CHAPTER 1

INTRODUCTION

1.1 GENERAL

Nowadays we can look many of the building of the world used the high technology especially in civil engineering. From day to day many products were produced with using new technology. For the construction we can see the new technologies in the term of the installation of the member, design and how the project conducted such as IBS system and software that used to analysis. Steel frame is one of the products. Steel frame is the building technique with a skeleton frame of the steel columns and horizontal steel beam. The functions of this steel frame in the construction are to support the main element such as floors, roof and walls that attached to the frame. There are many shape can be used such as square, tabular and I section. As the designer they have to select the steel sections from a discrete that contains certain designations of steel profiles (john wiley and sons 1984). Generally the steel structures are design to be pinned or fully rigid connections but in the reality the connections in steel frames are mostly semi rigid. The research for the type of these connections as well as on the numerical techniques of completing an analysis of flexible connected steel frames was developed (A.J numer,1988).

Today many cases of the steel frames failure were occurs in the buildings construction and it is concern by the engineer and designer. The steel frame structures are required to be design in the term of their strength and rigidity to satisfy the loads and serviceability limitation. The steel frames as we know mostly are ready-made. So, the discrete optimization technique should be proposed (S.A May, R.J Bailing, 1992). As a
result engineer must initiate their design of these frames by proposed the girders, beam and columns to satisfy the total sideways. To ensure the steel frame is safe to used approximate optimization techniques are useful for obtaining well balanced which must be checked for strength using the standards (D.E Grierson ,1994).

ANSYS CivilFEM is the software that has most advanced comprehensive and reputable finite element analysis. This software is the design software package available for the structural engineering project. This CivilFEM can support all types of advanced analysis supported by ANSYS. Finite element analysis is a numerical method of deconstructing a complex system in every part of member that called as element. The analysis that produced was linearly elastic because contact was embodied artificially by attaching and release nodes at each load step. Then, the correlation between 2-Dimensional and 3-Dimensional finite element was established with 2D models (A.R kukreti, T.M Murray, A.Abolmaali ,1987). The ANSYS software implements equations that govern the behavior of the elements and solves them all and create a comprehensive explanation of the system. The result from the analysis was useful in the range for which such validations were performed and finite elements models was developed for stiffened steel tee hanger connections (T.M Murray , A.R Kukreti, M.Ghassemieh 1989).

1.2 PROBLEM STATEMENT

The structure failure can be classified in the many cases. The structure will collapse and fail due to their design of the structural components. Usually the construction project is more focus on the design of the building compared to the stability and strength of the structures. Some of them just think logically and just ignore some of the analysis. Sometimes the design standards are unable to fulfill the requirement. Besides that, the structure is complicated and also takes a lot of time to analyze. As the person that responsible to the construction it is necessary to calculate, design and checking the steel frame to ensure the structure is safe to use.
As we know now many engineering software can be used to design the steel frame design. One of the software that can be used is ANSYS software. The designer just fill the data that the software wants such as the length, area and material to produced the results. Sometimes, the result that produce is inaccurate because they not familiar with the software and not have enough knowledge on the software. Inaccurate analysis data may affect the frame structure.

### 1.3 OBJECTIVES

As we know everything that we to conduct must have their objective. It is the goal or mission that we must to complete. It functions as the guidance to achieve the final objective. For this thesis there have a few objectives that related in term of structure and analysis. The objectives are:

1. To model the steel frame structure design for food court.
2. To determine the behavior of steel frame structure.
3. To check the steel framed pass all the code checking.
4. To analyze the steel frame structure in term of stress, strain and deflection.
5. To determine the main descriptors using Monte Carlo simulation methods.

### 1.4 SCOPE OF STUDY

This research are more focused on the design of steel frame structures by using standard Eurocode 3 that base on the steel design. The data that produced will used to generate the new equation by proving the parameters in the ANSYS software. We must study the types of the steel frame design and the characteristic of the structure to achieve the objectives of the research.