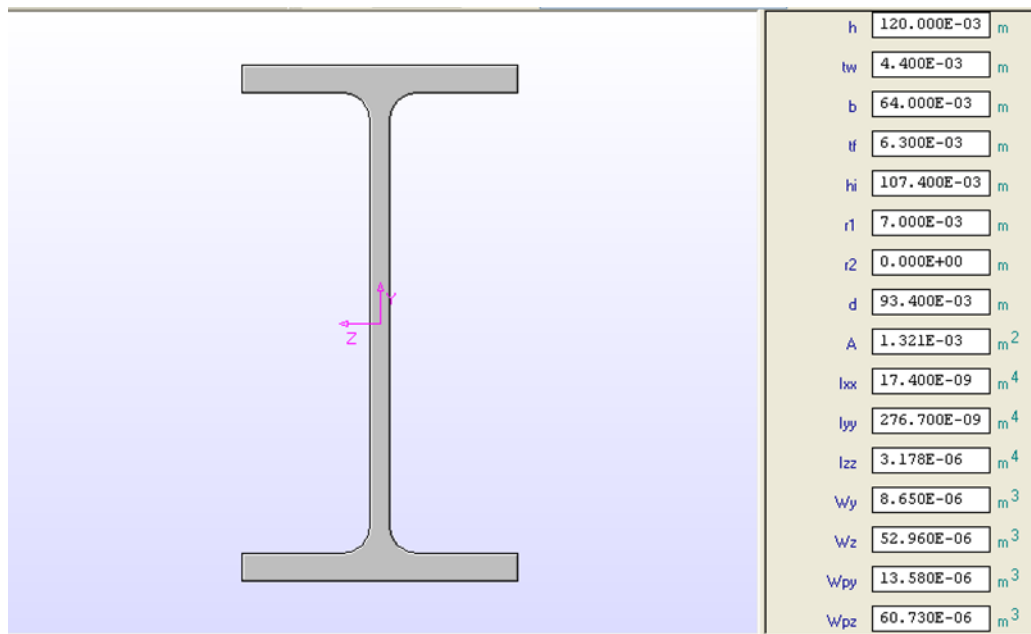


Figure 4.1: Boundary Condition For Cantiliever Steel Beam Hot Rolled IPE 120

4.3 Forces and Moment

According to Jalal et. al., (2014), shear force on cantilever beam is the sum of vertical forces acting on a particular section of a beam. While bending moment is the algebraic sum of moments about the centroidal axis of any selected section of all the loads acting up to the section.

Bending moment diagram and shear force diagram of a cantilever beam having point load at the end, several point loads. Over Whole Span ,U.D.L. not over the whole span,U.D.L. from support to some distance,U.D.L. Somewhere on the beam,Combination of Point Loads and U.D.L (Sariffudin, 2011).

4.3.1 Graph Result of Force and Moment

4.3.1.1 Bending Moment for Initial Load (2500N)

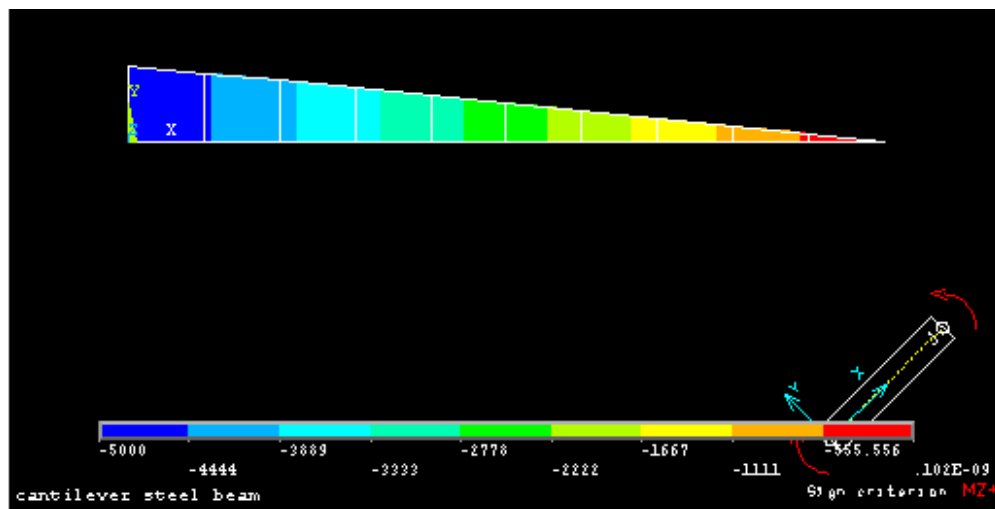


Figure 4.2: Bending Moment Z for Hot Rolled IPE 120 (2500N)

Figure 4.2 shows the bending moment at z-direction of cantilver steel beam, Hot Rolled IPE 120. The stucture is over hanging and loaded at the end of the cantiliver beam stucture with forces. The force places in y-direction due to the structure. The force used are 2500 Newton. The minimum banding moment of Z-direction is $-5.0 \times 10^3 \text{ N/mm}^2$. While the maximum bending moment of Z -direction is $0.102 \times 10^{-09} \text{ N/mm}^2$.

4.3.1.2 Bending Moment for forces of 3000N

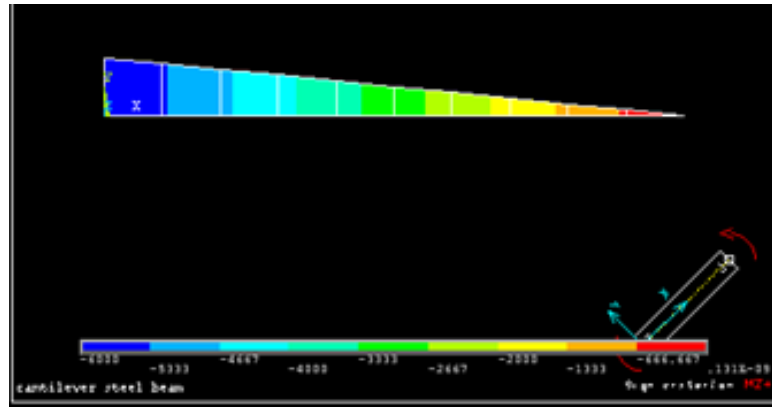


Figure 4.3: Bending Moment Z for Hot Rolled IPE 120 (3000N)

Figure above shows the bending moment at z-direction of cantilver steel beam, Hot Rolled IPE 120. The force used is 3000 Newton. The minimum banding moment of Z-direction is $-6.0 \times 10^3 \text{ N/mm}^2$. While the maximum bending moment of Z – direction is $0.131 \times 10^{-09} \text{ N/mm}^2$.

4.3.1.3 Bending Moment for forces of 3500N

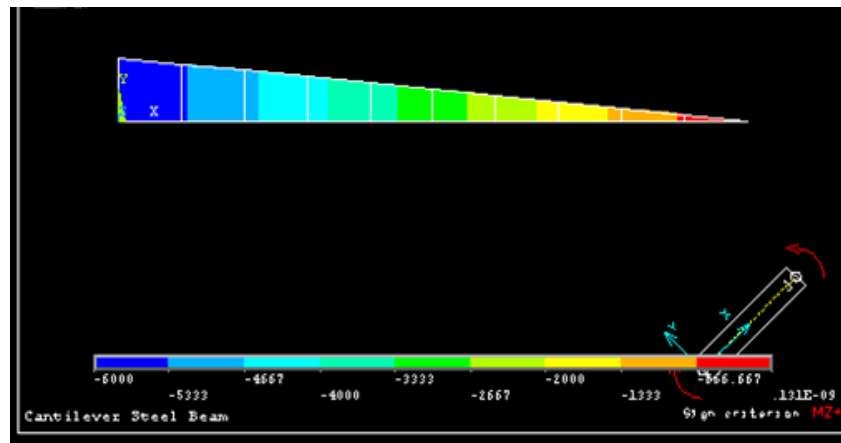


Figure 4.4: Bending Moment Z for Hot Rolled IPE 120

Figure 4.4 shows the bending moment at z-direction of cantilver steel beam, Hot Rolled IPE 120. The forces used is 3500 Newton. The minimum banding moment

of Z-direction is $-6.0 \times 10^3 \text{ N/mm}^2$. While the maximum bending moment of Z – direction is $0.121 \times 10^{-09} \text{ N/mm}^2$.

4.3.1.4 Bending Moment for forces of 4000N

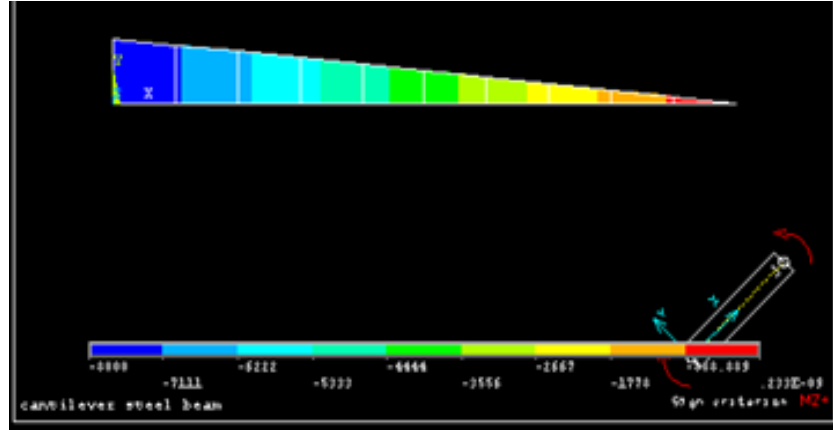


Figure 4.5: Bending Moment Z for Hot Rolled IPE 120 (4000N)

Figure 4.5 shows the bending moment at z-direction of cantilver steel beam, Hot Rolled IPE 120. The force places in y-direction due to the structure. The forces used is 4000 Newton. The minimum banding moment of Z-direction is $-7.0 \times 10^3 \text{ N/mm}^2$. While the maximum bending moment of Z –direction is $0.524 \times 10^{-09} \text{ N/mm}^2$.

4.3.1.5 Bending Moment for forces of 4500N

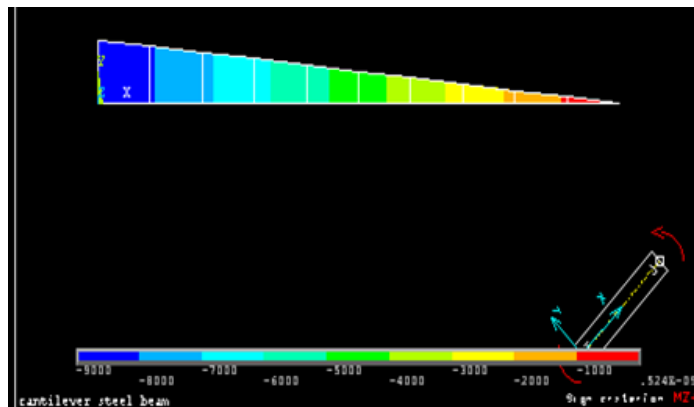


Figure 4.6: Bending Moment Z for Hot Rolled IPE 120 (4500N)

Figure 4.6 shows the bending moment at z-direction of cantilver steel beam, Hot Rolled IPE 120.. The force places in y-direction due to the structure. The force used is 4500 Newton. The minimum banding moment of Z-direction is $-9.0 \times 10^3 \text{ N/mm}^2$. While the maximum bending moment of Z –direction is $0.524 \times 10^{-09} \text{ N/mm}^2$.

4.3.1.6 Bending Moment for forces of 5000N

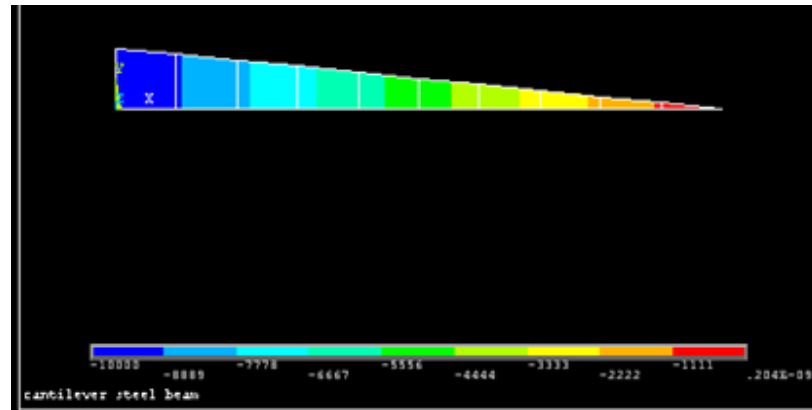


Figure 4.7: Bending Moment Z for Hot Rolled IPE 120 (5000N)

Figure 4.7 shows the bending moment at z-direction of cantilver steel beam, Hot Rolled IPE 120. The force used is 5000 Newton. The minimum banding moment of Z-direction is $-1.0 \times 10^4 \text{ N/mm}^2$. While the maximum bending moment of Z –direction is $0.204 \times 10^{-09} \text{ N/mm}^2$.

4.3.1.7 Bending Moment for forces of 5500N

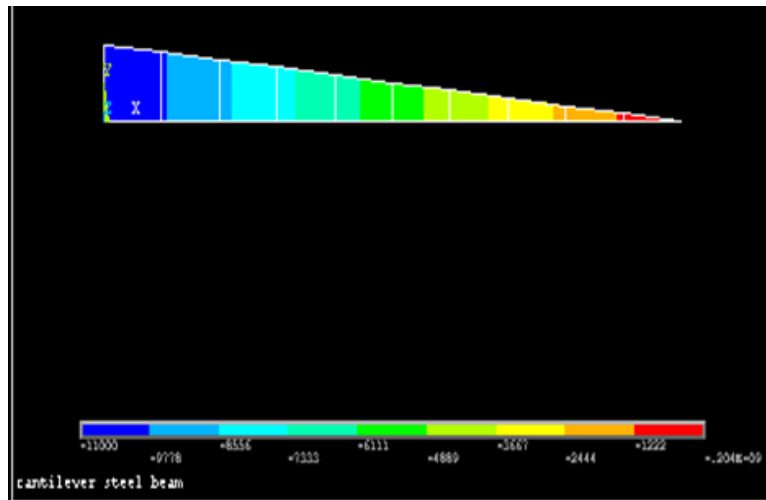


Figure 4.8: Bending Moment Z for Hot Rolled IPE 120 (5500N)

Figure 4.8 shows the bending moment at z-direction of cantilver steel beam, Hot Rolled IPE 120. The initial forces used are 5500 Newton. The minimum banding moment of Z-direction is $-11.0 \times 10^4 \text{ N/mm}^2$. While the maximum bending moment of Z-direction is $0.204 \times 10^{-09} \text{ N/mm}^2$.

4.3.1.8 Bending Moment for forces of 6000N

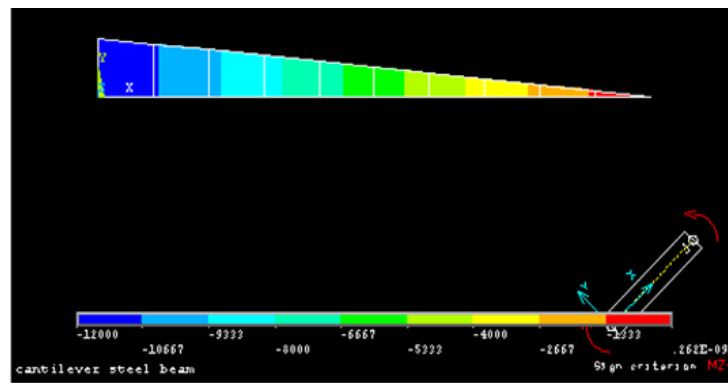


Figure 4.9: Bending Moment Z for Hot Rolled IPE 120 (6000N)

Figure 4.9 shows the bending moment at z-direction of cantilver steel beam, Hot Rolled IPE 120. The force used is 6000 Newton. The minimum banding moment of

Z-direction is $-12.0 \times 10^4 \text{ N/mm}^2$. While the maximum bending moment of Z – direction is $0.262 \times 10^{-09} \text{ N/mm}^2$.

4.3.1.9 Bending Moment for forces of 6500N

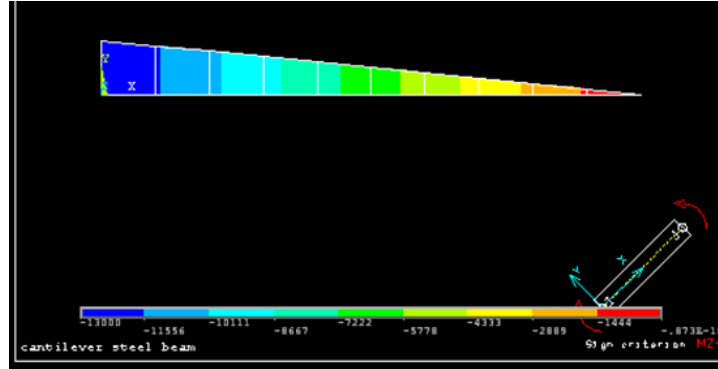


Figure 4.10: Bending Moment Z for Hot Rolled IPE 120 (6500N)

Figure 4.10 shows the bending moment at z-direction of cantilver steel beam, Hot Rolled IPE 120. The force used is 6500 Newton. The minimum banding moment of Z-direction is $-13.0 \times 10^4 \text{ N/mm}^2$. While the maximum bending moment of Z – direction is $0.873 \times 10^{-10} \text{ N/mm}^2$.

4.3.1.10 Bending Moment for forces of 7000N

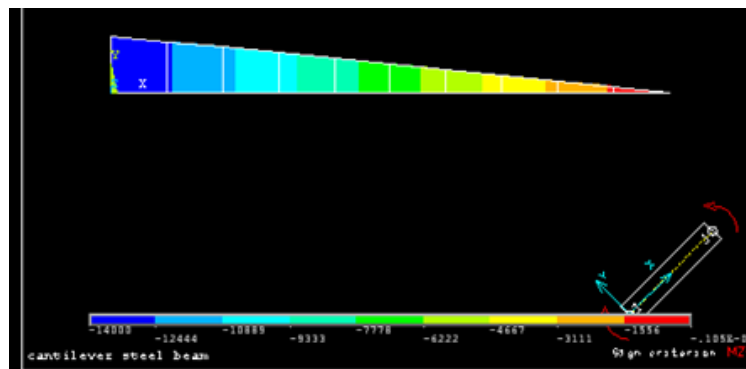


Figure 4.11: Bending Moment Z for Hot Rolled IPE 120 (7000N)

Figure 4.11 shows the bending moment at z-direction of cantilver steel beam, Hot Rolled IPE 120. The force used is 7000 Newton. The minimum banding moment of

Z-direction is $-14.0 \times 10^4 \text{ N/mm}^2$. While the maximum bending moment of Z – direction is $0.105 \times 10^{-08} \text{ N/mm}^2$.

4.4 Table of Stresses Result From Ansys

			Stress and Strain Sx in Different load (10 ⁶)(Pa)				
Element		Load/Forces N	2500	3000	3500	4000	4500
1	i	min	-94.399	-113.28	-132.16	-151.04	-169.92
		max	94.399	113.278	132.159	151.038	169.918
	j	min	-84.959	-101.95	-119	-135.94	-152.93
		max	84.959	101.951	118.995	135.935	152.926
2	i	min	-84.959	-101.95	-118.94	-135.94	-152.93
		max	84.959	101.951	118.943	135.935	152.926
	j	min	-75.519	-90.623	-105.73	-120.83	-135.94
		max	75.519	-90.623	105.727	120.831	135.935
3	i	min	-75.519	-90.623	-106.73	-120.83	-135.94
		max	75.519	-90.623	106.727	120.831	135.935
	j	min	-66.079	-79.295	-92.511	-105.73	-118.94
		max	66.079	79.295	92.511	105.727	118.943
4	i	min	-66.079	-79.295	-92.511	-105.73	-118.94
		max	66.079	79.295	92.511	105.727	118.943
	j	min	-56.639	-67.967	-79.295	-90.623	-101.95
		max	56.639	67.967	79.295	90.623	101.951
5	i	min	-56.639	-67.967	-79.295	-90.623	-101.95
		max	56.639	67.967	79.295	90.623	101.951
	j	min	-47.2	-56.639	-66.079	-75.519	-84.959
		max	47.2	56.639	66.079	75.519	84.959

Figure 4.12(a): Table of Stress of Cantiliver Steel beam, Hot Rolled IPE 120

6	i	min	-47.2	-56.639	-66.079	-75.519	-84.959
		max	47.2	56.639	66.079	75.519	84.959
	j	min	-37.76	-45.312	-52.863	-60.415	-67.967
		max	37.76	45.312	52.863	60.415	67.967
7	i	min	-37.76	-45.312	-52.863	-60.415	-67.967
		max	37.76	45.312	52.863	60.415	67.967
	j	min	-28.32	-33.984	-39.648	-45.312	-50.976
		max	28.32	33.984	39.648	45.312	50.976
8	i	min	-28.32	-33.984	-39.648	-45.312	-50.976
		max	28.32	33.984	39.648	45.312	50.976
	j	min	-16.88	-22.656	-26.432	-30.208	-33.984
		max	16.88	22.656	26.432	30.208	33.984
9	i	min	-16.88	-22.656	-26.432	-30.208	-33.984
		max	16.88	22.656	26.432	30.208	33.984
	j	min	-9.44	-11.328	-13.216	-15.104	-16.992
		max	9.44	11.328	13.216	15.104	16.992
10	i	min	-9.42	-11.328	-13.216	-15.104	-16.992
		max	9.42	11.328	13.216	15.104	16.992
	j	min	-1.923	-2.473	-9.891	-4.396	-7.422
		max	1.923	2.473	9.891	4.396	7.422

Figure 4.12(b): Table of Stress of Cantiliver Steel beam, Hot Rolled IPE 120

			Stress and Strain Sx in Different load (Pa)				
Element		Load/Forces N	5000	5500	6000	6500	7000
1	i	min	-188.8	-207.68	-226.56	-245.44	-264.32
		max	188.798	207.678	226.558	245.437	264.317
	j	min	-169.92	-186.91	-203.9	-220.89	-237.89
		max	169.918	186.91	203.902	220.894	237.885
2	i	min	-169.92	-186.91	-203.9	-220.89	-237.89
		max	169.918	186.91	203.902	220.894	237.885
	j	min	-151.04	-166.14	-181.25	-196.35	-211.45
		max	151.038	166.142	181.246	196.35	211.454
3	i	min	-151.04	-166.14	-181.25	-196.35	-211.45
		max	151.038	166.142	181.246	196.35	211.454
	j	min	-132.16	-145.37	-158.69	-171.81	-185.02
		max	132.159	145.374	158.69	171.806	185.022
4	i	min	-132.16	-145.37	-158.69	-171.81	-185.02
		max	132.159	145.374	158.69	171.806	185.022
	j	min	-113.28	-124.51	-135.94	-147.26	-158.59
		max	113.279	124.507	135.935	147.262	158.59
5	i	min	-113.28	-124.51	-135.94	-147.26	-158.59
		max	113.279	124.507	135.935	147.262	158.59
	j	min	-94.399	-103.84	-113.28	-122.72	-132.16
		max	94.399	103.839	113.279	122.719	132.159

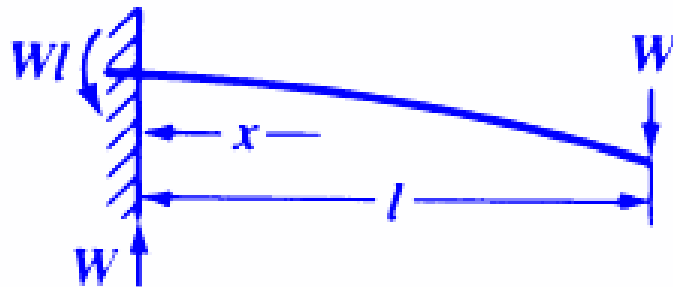
Figure 4.13(a): Table of Stress of Cantiliver Steel beam, Hot Rolled IPE 120

6	i	min	-94.399	-103.84	-113.28	-122.72	-132.16
		max	94.399	103.839	113.279	122.719	132.159
	j	min	-75.519	-83.071	-90.623	-98.176	-105.73
		max	75.519	83.071	90.623	98.176	105.727
7	i	min	-75.519	-83.071	-90.623	-98.176	-105.73
		max	75.519	83.071	90.623	98.176	105.727
	j	min	-56.639	-62.303	-67.967	-73.631	-79.295
		max	56.639	62.303	67.967	73.631	79.295
8	i	min	-56.639	-62.303	-67.967	-73.631	-79.295
		max	56.639	62.303	67.967	73.631	79.295
	j	min	-37.76	-41.536	-45.312	-49.088	-52.863
		max	37.76	41.536	45.312	49.088	52.863
9	i	min	-37.76	-41.536	-45.312	-49.088	-52.863
		max	37.76	41.536	45.312	49.088	52.863
	j	min	-19.88	-20.768	-22.656	-24.544	-26.432
		max	19.88	20.768	22.656	24.544	26.432
10	i	min	-19.88	-20.768	-22.656	-24.544	-26.432
		max	19.88	20.768	22.656	24.544	26.432
	j	min	-3.846	-3.846	-4.945	-1.648	-19.781
		max	3.846	3.846	4.945	1.648	19.781

Figure 4.13(b): Table of Stress of Cantiliver Steel beam, Hot Rolled IPE 120

4.5 Value of Stress From Theoretical Calculation

Stress for Cantilevered Beam With One Load Applied At End



4.14: Over Hanging Cantiliver Steel Beam

4.5.1 Calculation of Stress

Stress at specific point

$$S=W/Z(l-x) \quad (\text{Eq. 1})$$

Stress at the support (must be constant cross section)

$$S=Wl/Z \quad (\text{Eq. 2})$$

Section modulus of cross section of the beam

$$Z=I/z \quad (\text{Eq. 3})$$

Where,

W is Load/forces acting on beam

l is some distance as indicated

Z is section modulus of cross section of the beam, I/z

I is moment of inertia

Z is distances from natural axis to extrem fiber (edge)

$$h=120.0\text{E-}3$$

$$t_w=4.4\text{E-}3$$

$$b=63\text{E-}3$$

$$t_f=6.3\text{E-}3$$

$$d=93.4\text{E-}3$$

$$A=1.321\text{E-}3$$

$$I_{xx}=17.4\text{E-}9$$

$$I_{yy}=276.7\text{E-}9$$

$$I_{zz}=3.17\text{E-}6$$

$$I = \sum (I_i + A_i D_i^2), i=(1,2,3...) \quad (\text{Eq. 4})$$

Where,

I_i = The moment of inertia of the individual segment about its own centroid axis

A_i = The area of the individual segment

D_i = The vertical distance from the centroid of the segment to the neutral axis (NA)

To get I total, divide the flange into some segment,

I total= 2.9231E-6

4.5.2 Theoretical Value of Stress

Load/forces N	Stress and Strain S_x in Different load (M)(Pa)									
	1	2	3	4	5	6	7	8	9	10
2500	92.364	82.102	71.839	128.28	102.627	41.051	30.788	20.525	10.263	102.627
3000	110.837	98.522	86.207	78.207	61.576	49.261	36.946	24.631	12.315	123.153
3500	129.31	114.943	100.575	86.207	71.839	57.471	43.103	28.736	14.368	209.986
4000	147.784	131.363	114.943	98.522	82.102	65.682	42.261	32.841	16.42	239.984
4500	161.256	147.784	129.311	110.838	92.365	73.892	55.419	36.946	18.473	269.982
5000	184.729	164.203	143.678	123.152	102.627	82.102	61.576	41.051	20.525	299.979
5500	203.2	180.624	158.045	135.467	112.889	65.682	158.045	45.156	22.578	329.977
6000	221.675	197.145	172.414	147.784	123.153	98.522	73.892	49.261	24.631	7.389
6500	240.148	213.464	186.781	160.098	133.415	106.732	80.049	53.366	26.683	389.97
7000	258.62	258.62	201.149	172.414	143.678	114.942	80.207	57.47	28.736	419.971

Figure 4.15 : Value of Stress and Strain From Theoretical Calculation Hot Rolled IPE 120

Figure 4.13 and 4.14 shows the result of stress and strain from ANSYS software. All boundary condition have been setting out properly to get these output. The result was given in range form of maximum and minimum values. While figure 4.15 shown the theoretical values of stress and strain of cantiliver steel beam Hot Rolled IPE 120. There are some different between the theoretical and ANSYS simulation result. A theoretical analysis would only include reference to those works that are

necessary for the analysis (although subjectively omitting works that may run counter to the analysis is not acceptable in a scientific work). The analysis should be rigorous. A conceptual analysis may be the same as a theoretical analysis, but this category allows for more flexibility and less rigour. (J.H. Martin Willison et. al., 2014).

4.6 Graph Result of Stress and Strain

4.6.1 Graph Result of Stress and Strain for 2500N of load

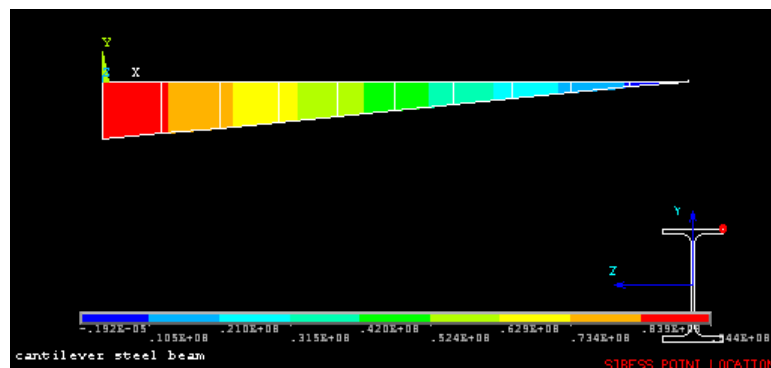


Figure 4.16 : Graph of Stress SX at Point 50 (2500N)

Figure 4.16 shows the minimum and maximum stress for I beam IPE 120 under 2500 Newton of load. The figure clearly shows the minimum stress bear for I beam IPE 120 under 2500 newton load is $-0.192 \times 10^{-5} \text{ N/mm}^2$. The maximum stress bear by I beam IPE 120 is $0.944 \times 10^8 \text{ N/mm}^2$.

The initila loads use in the simulation test is 2500 Newton. The result is due to the principle of stgress which gives meanof load distributed along the cross=sectional area of structure. Futhermore comparison is made among the different amount of load use in this test. By increasing the load from 2500 N to 7000 N, the maximum stress are found to be increased.

4.6.2 Graph Result of Stress and Strain for 3000N of load

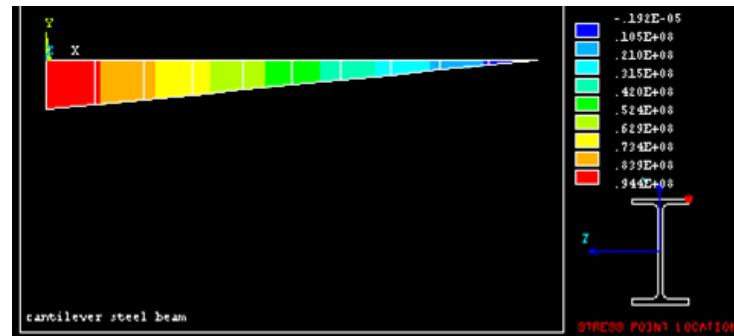


Figure 4.17 : Graph of Stress SX at Point 50 (3000N)

Figure 4.17 shows the minimum and maximum stress for I beam IPE 120 under 3000 Newton of load. The figure clearly shows the minimum stress bear for I beam IPE 120 under 3000 newton load is $-0.192 \times 10^{-5} \text{ N/mm}^2$. The maximum stress bear by I beam IPE 120 is $0.944 \times 10^8 \text{ N/mm}^2$. The minimum and maximum stress for 3000 Newton of load is same like stress result of 2500 newton.

4.6.3 Graph Result of Stress and Strain for 3500N of load

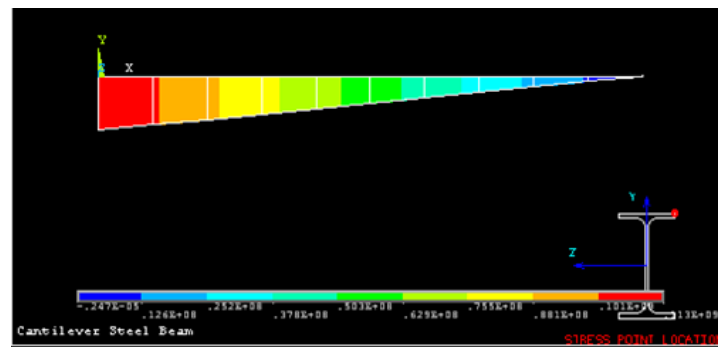


Figure 4.18 : Graph of Stress SX at Point 50 (3500N)

Figure 4.18 shows the minimum and maximum stress for I beam IPE 120 under 3500 Newton of load. The figure clearly shows the minimum stress bear for I beam IPE 120 under 2500 newton load is $-0.247 \times 10^{-5} \text{ N/mm}^2$. The maximum stress bear by I beam IPE 120 is $0.113 \times 10^9 \text{ N/mm}^2$.

4.6.4 Graph Result of Stress and Strain for 4000N of load

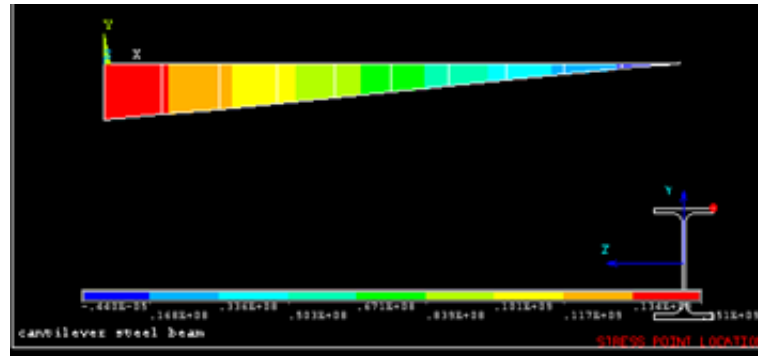


Figure 4.19 : Graph of Stress SX at Point 50 (4000N)

Figure 4.19 shows the minimum and maximum stress for I beam IPE 120 under 4000 Newton of load. The figure clearly shows the minimum stress bear for I beam IPE 120 under 2500 newton load is $-0.989 \times 10^{-5} \text{ N/mm}^2$. The maximum stress bear by I beam IPE 120 is $0.121 \times 10^8 \text{ N/mm}^2$.

4.6.5 Graph Result of Stress and Strain for 4500N of load

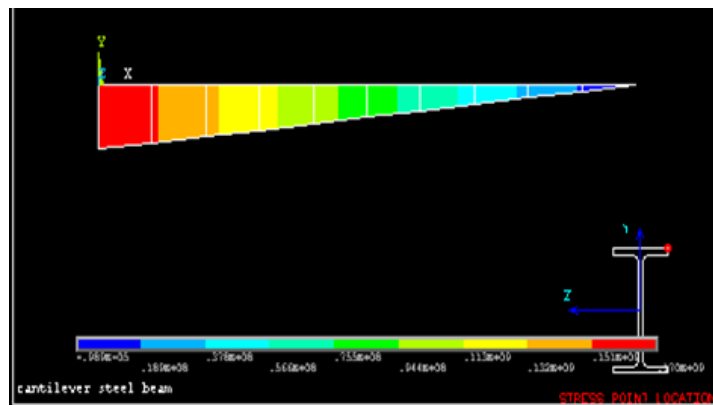


Figure 4.20 : Graph of Stress SX at Point 50 (4500N)

Figure 4.20 shows the minimum and maximum stress for I beam IPE 120 under 4500 Newton of load. The figure clearly shows the minimum stress bear for I beam IPE 120 under 2500 newton load is $-0.981 \times 10^{-5} \text{ N/mm}^2$. The maximum stress bear by I beam IPE 120 is $0.170 \times 10^9 \text{ N/mm}^2$.

4.6.6 Graph Result of Stress and Strain for 5000N of load

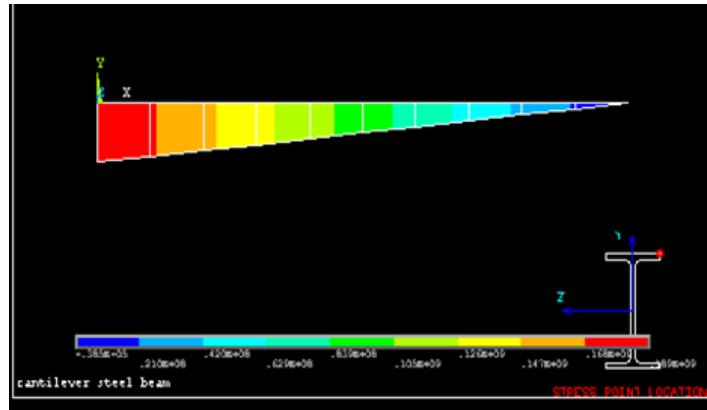


Figure 4.21 : Graph of Stress SX at Point 50 (5000N)

Figure 4.21 shows the minimum and maximum stress for I beam IPE 120 under 5000 Newton of load. The figure clearly shows the minimum stress bear for I beam IPE 120 under 5000 newton load is $-0.385 \times 10^{-5} \text{ N/mm}^2$. The maximum stress bear by I beam IPE 120 is $0.185 \times 10^9 \text{ N/mm}^2$.

4.6.7 Graph Result of Stress and Strain for 5500N of load

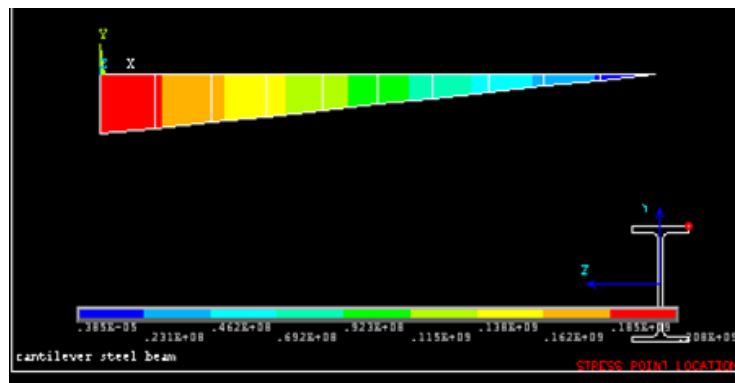


Figure 4.22 : Graph of Stress SX at Point 50 (5500N)

Figure 4.22 shows the minimum and maximum stress for I beam IPE 120 under 5500 Newton of load. The figure clearly shows the minimum stress bear for I beam IPE 120 under 5500 newton load is $-0.385 \times 10^{-5} \text{ N/mm}^2$. The maximum stress bear by I beam IPE 120 is $0.208 \times 10^9 \text{ N/mm}^2$.

4.6.8 Graph Result of Stress and Strain for 6000N of load

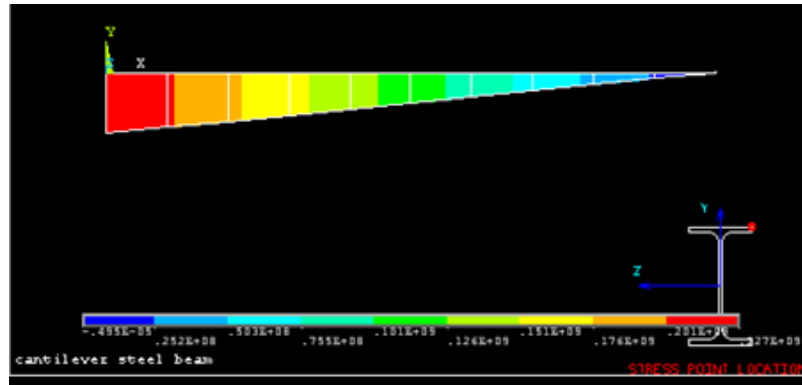


Figure 4.23 : Graph of Stress SX at Point 50 (6000N)

Figure 4.23 shows the minimum and maximum stress for I beam IPE 120 under 6000 Newton of load. The figure clearly shows the minimum stress bear for I beam IPE 120 under 6000 newton load is $-0.495 \times 10^{-5} \text{ N/mm}^2$. The maximum stress bear by I beam IPE 120 is $0.227 \times 10^9 \text{ N/mm}^2$.

4.6.9 Graph Result of Stress and Strain for 6500N of load

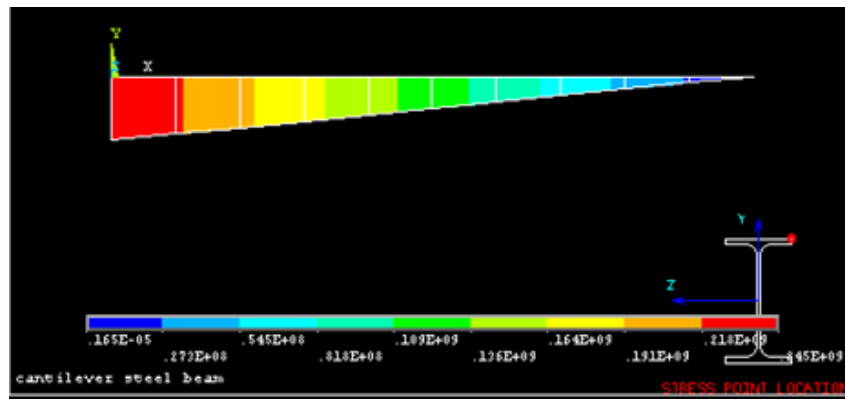


Figure 4.24 : Graph of Stress SX at Point 50 (6500N)

Figure 4.16 shows the minimum and maximum stress for I beam IPE 120 under 6500 Newton of load. The figure clearly shows the minimum stress bear for I beam IPE 120 under 6500 newton load is $0.165 \times 10^{-5} \text{ N/mm}^2$. The maximum stress bear by I beam IPE 120 is $0.245 \times 10^9 \text{ N/mm}^2$.

4.6.10 Graph Result of Stress and Strain for 7000N of load

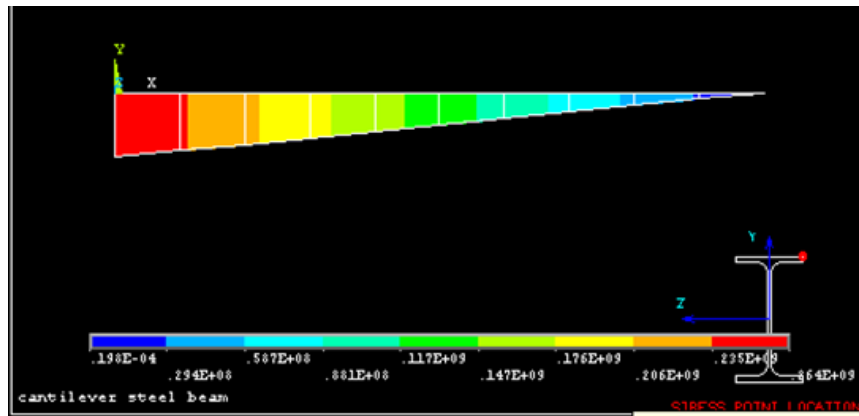


Figure 4.25 : Graph of Stress SX at Point 50 (7000N)

Figure 4.16 shows the minimum and maximum stress for I beam IPE 120 under 7000 Newton of load. The figure clearly shows the minimum stress bear for I beam IPE 120 under 7000 newton load is $0.198 \times 10^{-4} \text{ N/mm}^2$. The maximum stress bear by I beam IPE 120 is $0.264 \times 10^9 \text{ N/mm}^2$.

4.7 Graph Relationship of Load and Maximum Stress

Maximum Stress vs Load for Cantiliver Steel I Beam IPE 120

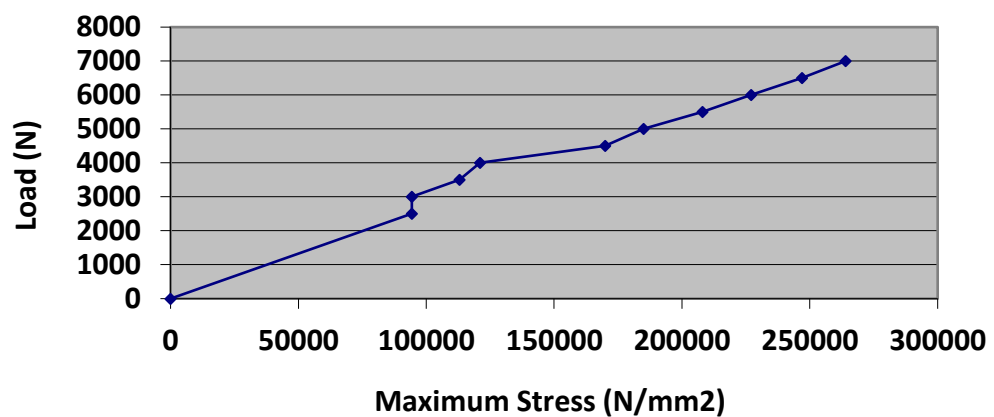


Figure 4.26: Maximum Stress vs Load for Cantiliver Steel I Beam IPE 120

Figure 4.26 shows the relation of cantiliver steel I beam IPE 120 between load factor and stress. The graph shows the maximum stress is not directly prportional to the load applied. Hence the maximum stress are not increased uniformly when the total load applied is increased.

Accroding to Bayan et. al., (2011), the applied load increased gradually until unloading occurred where continuous measurements were taken for the applied loads, the deflections of the loaded points including the rotation of the members throughout the test.

CHAPTER 5

CONCLUSION AND RECOMMENDATIONS

5.1 Introduction

Cantiliver Steel structure is suitable for building construction for a certain condition to their high structural performance and durability. The objectives of this present study are to analyze the forces and stresses in a selective size steel cantilever beam and find the maximum vertical force (Newton) for beam which is hot rolled shape IPE 120 and is made of Fe430 steel by using Finite Element Method (FEM). Linear and nonlinear analysis has been done to analyze the deformed mesh, displacement, stress and strain between different value of load applied at end of overhanging cantiliver steel beam. The analysis data was conducted to obtain the compressive conclusion towards achieving the objective of the project.

5.2 Conclusion

The following conclusions are drawn from the test result and discussion obtained several conclusions for this project are prepared, which are:

- i. Based on the Stress and Strain Result from ANSYS software the maximum moment for Cantiliver Steel I Beam IPE 120 increased as the load applied increases, but not linearly.

- ii. In the range of load between 2500N to 7000N, the most minimum moment cantiliver steel I beam IPE 120 is when the load applied is 7000N.
- iii. In the range of load between 2500N to 7000N, the most maximum moment for cantiliver steel I beam IPE 120 is when the load applied is 7000N.
- iv. Based on the Stress and Strain Result from ANSYS software the maximum stress fro cantiliver steel I beam IPE 120 increased as the load applied increases, but not linearly.
- v. The most minimum stress and strain occured at load applied about 7000N.
- vi. The most maximum stress and strain occured at load applied aboiut 7000N.

5.3 Recommendations

Futher studies on the research need to be conducted in the future in order to come out with better and good results. Several recommendations are proposed for the future studies of this analysis of cantiliver steel I beam IPE 120 using ANSYS software. The following recommendations might be useful for the future investigations to achieve the objectives of this research :

- i. Different types of beam size can be done to get better results.
- ii. The study can be continued by using a variety of steel type to investigate their effect on the result of stress and strain .
- iii. Different boundary condition can be done to get the better results.
- iv. Other types of element can be used for the same types of I beam.

- v. Laboratory test should be done parallel with the ANSYS modelling process to ensure that all the parameters needed for computer input are available.
- vi. A larger capacity of computer should be prepared for ANSYS analysis as the analysis work involved is too heavy.

REFERENCES

- A.M Jamadar, P. K. (2014). Steel Beam. *International Journal of Engineering and Innovative Thechnology*, Vol 2, Issue 1.
- Bayan, A. S. (2010). Cold-Frame Steel Frame with Bolted Moment Connection. *International Journal of Civil and Structural Engineering*, Vol. 1, No. 3.
- Bayan, A. S. (2011). Finite Element Analysis of Cold- Formed Steel Connection. *International Journal of Civil and Structure Engineering*, Vol. 5 No.2.
- Chri, B. T. (2013). The Bending of Beam and The Second Moment of Area. *The Plymouth Studient Scientisrt*, 328-339.
- Crisan A, U. V. (2012). Behaviour of Cold-Formed Steel Perforated Sections In Compression. *Experimental Investigations*, 86-96.
- Davidson, B. &. (2012). STEEL DESIGNERS' MANUAL. *BLACKWELL PUBLISHING LTD*, 40-62.
- Elsayed, M. M.-E.-E. (2012). Finite Element Analysis of Beam-to-Column Joints. *Alexandra Engineering Journal*, , 91-104.
- G Lackshmi, N. (2009). Finite Element Analysis. *BS Publications*, 20-44.
- H, S. K. (2008). Lateral Torsional Buckling of Steel Beam. *Iranian Journal of Sciences & Thechnology, Transaction B, Engineering*, Vol 32, No. B2, pp 153-156.
- L, H.-H. (2010). *Finite Element Simulation*. ANSYS Workbench 14: SDC Publication.
- Lee, H.-H. (2011). *Finite Element Simulations with ANSYS Workbench 15*. Kuala Lumpur: SDC Publication.
- Mitra, N. (2014). Nomograph for Design of Channel Section Beam. *Engineering Journal, American Institute of Steel Construction*, Vol. 12, pp.68-69.
- Mohamed, H.-Z. D. (2009). Non Linear Finite Element Analysis. *Structure Engineering*, 353-368.
- Ross, M. J. (2014). Optimal Design Of Long- Span Steel Portal Frames Using Fabricated Beams. *Journal of Constructional Steel Research*, 104-114.
- S. M. Musa, A. V. (2011). Finite Element Analysis For Structure. *Finite Element Analysis: A Primer*, 50-62.
- Sariffudin, B. A. (2011). Force and Moment of Cantilever Steel Beam. *Finite Element Analysis*, 234-321.

- Tan., L. &. (2009). Experimental Studies on Steel Beam Curved in P.Lan. *Journal of Structural Engineering*, 324-356.
- Varma, P. D. (2015). Design of Beam Flexure and Shear 2.1. *Design of Beam Structure*, Chapter 2.
- Xiang., Y. L. (2017). Stress and Strain Curves for Hot-Rolled Steels. *Journal of Constructional Steel Research*, 422-432.
- Xiaoyue Zhang, a. M. (2017). The Influence of Different Steel Beam Profiles. *Canadian Journal of Civil Engineering*, 611-618.
- Yu C., X. C. (2018). Test and Behaviour of Hot- Rolled steel Beam with Flange Upstands. *Journal of Constructional of Steel Research* , Vol. 144, pp. 53-64.
- Zienkiewicz, O. a. (2014). The Finite Element Method. *Butherworth-Heinemann*, 5th edition.
- A.M Jamadar, P. K. (2014). Steel Beam. *International Journal of Engineering and Innovative Thechnology*, Vol 4, Issue 4.
- Chri, B. T. (2013). The Bending of Beam and The Second Moment of Area. *The Plymouth Student Scientist*, 403-504.
- Davidson, B. &. (2012). STEEL DESIGNERS' MANUAL. *BLACKWELL PUBLISHING LTD*, 44-56.
- G Lackshmi, N. (2009). Finite Element Analysis. *BS Publications*, 30-56.
- Mitra, N. (2014). Nomograph for Design of Channel Section Beam. *Engineering Journal, American Institute of Steel Construction*, Vol. 10, pp.11-34.
- Mohamed, H.-Z. D. (2009). Non Linear Finite Element Analysis. *Structure Engineering*, 456-653.
- S. M. Musa, A. V. (2011). Finite Element Analysis For Structure. *Finite Element Analysis: A Primer*, 75-178.
- Xiang., Y. L. (2017). Stress and Strain Curves for Hot-Rolled Steels. *Journal of Constructional Steel Research*, 543-567.
- Xiaoyue Zhang, a. M. (2017). The Influence of Different Steel Beam Profiles. *Canadian Journal of Civil Engineering*, 611-618.
- Yu C., X. C. (2018). Test and Behaviour of Hot- Rolled steel Beam with Flange Upstands. *Journal of Constructional of Steel Research* , Vol. 12, pp. 12-34.
- Zienkiewicz, O. a. (2014). The Finite Element Method. *Butherworth-Heinemann*, 5th edition.

APPENDIX A SAMPLE APPENDIX 1

THEORETICAL CALCULATION OF STRESS

Stress at specific point

$$S = W/Z(l-x)$$

Stress at the support (must be constant cross section)

$$S = Wl/Z$$

Where,

W is Load/forces acting on beam

l is some distance as indicated

Z is section modulus of cross section of the beam, I/z

I is moment of inertia

Z is distances from natural axis to extrem fiber (edge)

$$h = 120.0E-3$$

$$t_w = 4.4E-3$$

$$b = 63E-3$$

$$t_f = 6.3E-3$$

$$d = 93.4E-3$$

$$A = 1.321E-3$$

$$I_{xx} = 17.4E-9$$

$$I_{yy} = 276.7E-9$$

$$I_{zz} = 3.17E-6$$

$$I_{\text{total}} = \text{Submation of } (I_i + A_i d_i^2)$$

Where,

I_i = The moment of inertia of the individual segment about its own centroid axis

A_i = The area of the individual segment

D_i = The vertical distance from the centroid of the segmnet the neutral axis (NA)

To get I total, divide the flange into some segment,

Segment 1

$$b=64\text{E-}3, h=6.3\text{E-}3$$

$$I1=b(h^3)$$

$$I1=1.334\text{E-}9$$

$$A1=b \times h$$

$$A1=4.032\text{E-}9$$

$$d_1=y_1-Y$$

$$d_1=0.05685$$

Segment 2

$$b=4.4\text{E-}3, h=107.4\text{E-}3$$

$$I2=b(h^3)$$

$$I2=4.542\text{E-}7$$

$$A2=b \times h$$

$$A2=4.726\text{E-}4$$

$$d_2=y_2-Y$$

$$d_2=0$$

Segment 3

$$b=64\text{E-}3, h=6.3\text{E-}3$$

$$I3=b(h^3)$$

$$I3=1.334\text{E-}9$$

$$A3=b \times h$$

$$A3=4.032\text{E-}4$$

$$d_3=y_3-Y$$

$$d_3=0.0537$$

$$I_{\text{total}}= 2.9231\text{E-}6$$

Calculation of Stress

Initial value of force= 2500N

(Stress at specific point)

$$S=W/Z(i-x)$$

$$i=2.0$$

0.2m from loading(element 1)

$$x=0.2$$

$$W=2500$$

$$Z=I/z$$

$$Z=4.872E-5$$

$$S=92.364E6 \text{ Pa}$$

0.4m from loading (element 2)

$$x=0.4$$

$$W=2500$$

$$Z=I/z$$

$$Z=4.872E-5$$

$$S=82.102E6 \text{ Pa}$$

0.6m from loading(element 3)

$$x=0.6$$

$$W=2500$$

$$Z=I/z$$

$$Z=4.872E-5$$

$$S=71.839E6 \text{ Pa}$$

0.8m from loading (element 4)

$$x=0.8$$

$$W=2500$$

$$Z=I/z$$

$$Z=4.872E-5$$

$$S=128.28E6 \text{ Pa}$$

1.0m from loading (element 5)

$$x=1.0$$

$$W=2500$$

$$Z=I/z$$

$$Z=4.872E-5$$

$$S=102.627E6 \text{ Pa}$$

1.2m from loading (element 6)

$$x=1.2$$

$$W=2500$$

$$Z=I/z$$

$$Z=4.872E-5$$

$$S=41.051E6 \text{ Pa}$$

1.4m from loading (element 7)

$$x=1.4$$

$$W=2500$$

$$Z=I/z$$

$$Z=4.872E-5$$

$$S=30.788E6 \text{ Pa}$$

1.6m from loading (element 8)

$$x=1.6$$

$$W=2500$$

$$Z=I/z$$

$$Z=4.872E-5$$

$$S=20.525E6 \text{ Pa}$$

1.8m from loading (element 9)

$$x=1.8$$

$$W=2500$$

$$Z=I/z$$

$$Z=4.872E-5$$

$$S=10.263\text{E}6 \text{ Pa}$$

Stress at Support (element 10)

$$S=Wl/Z$$

$$W=2500$$

$$L=2.0$$

$$Z=4.872\text{E}-5$$

$$S=102.627\text{E}6 \text{ Pa}$$

APPENDIX B SAMPLE APPENDIX 2

BENDING OF EXTREAM STRESS ANSYS TABLE

BEAM ELEMENT 1, END I, SECTION 1 - STRESSES AND STRAINS

	SX (Pa)
Minimum Value	-94.399E+06
Location of Min. Val.	111
Maximum Value	94.399E+06
Location of Max. Val.	1
Maximum Absolute Value	94.399E+06
Location of Max. Abs. Val.	1

BEAM ELEMENT 1, END J, SECTION 1 - STRESSES AND STRAINS

	SX (Pa)
Minimum Value	-84.959E+06
Location of Min. Val.	111
Maximum Value	84.959E+06
Location of Max. Val.	1
Maximum Absolute Value	84.959E+06
Location of Max. Abs. Val.	1

Figure B1: 2500N LOAD

BEAM ELEMENT 2, END I, SECTION 1 - STRESSES AND STRAINS

	SX (Pa)
Minimum Value	-84.959E+06
Location of Min. Val.	111
Maximum Value	84.959E+06
Location of Max. Val.	1
Maximum Absolute Value	84.959E+06
Location of Max. Abs. Val.	1

BEAM ELEMENT 2, END J, SECTION 1 - STRESSES AND STRAINS

	SX (Pa)
Minimum Value	-75.519E+06
Location of Min. Val.	111
Maximum Value	75.519E+06
Location of Max. Val.	1
Maximum Absolute Value	75.519E+06
Location of Max. Abs. Val.	1

Figure B2: 3000N LOAD

BEAM ELEMENT 3, END I, SECTION 1 - STRESSES AND STRAINS

	SX (Pa)
Minimum Value	-75.519E+06
Location of Min. Val.	111
Maximum Value	75.519E+06
Location of Max. Val.	1
Maximum Absolute Value	75.519E+06
Location of Max. Abs. Val.	1

BEAM ELEMENT 3, END J, SECTION 1 - STRESSES AND STRAINS

	SX (Pa)
Minimum Value	-66.079E+06
Location of Min. Val.	111
Maximum Value	66.079E+06
Location of Max. Val.	1
Maximum Absolute Value	66.079E+06
Location of Max. Abs. Val.	1

Figure B3: 3500N LOAD

BEAM ELEMENT 4, END I, SECTION 1 - STRESSES AND STRAINS

	SX (Pa)
Minimum Value	-66.079E+06
Location of Min. Val.	111
Maximum Value	66.079E+06
Location of Max. Val.	1
Maximum Absolute Value	66.079E+06
Location of Max. Abs. Val.	1

BEAM ELEMENT 4, END J, SECTION 1 - STRESSES AND STRAINS

	SX (Pa)
Minimum Value	-56.639E+06
Location of Min. Val.	111
Maximum Value	56.639E+06
Location of Max. Val.	1
Maximum Absolute Value	56.639E+06
Location of Max. Abs. Val.	1

Figure B4: 4000N LOAD

BEAM ELEMENT 5, END I, SECTION 1 - STRESSES AND STRAINS

	SX (Pa)
Minimum Value	-56.639E+06
Location of Min. Val.	111
Maximum Value	56.639E+06
Location of Max. Val.	1
Maximum Absolute Value	56.639E+06
Location of Max. Abs. Val.	1

BEAM ELEMENT 5, END J, SECTION 1 - STRESSES AND STRAINS

	SX (Pa)
Minimum Value	-47.200E+06
Location of Min. Val.	111
Maximum Value	47.200E+06
Location of Max. Val.	1
Maximum Absolute Value	47.200E+06
Location of Max. Abs. Val.	1

Figure B5: 4500N LOAD

BEAM ELEMENT 6, END I, SECTION 1 - STRESSES AND STRAINS

	SX (Pa)
Minimum Value	-47.200E+06
Location of Min. Val.	111
Maximum Value	47.200E+06
Location of Max. Val.	1
Maximum Absolute Value	47.200E+06
Location of Max. Abs. Val.	1

BEAM ELEMENT 6, END J, SECTION 1 - STRESSES AND STRAINS

	SX (Pa)
Minimum Value	-37.760E+06
Location of Min. Val.	111
Maximum Value	37.760E+06
Location of Max. Val.	1
Maximum Absolute Value	37.760E+06
Location of Max. Abs. Val.	1

Figure B6: 5000N LOAD

BEAM ELEMENT 7, END I, SECTION 1 - STRESSES AND STRAINS

	SX (Pa)
Minimum Value	-37.760E+06
Location of Min. Val.	111
Maximum Value	37.760E+06
Location of Max. Val.	1
Maximum Absolute Value	37.760E+06
Location of Max. Abs. Val.	1

BEAM ELEMENT 7, END J, SECTION 1 - STRESSES AND STRAINS

	SX (Pa)
Minimum Value	-28.320E+06
Location of Min. Val.	111
Maximum Value	28.320E+06
Location of Max. Val.	1
Maximum Absolute Value	28.320E+06
Location of Max. Abs. Val.	1

Figure B7: 5500N LOAD

BEAM ELEMENT 8, END I, SECTION 1 - STRESSES AND STRAINS

	SX (Pa)
Minimum Value	-28.320E+06
Location of Min. Val.	111
Maximum Value	28.320E+06
Location of Max. Val.	1
Maximum Absolute Value	28.320E+06
Location of Max. Abs. Val.	1

BEAM ELEMENT 8, END J, SECTION 1 - STRESSES AND STRAINS

	SX (Pa)
Minimum Value	-18.880E+06
Location of Min. Val.	111
Maximum Value	18.880E+06
Location of Max. Val.	1
Maximum Absolute Value	18.880E+06
Location of Max. Abs. Val.	1

Figure B8: 6000N LOAD

BEAM ELEMENT 9, END I, SECTION 1 - STRESSES AND STRAINS

	SX (Pa)
Minimum Value	-18.880E+06
Location of Min. Val.	111
Maximum Value	18.880E+06
Location of Max. Val.	1
Maximum Absolute Value	18.880E+06
Location of Max. Abs. Val.	1

BEAM ELEMENT 9, END J, SECTION 1 - STRESSES AND STRAINS

	SX (Pa)
Minimum Value	-9.440E+06
Location of Min. Val.	111
Maximum Value	9.440E+06
Location of Max. Val.	1
Maximum Absolute Value	9.440E+06
Location of Max. Abs. Val.	1

Figure B9: 6500N LOAD

BEAM ELEMENT 10, END I, SECTION 1 - STRESSES AND STRAINS

	SX (Pa)
Minimum Value	-9.440E+06
Location of Min. Val.	111
Maximum Value	9.440E+06
Location of Max. Val.	1
Maximum Absolute Value	9.440E+06
Location of Max. Abs. Val.	1

BEAM ELEMENT 10, END J, SECTION 1 - STRESSES AND STRAINS

	SX (Pa)
Minimum Value	-1.923E-06
Location of Min. Val.	1
Maximum Value	1.923E-06
Location of Max. Val.	111
Maximum Absolute Value	1.923E-06
Location of Max. Abs. Val.	1

Figure B10: 7000N LOAD