



FINITE ELEMENT SIMULATION OF CASTELLATED BEAM BY USING  
CIVILFEM WITH ANSYS SOFTWARE

TAN POH AUN

A thesis submitted in fulfilment of the requirements for the award of the degree of

B. ENG (HONS.) CIVIL ENGINEERING

Faculty Of Civil Engineering and Earth Resources

UNIVERSITI MALAYSIA PAHANG

JUNE 2014

## ABSTRACT

Nowadays in the construction of modern buildings, it is necessary to accommodate pipes and ducts necessary services, such as air conditioning, water supply, sewerage, electricity, computer networks and telephone networks. Finite element method is the leading technique for analysing the behaviour of structures when subjected to a variety of loads. In order to minimize storey height and achieve economic requirements most of industrial building using castellated beam. Besides that, the intention of castellated beam used in the steel structure is without any additional material also will increase the beam's web depth as well as the strength of the beam. The finite element results are compared in term of load deflection curve, pure bending, shear, and lateral torsional buckling, showing a good agreement. Hence, from the results this study was conducted that finite element analysis is able to give a similar result as the analytical results. Moreover, this results presents the castellated beam results by using the ANSYS software. The objective of this study is to obtain the maximum deflection, maximum von mises strain, maximum von mises stress by Probabilistic Design System (PDS)

## ABSTRAK

Kini dalam pembinaan bangunan-bangunan moden, adalah perlu untuk menampung paip dan saluran perkhidmatan yang diperlukan, seperti penyaman udara, bekalan air, pembedungan, elektrik, rangkaian komputer dan rangkaian telefon. Kaedah unsur terhingga adalah teknik utama untuk menganalisis kelakuan struktur apabila dikenakan pelbagai beban. Dalam usaha untuk mengurangkan ketinggian tingkat dan mencapai keperluan ekonomi yang paling bangunan industri yang menggunakan rasuk istana. Selain itu, niat untuk rasuk istana digunakan dalam struktur keluli adalah tanpa apa-apa bahan tambahan juga akan meningkatkan kedalaman web rasuk dan juga kekuatan rasuk. Keputusan unsur terhingga dibandingkan dari segi lengkung beban pesongan, lenturan tulen, ricih, dan lengkukan kilasan sisi, menunjukkan perjanjian yang baik. Oleh itu, daripada hasil kajian ini telah dijalankan bahawa analisis unsur terhingga mampu memberikan hasil yang sama seperti hasil analisis. Lebih-lebih lagi, ini menyebabkan membentangkan keputusan rasuk istana dengan menggunakan perisian ANSYS itu. Objektif kajian ini adalah untuk mendapatkan pesongan maksimum, maksimum von Mises terikan, maksimum von Mises tekanan dengan Rekabentuk Sistem berkebarangkalian (PDS).

## TABLE OF CONTENTS

<b>SUPERVISOR’S DECLARATION</b> .....	<b>ii</b>
<b>STUDENT’S DECLARATION</b> .....	<b>iii</b>
<b>ACKNOWLEDGEMENT</b> .....	<b>iv</b>
<b>ABSTRACT</b> .....	<b>v</b>
<b>ABSTRAK</b> .....	<b>vi</b>
<b>LIST OF FIGURES</b> .....	<b>x</b>
<b>LIST OF TABLES</b> .....	<b>xiii</b>

### CHAPTER 1: INTRODUCTION

1.1 General .....	1
1.2 Problem statement .....	2
1.3 Objectives of research .....	2
1.4 Scope of study .....	3
1.5 Expected Outcome .....	3

### CHAPTER 2: LITERATURE REVIEW

2.1 Introduction .....	2
2.2 Castellated Beam Analysis.....	2
2.3 Finite Element Method (FEM).....	6
2.4 ANSYS.....	8

### CHAPTER 3: METHODOLOGY

3.1 Introduction .....	5
3.2 Method use to Analyse Structure .....	12
3.3 Design process.....	13

3.3.1	ANSYS Preprocessor: Defining the problem.....	13
3.3.2	ANSYS Solution Processor: Assigning loads and solving.....	13
3.3.3	ANSYS Post-processing: Viewing the results.....	14
3.4	ANSYS Preprocessor Process.....	14
3.4.1	Defining an Analysis Title.....	14
3.4.2	Defining Element Type and Real Constants.....	15
3.4.3	Defining Material Properties.....	16
3.4.4	Creating the Model Geometry.....	17
3.4.5	Meshing.....	19
3.5	ANSYS Solution Processor.....	20
3.5.1	Define Analysis Type.....	20
3.5.2	Boundary Condition.....	22
3.5.3	Applying Load.....	23
3.5.4	Solving.....	23
3.6	ANSYS Post-Processing.....	24
3.6.1	Deformed Shape.....	24
3.6.2	Displacement.....	25
3.7	Probabilistic Design System (PDS).....	26

## **CHAPTER 4: RESULT ANALYSIS AND DISCUSSION**

4.0	Introduction.....	27
4.2	Probabilistic Model Information.....	28
4.2.1	Random Input Variables.....	28
4.2.2	Correlations.....	32
4.2.3	Probabilistic Analysis Summary.....	32
4.2.4	Probabilistic Analysis Results.....	33
4.4	Sample History Plots.....	34
4.4.1	Sampled Values for Output Parameter MAXIMUM DEFLECTION.....	35
4.4.2	Sampled Value for Output Parameter MAX DEFLECTION.....	37
4.4.3	Sampled Values for Output Parameter MAXVONMISESSTRESS.....	39

4.4.4	Sampled Values for Output Parameter MAXVONMISESSTRAIN .....	41
4.5	Histogram Plots .....	43
4.5.1	Histogram of Input Variables .....	43
4.5.2	Histogram of Output Parameters .....	46
4.6	Cumulative Distribution Function Plots .....	48
4.6.1	Cumulative Distribution Function of Input Variables .....	49
4.6.2	Cumulative Distribution Function of Random Output Parameters .....	52
4.7	Sensitivity Plots .....	56
<b>CHAPTER 5: CONCLUSION AND RECOMMENDATION</b>		
5.1	Conclusion .....	60
5.2	Recommendation .....	61
<b>REFERENCES.....</b>		<b>62</b>
<b>APPENDIX A.....</b>		<b>64</b>
<b>APPENDIX B.....</b>		<b>71</b>

## LIST OF FIGURES

<b>Figure 2.1:</b> Vierendeel Truss Analogy.....	5
<b>Figure 2.2:</b> Mathematical Formulation of Castellated Beam.....	6
<b>Figure 2.3:</b> Standard Castellated Section.....	6
<b>Figure 2.4:</b> Linear algebraic equations.....	7
<b>Figure 2.5:</b> Non-composite finite element model.....	7
<b>Figure 2.6:</b> Castellated Beam model.....	8
<b>Figure 2.7:</b> Typical Finite Element Mesh for a Castellated Beam.....	9
<b>Figure 2.8:</b> Applying load on castellated beam.....	9
<b>Figure 3.1:</b> Flow chart of the modelling process.....	12
<b>Figure 3.2:</b> Change title steps.....	15
<b>Figure 3.3:</b> Change title window.....	15
<b>Figure 3.4:</b> Definition of element types to be used.....	16
<b>Figure 3.5:</b> Selection of element types from the library.....	16
<b>Figure 3.6:</b> Define Material Model Behaviour.....	17
<b>Figure 3.7:</b> Linear isotopic material properties.....	17
<b>Figure 3.8:</b> Create key point process.....	17
<b>Figure 3.9:</b> Area reflect in Y-Z Plane.....	18
<b>Figure 3.10:</b> Model line to area.....	19
<b>Figure 3.11:</b> Global Element Sizes.....	19
<b>Figure 3.12:</b> Model Meshing.....	20
<b>Figure 3.13:</b> Dialog boxes for selecting the type pf analysis for structural.....	20
<b>Figure 3.14:</b> Solution Controls dialog box.....	21
<b>Figure 3.15:</b> Boundary Condition for Support.....	22
<b>Figure 3.16:</b> Applying Load on Castellated Beam.....	23
<b>Figure 3.17:</b> ANSYS Responds.....	24
<b>Figure 3.18:</b> Deformed Shape.....	24
<b>Figure 3.19:</b> Deflection value in nodal solution.....	25
<b>Figure 4.1:</b> Model Geometry and Finite Element Mesh.....	28
<b>Figure 4.2:</b> PDF & CDF of Input Random Variable FORCE_1.....	29

<b>Figure 4.3:</b> PDF & CDF of Input Random Variable FORCE_2.....	30
<b>Figure 4.4:</b> PDF & CDF of Input Random Variable HEAT.....	30
<b>Figure 4.5:</b> PDF & CDF of Input Random Variable HOLE_RADIUS.....	31
<b>Figure 4.6:</b> PDF & CDF of Input Random Variable ELASTIC_MODULUS.....	31
<b>Figure 4.7:</b> PDF & CDF of Input Random Variable POISSON_RATIO.....	32
<b>Figure 4.8:</b> Sampled Values for Output Parameter MAXIMUM DEFLECTION.....	35
<b>Figure 4.9:</b> Mean Values for Output Parameter MAXIMUM DEFLECTION.....	36
<b>Figure 4.10:</b> Standard Deviation Values for Output Parameter MAXIMUM DEFLECTION.....	36
<b>Figure 4.11:</b> Sampled Values for Output Parameter MAX_DEFLECTION.....	37
<b>Figure 4.12:</b> Standard Deviation Values for Output Parameter MAX_DEFLECTION.....	38
<b>Figure 4.13:</b> Mean Values for Output Parameter MAX_DEFLECTION.....	38
<b>Figure 4.14:</b> Sampled Values for Output Parameter MAXVONMISESSTRESS.....	39
<b>Figure 4.15:</b> Standard Deviation Values for Output Parameter MAXVONMISESSTRESS.....	40
<b>Figure 4.16:</b> Mean Values for Output Parameter MAXVONMISESSTRESS.....	40
<b>Figure 4.17:</b> Sampled Values for Output Parameter MAXVONMISESSTRAIN.....	41
<b>Figure 4.18:</b> Standard Deviation Values for Output Parameter MAXVONMISESSTRAIN.....	42
<b>Figure 4.19:</b> Mean Values for Output Parameter MAXVONMISESSTRAIN.....	42
<b>Figure 4.20:</b> Histogram of Input Variable FORCE_1.....	43
<b>Figure 4.21:</b> Histogram of Input Variable FORCE_2.....	44
<b>Figure 4.22:</b> Histogram of Input Variable HEAT.....	44
<b>Figure 4.23:</b> Histogram of Input Variable HOLE_RADIUS.....	45
<b>Figure 4.24:</b> Histogram of Input Variable ELASTIC_MODULUS.....	45
<b>Figure 4.25:</b> Histogram of Input Variable POISSON_RATIO.....	46
<b>Figure 4.26:</b> Histogram of MAXIMUM DEFLECTION.....	46
<b>Figure 4.27:</b> Histogram of MAX_DEFLECTION.....	47
<b>Figure 4.28:</b> Histogram of MAXVONMISESSTRESS.....	47
<b>Figure 4.29:</b> Histogram of MAXVONMISESSTRAIN.....	48

<b>Figure 4.30:</b> CDF of input variable FORCE_1.....	49
<b>Figure 4.31:</b> CDF of input variable FORCE_2.....	49
<b>Figure 4.32:</b> CDF of input variable HEAT.....	50
<b>Figure 4.33:</b> CDF of input variable HOLE_RADIUS.....	50
<b>Figure 4.34:</b> CDF of input variable ELASTIC_MODULUS.....	51
<b>Figure 4.35:</b> CDF of input variable POISSON_RATIO.....	51
<b>Figure 4.36:</b> CDF of output parameter MAXIMUM DEFLECTION.....	52
<b>Figure 4.37:</b> CDF of output parameter MAX_DEFLECTION.....	53
<b>Figure 4.38:</b> CDF of output parameter MAXVONMISESSTRESS.....	54
<b>Figure 4.39:</b> CDF of output parameter MAXVONMISESSTRAIN.....	55
<b>Figure 4.40:</b> Sensitivity plots for MAXIMUM DEFLECTION.....	56
<b>Figure 4.41:</b> Sensitivity plots for MAX_DEFLECTION.....	57
<b>Figure 4.42:</b> Sensitivity plots for MAXVONMISESSTRESS.....	58
<b>Figure 4.43:</b> Sensitivity plots for MAXVONMISESSTRAIN.....	59

**LIST OF TABLES**

<b>Table 1.1:</b> Properties of castellated beam.....	3
<b>Table 3.1:</b> Coordinate of keypoint.....	18
<b>Table 3.2:</b> Solution Controls Options.....	21
<b>Table 4.1:</b> Random Input Variable Specifications.....	29
<b>Table 4.2:</b> Specification for Monte Carlo Simulation Analysis.....	32
<b>Table 4.3:</b> Result Set – Statics of the Random Input Variables.....	33
<b>Table 4.4:</b> Results Set – Statics of the Random Output Parameters.....	34

## CHAPTER 1

### INTRODUCTION

#### 1.1 General

In the construction of modern buildings, it is necessary to accommodate pipes and ducts necessary services, such as air conditioning, water supply, sewerage, electricity, computer networks and telephone networks, in order to minimize storey height and achieve economic requirements. Usually, these pipes and tubes are placed under the beam soffit, for aesthetic reasons, covered ceiling, thus creating a corner (Chen, et al., 2008). Due to web openings presence leads to change of the stress distribution in the beam. The highest stress may or may not occur in the mid span. The changes of beam stress are depending on the shape, size and position of the opening (Akwasi Manu Assenso Antwi, 2012).

Besides that, the intention of castellated beam used in the steel structure is without any additional material also will increase the beam's web depth as well as the strength of the beam. This study aim to identify the simulation of castellated beam with different loading applied. Finite element method is the leading technique for analysing the behaviour of structures when subjected to a variety of loads. The finite element results are compared in term of load deflection curve, pure bending, shear, and lateral torsional buckling, showing a good agreement (Soltani, et al., 2012).

## 1.2 Problem statement

In recent years, researches had been done on different types of opening with respect to T-beams, precast beams, steel beams and deep beams but circular beams are the ones that having little information on it. Therefore, it is essential that having some experimental construction in the lab which is vital in presenting the performance of opening in the beams; opening on beam web is more than half of its depth that corresponded to the restricted detail of typical rectangular beams (Saksena & Patel, 2013).

There were several experimental and analytical studies that have been use in analysing the various structural aspects of castellated beam with the effect of web openings, with different modes of failures and the depth of web opening increasing with the effect of deflection (Wakchaure, et al., 2012). As a result, six different modes of failure have been reported by Kerdal and Nethercot in year 1984 (Kerdal & Nethercot, 1984). The castellated beam with six possible failure modes such as flexure mechanism, lateral torsional buckling, Vierendeel mechanism, rupture in welding joint, shear of web post, and compression buckling (Wakchaure, et al., 2012).

Besides that, finite element method is very widely used for solving engineering problem. It give difficulties to engineering if without software calculation cause the manual calculation took long time to calculate and sometime will make mistake.

## 1.3 Objectives of research

1. To obtain the maximum deflection, max von mises strain, max von mises stress by Probabilistic Design System (PDS).
2. To analyse the reaction forces, deformation, deflection, axial stress and heat transfer.

#### 1.4 Scope of study

1. To identify the effect of the various load acting on the castellated beam by using ANSYS software.
2. ANSYS 12.0 + CivilFEM 12.0 software is to obtain analysis result and compare with finite element method (FEM).

**Table 1.1:** Properties of castellated beam

Length of the beam	3 m
Steel Grade	S275
Poisson Ratio, $\nu$	0.3
Elastic Modulus, E	210E+09
Type of Steel	IPE 240
Type of opening	Hexagonal Opening
Number of opening	15
Radius	0.07 m
1 <sup>st</sup> Support	L/6
2 <sup>nd</sup> Support	L - L/6
1 <sup>st</sup> Point Load	L/3
Point Load	L - L/3

#### 1.5 Expected Outcome

This paper is to obtain the result of simulation castellated beam by using ANSYS software and compare the result with finite element method (FEM). Moreover, this study can proved that finite element formulation gives more accurate results in the global behaviour of the beam system.

## **CHAPTER 2**

### **LITERATURE REVIEW**

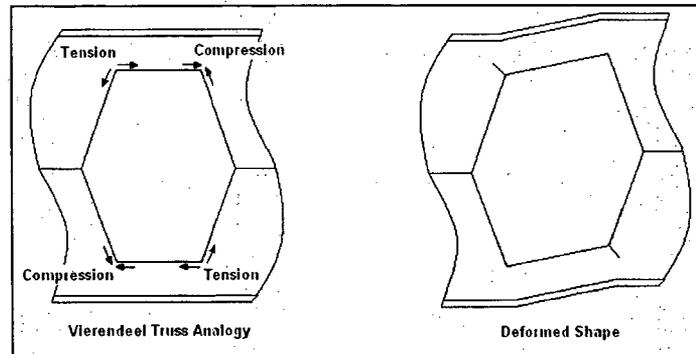
#### **2.1 Introduction**

In recent years there are continually growing rapidly by using steel structure in the construction industry across world. Castellated beam is one type of the steel structure's beam, by increasing the depth of beam also will increase the strength of beam without additional material (Billy Milligan, 2001). This chapter is to describe the model using finite element software package ANSYS +CivilFEM in analysing the behaviour of castellated steel beams having an I-shaped cross-section.

#### **2.2 Castellated Beam Analysis**

There are several method were used to analyse the castellated beam. Early methods in analysing the castellated beam was calculation of section properties excluded the web and flange but analyses the compression flange as a strut. In 1982, Kerdal and Nethercot discovered that the section properties at the middle of castellated beam or hole must be calculated. Besides that, they also found that the design for castellated beams can be derived from the equation for the behaviour of hot-rolled sections (Kerdal & Nethercot, 1982). Common method used to analyse the castellated beam is Vierendeel analysis (Billy Milligan, 2001).

This method is to consider the longitudinal stresses due to bending and the local bending from shear force which can increase the stresses in the I-section (Srimanit & Dast, 1977).



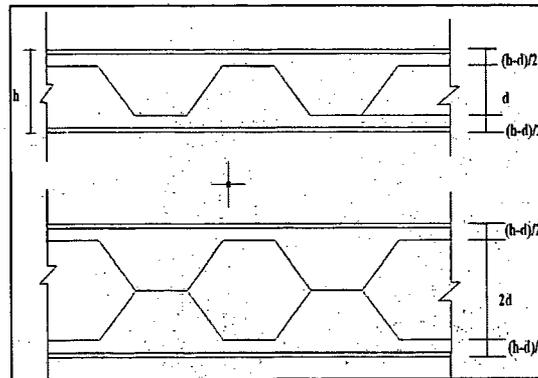
**Figure 2.1:** Vierendeel Truss Analogy

In 1984, Kerdal and Nethercot realized that the properties of lateral-torsional buckling were found in all the ultimate strength tests (Kerdal & Nethercot, 1984). They also found out that the castellated beams had six failure modes;

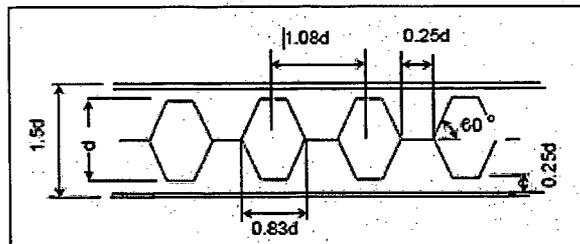
1. Formation of a Vierendeel mechanism
2. Lateral-torsional buckling of the web post
3. Rupture of the welded joint
4. Lateral-torsional buckling of the entire span
5. Web post buckling
6. Formation of a flexure mechanism

Any six failure modes were usually involved in the combination of test specimens. Providing the adequate lateral bracing was to evaluate the failure modes other than lateral-torsional buckling in the test specimens and the beams would reach their full-strength (Kerdal & Nethercot, 1984). Due to lack of lateral support to the compression flange, non-composite castellated beams are more susceptible to lateral-torsional buckling than composite beams.

Furthermore, the castellated beams have the same behaviour of lateral-torsional buckling as a plain webbed beams and the lateral-torsional buckling was influence by the holes (Wakchaure, et al., 2012).



**Figure 2.2:** Mathematical Formulation of Castellated Beam



**Figure 2.3:** Standard Castellated Section

### 2.3 Finite Element Method (FEM)

There are two ways to include FEM formulation, which is based on the direct variational method, such as the Rayleigh - Ritz method and other methods relevant as Galerkin weighted residual method. Based on the variation method, the basic equations are derived from the stationary conditions of the functional for the boundary-value problems. The advantage of this formulation is not necessary for the process of deriving functional, so

it is easy to analyse based on the finite element method (FEM) for the preparation of the weighted residuals.

In order to define the building design response with simulation loading condition, Finite Element Analysis is a way for different building condition. The meaning of element is the design of model using discrete building blocks which to analyse how the elements responds to a certain load with each exact equations. During the 1950s and 1960s, academic and industrial researchers were created the finite element method of structural analysis.

$$[K]\{q\} = \{F\}$$

Stiffness matrix      Nodal displacement vector      Nodal vector force

Figure 2.4: Linear algebraic equations

The purpose of this study is to manipulate the finite element software package ANSYS 12.0 programs to simulate the behaviour of castellated beam for load deflection plots and stress-strain plots. During 1996, a single web-post in non-composite castellated beams on the buckling of the web-post has been done by Zaarour and Redwood using FEM (Zaarour & Redwood, 1996).

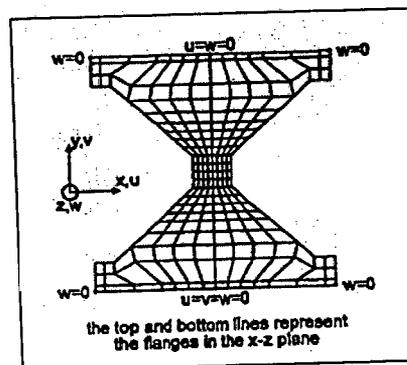
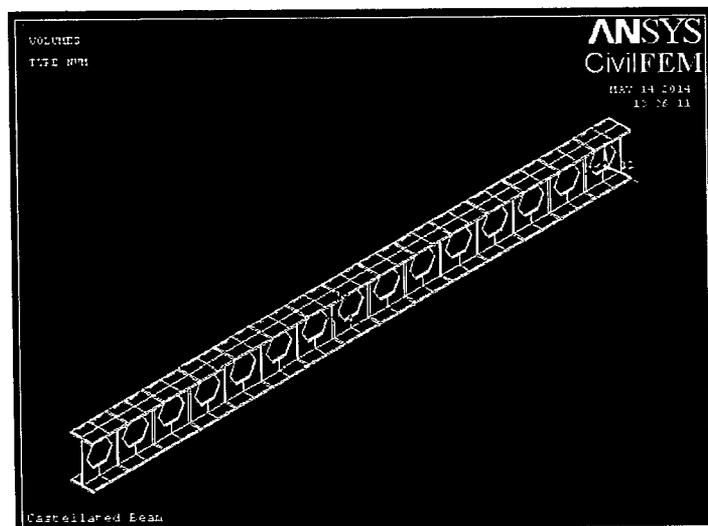


Figure 2.5: Non-composite finite element model

## 2.4 ANSYS

ANSYS software is the most advanced package for single- and multi-physics simulations, providing enhanced tools and features that enable engineers to complete their work in an efficient manner. ANSYS includes significant capabilities, expanding functionality, and integration with almost all CAD drawing software, such as Pro/Engineer, AutoCAD, and Solid Edge. In addition, ANSYS has the best-in-class solver technologies, an integrated coupled physics for complex simulations, integrated meshing technologies customizable for physics, and computational fluid dynamics (CFD). ANSYS can solve problems in structural, thermal, fluid, acoustics, and multi-physics.

In years 2012, Wakchaure and Sagade have done a study which is related to Castellated Beam using finite element software package ANSYS 14.0 on a three-dimensional (3D) finite element model (Wakchaure & Sagade, 2012). Besides that, the previous study of ANSYS has provided information about developing finite element models by Zirakian and Showkati such as in the form of failure loads, failure modes, load-lateral deflection curves and load-strain curves (Zirakian & Showkati, 2007).



**Figure 2.6:** Castellated Beam model

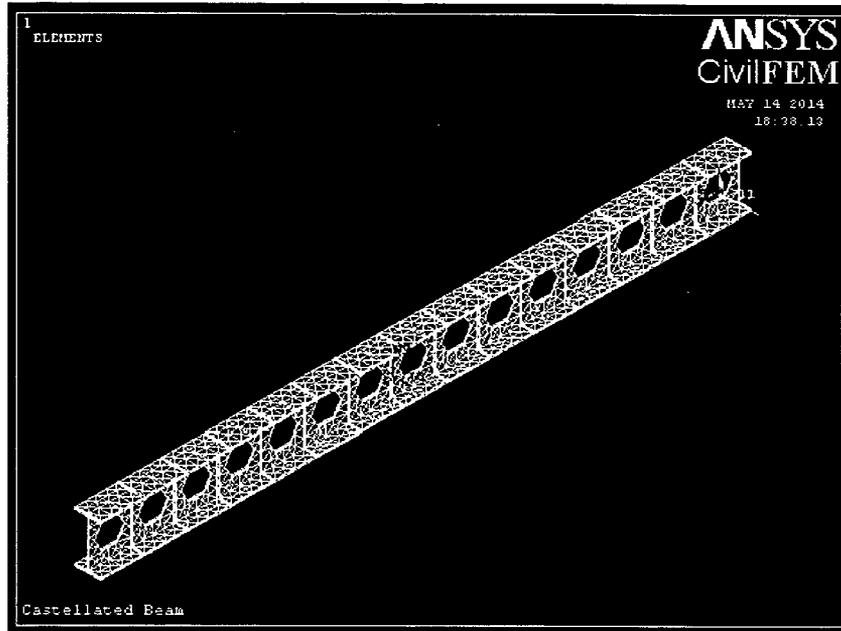


Figure 2.7: Typical Finite Element Mesh for a Castellated Beam

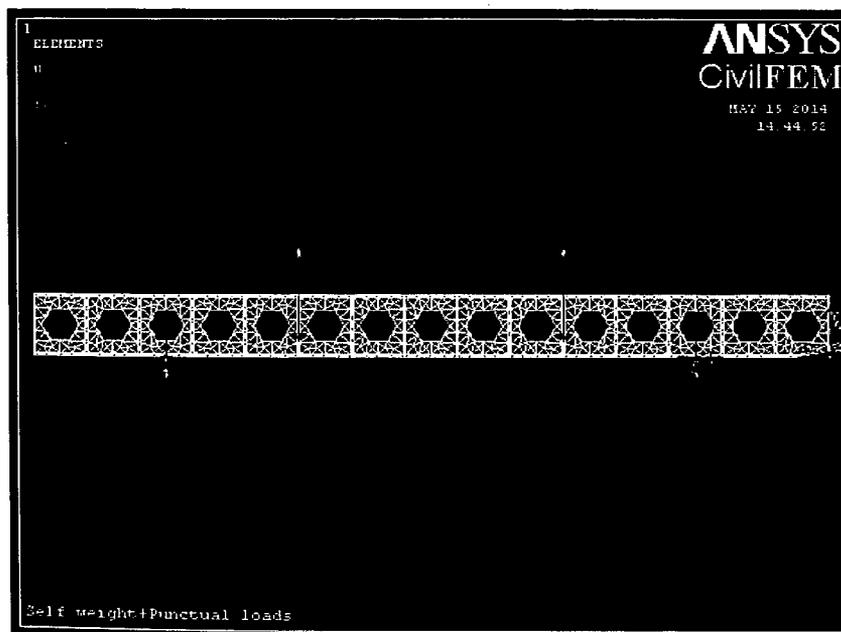


Figure 2.8: Applying load on castellated beam

Based on this study, finite element software package ANSYS 12.0 can be categorized into three groups as shown in below:

**1. Preprocessing**

- Specify title
- Set code
- Set units
- Define material
- Define element type
- Define section
- Define Beam properties
- Define Nodes and Elements
- Save the database

**2. Solution**

- Apply displacement constraints
- Apply force load
- Solve

**3. Post-processing**

- Enter the postprocessor and read results
- Plot bending moment
- Plot the bending stress in Y top
- List bending extreme stresses
- Plot bending stress distribution inside the cross-section

## **CHAPTER 3**

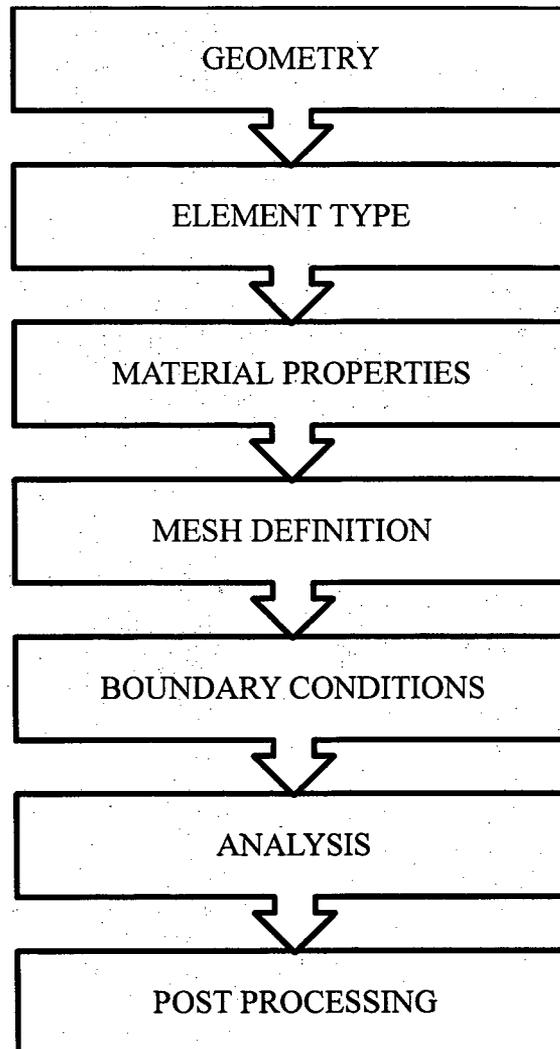
### **METHODOLOGY**

#### **3.1 Introduction**

This chapter will discuss the method on how to analyse the castellated beam and its behaviour problem. ANSYS is use to analyse and determine the strength of castellated beam under different loading applied. In ANSYS 12.0, there are three categorize group which are preprocessing, solution and post processing. Preprocessing stage consists of the units system, active code, materials, element types, section, and model geometry definition. For the solution stage, it consists of the analysis type and its options, apply loads and initiate the finite element solution whereas at post processing stage consists of reviewing the analysis results through graphic displays and tabular listings.

### 3.2 Method use to Analyse Structure

The modelling procedure is the following:



**Figure 3.1:** Flow chart of the modelling process

### 3.3 Design process

#### 3.3.1 ANSYS Preprocessor: Defining the problem

Model generation is conducted in this processor, which involves material definition, creation of a solid model, and finally, meshing. Important tasks within this processor are:

- Specify the title
- Set code and units
- Define material
- Define element type
- Define section
- Define beam & shell properties
- Define nodes and elements
- Mesh the model

#### 3.3.2 ANSYS Solution Processor: Assigning loads and solving

This processor is used for obtaining the solution for the finite element model that is generated within the *Preprocessor*. Important tasks within this processor are:

- Apply displacement constraints
- Apply force load
- Define analysis type and analysis option
- Specify boundary condition
- Obtain solution

### 3.3.3 ANSYS Post-processing: Viewing the results

In this processor, the results at a specific time (if analysis type is transient) over the entire or a portion of the model are reviewed. This includes the plotting of contours, vector displays, deformed shapes, and listings of the results in tabular format. Important tasks within this processor are:

- Enter the postprocessor and read results
- Plot the deformed shape
- Plot bending stress in Z top fibre
- List bending stress in Z top fibre

## 3.4 ANSYS Preprocessor Process

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

### 3.4.1 Defining an Analysis Title

#### Utility Menu > File > Change Title

Which brings up the dialog box shown in Figure 3.2. After entering desired title in *text box*, clicking on **OK** completes the specification of the title. The */TITLE* command, defines a title for the analysis. ANSYS includes the title on all graphics displays and on the solution output.