CHAPTER 1

INTRODUCTION

1.1 GENERAL

The finite element analysis is a numerical technique. It was an extension of matrix method of structural analysis. In finite element method, actual component is replaced by a simplified model, identified by a finite number of element connected at common point called nodes, with an assumed behavior of each element to the set of applied load, and evaluating the unknown variable such as displacement.

Since sixty years ago, the offshore structure is the bigger structure in oil and gas industry and it growing quickly over time (M.H. Patel 1989). Fix steel structure (D.J. Wisch 1998) was studied that the design of structure will be considered by load and the increasing of water depth. A simple model whole shape frame jacket will be proposed and modelled by the software ANSYS in three directions, then finite element calculation were carried out through the software to obtain the deflection, stress and strain and buckling of steel structure jacket. All these analysis were based on finite element method. Probabilistic analysis will be carried out by changing the input parameters and properties of steel to find out which affect the frame effectively.

The analysis of steel frame jackets can be done by hand calculation easily if the structure of frame is simple. However, if the structure of steel frame is complicated, the
analysis will involve many long and mathematical calculation may take long time to complete all the calculation. Nevertheless, with the increasing of computer technology all around the word, people nowadays are able to solve the analysis of a structure by using software in computer such as STAAD Pro, SACS, SAP 2000, and etc.

### 1.2 PROBLEM STATEMENT

Jackets were among the very first structures to be used in the offshore industry, as stable platforms mainly for oil and gas production facilities. Frame is one of the important elements of this frame structure. Unfit or improper design can lead to the failure of a structure. There are several methods can be used to analyzes the behavior of steel jacket frame. However, among all these methods, finite element method will be a very effective method to obtain the strength and behavior of steel structure.

Most of the times, engineers need to find out an accurate data of deflection and stress for the structure before he start design. A lot of time had been wasted for that works. By applying the probabilistic analysis, a range of result can be obtained by input the loading and the wind load. So, engineers works can be reduced and save more time for others.

### 1.3 OBJECTIVE

This thesis have a few objectives that related to the design of steel structure. There are many thing that should be considered to make this project more reasonable with the objective. The objective that use for this project must be achieve. The ultimate purpose of this study is to study the strength and behavior of steel jacket frame by using a finite element analysis software ANSYS.

i. To determine the force and moment, stress & strain and deflection on steel jacket structure using ANSYS software.

ii. To verify the steel jacket frame is stable against the variable load

iii. To determine probabilistic analysis of steel jacket frame
1.4 SCOPE OF STUDY

This research will mainly focus on the steel frame of jacket platform. In order to achieve the objective of this research, there are a few scope of work that important to be revised and followed, such as:

i. Result of analysis on steel structure (stress, strain, deflection)
ii. Develop a model on ANSYS software for analyzing steel jacket structure
iii. Conduct probabilistic response analysis on steel jacket structure

First step is install the ANSYS software to the computer and do all the tutorials that given on this software. The basic tutorial will increase the skills on modelling and analyzing all structure in ANSYS. Then, the properties of structure steel must be define which include their dimensions and the section on jacket structure. The type of structure can be determine by referring plans or assumptions. Preprocessing is the step to define lines and key points. It will be continue to next step. All environmental load and supports will assign to the modelling structure. After that, determine the solution of the structure which is deflection and stress. Finally, carry out probabilistic analysis and observe the result of that occur on structure.

1.5 EXPECTED OUTCOME

This research claims to find out the behaviors and passing code checking of 3D portal frame. The behaviors are deformation, deflection, tension checking, compression checking and lateral torsional buckling checking according to Eurocode 3.

1.6 SIGNIFICANCE OF STUDY

This research will be a significance endeavor in providing the information about the steps and results of analyzing steel jacket frame by using ANSYS. Maximum deformation, deflection axial stress and strain are able to be determined. It is really good to practice to analyze and make sure it is safe before constructing it. This definitely will enhance the reputation of Malaysia with advance engineering skills.