

FINITE ELEMENT ANALYSIS OF STEEL FRAME STRUCTURE FOR FOOD
COURT USING ANSYS

MOHD KHAIRUL AZAM BIN MD KHAIRUDIN

Report submitted in partial fulfillment of the requirements
for the award of degree of
B.Eng. (Hons.) Civil Engineering

Faculty of Civil Engineering & Earth Resources
UNIVERSITI MALAYSIA PAHANG

JUNE 2015

ABSTRACT

In this research, a 3D steel frame structure has been analyze using ANSYS software and manual calculation. This research is to determine the strain, stress, maximum deflection, deformation and also checking the structure according to Eurocode 3. As we know the uses of steel frame structure in the construction is widely used. It easy to conduct and construct and at the same time reduced the duration time. The type of material and the geometry for the structure was satisfied in cases it passed all the designing for tensile, buckling and compression. From the result of simulation we can know the behavior of the structure under the input parameter that applied. Then, from the probabilistic analysis, we collect the result of cumulative distribution function, the histogram plot for input and output parameter, sensitivity plot and simple history plot for all parameter.

ABSTRAK

Di dalam kajian ini, kerangka besi 3D dianalisis dengan menggunakan program ANYS dan juga pengiraan secara manual. Analisis ini adalah untuk menentukan tekanan, terikan, pesongan, kecacatan bentuk dan memeriksa struktur berdasarkan Eurocode 3. Seperti yang kita ketahui penggunaan struktur besi di dalam pembinaan seringkali digunakan. Besi kerangka mudah digunakan, dibina dan dalam masa yang sama dapat mengurangkan penggunaan masa pembinaan. Jenis bahan dan geometri untuk struktur telas di kenalpasti dan dibenarkan apabila bahan melepasi pengiraan tensil, lengkungan dan tekanan. Daripada keputusan yang simulasi yang diperolehi kita boleh mengetahui sifat pada struktur dengan meletakkan parameter dalaman. Selepas itu daripada analisa probabilistic kita boleh memperolehi keputusan untuk kumulatif fungsi distribusi, lakaran histogram untuk parameter dalaman dan luaran, lakaran sensitive dan lakaran penerangan untuk semua jenis parameter yang terlibat.

TABLE OF CONTENTS

SUPERVISOR’S DECLARATION	ii
STUDENT’S DECLARATION	iii
DECICATION	v
ACKNOWLEDGEMENTS	iv
ABSTRACT	vi
ABSTRAK	vii
TABLE OF CONTENTS	viii
LIST OF TABLES	xii
LIST OF FIGURES	xiii
LIST OF SYMBOLS	xvii
LIST OF ABBREVIATIONS	xviii

CHAPTER 1 INTRODUCTION

1.1	Introduction	1
1.2	Problem Statement	2
1.3	Objective	3
1.4	Scope of Study	
1.5	Expected Outcome	4
1.6	Significance of Study	4

CHAPTER 2 LITERATURE REVIEW

2.1	Introduction	5
2.2	Load on Structure	6
	2.2.1 Static Force	6
	2.2.2 Wind Loads	7
2.3	Frame	7
	2.3.1 Steel Frame	7
2.4	Finite Element Analysis	8
	2.4.1 Finite Element Analysis Function	9
	2.4.2 Advantage of Parametric Design	9

CHAPTER 3 METHODOLOGY

3.1	Introduction	10
3.2	Design Process	12
	3.2.1 Civil Preprocessor Phase	
	3.2.1.1 Title	12
	3.2.1.2 Code and Unit	12
	3.2.1.3 Material	14
	3.2.1.4 Type of Elements	16
	3.2.1.5 Steel Cross Section	17
	3.2.1.6 Beam and Shell properties	18
	3.2.1.7 Create Nodes	19
	3.2.1.8 Elements	21
	3.2.2 Solution Phase	21
	3.2.2.1 Apply Displacement	21
	3.2.2.2 Apply Force/Moment	22

3.2.2.3 Solving the Model	23
---------------------------	----

CHAPTER 4 RESULTS AND ANALYSIS

4.1	Introduction	33
4.2	Probabilistic Design	34
4.2.1	Monte Carlo Simulation	35
4.2.2	Probabilistic model Information	36
4.2.3	Probabilistic Analysis Result	40
4.2.3.1	Statistic of the Probabilistic Results	40
4.2.3.2	Sample History Plots.	42
4.2.3.3	Histogram Plots	45
4.2.3.4	Cumulative Distribution Function Plots	49
4.2.3.5	Sensitivity Plots	53
4.2.3.6	Correlation Coefficient Matrix	56

CHAPTER 5 CONCLUSIONS AND RECOMMENDATION

5.1	Conclusion	60
5.2	Recommendation	61

REFERENCES	63
-------------------	----

APPENDICES

A1	Eurocode Checking of Steel Frame	64
A2	Stiffness Matrix of Steel Frame	73
A3	Element Forces of Steel Frame	79
B1	Log File	87

LIST OF TABLES

Table No.	Title	Page
3.1.1	Nodes for steel frame structures	19
4.2.2.1	Random input variable specifications	36
4.2.3.1.1	Statistic of the Random Input Variables	41
4.2.3.1.2	Statistic of the Random Output Variables	41
4.2.3.6.1	Linear Correlation Coefficients between Input Variables	56
4.2.3.6.2	Spearman Rank Order Correlation Coefficients between Input Variables	57
4.2.3.6.3	Linear Correlation Coefficients between Input and Output Variables	57
4.2.3.6.4	Spearman Rank Order Correlation Coefficients between Input and Output Variables	58
4.2.3.6.5	Linear Correlation Coefficients between Output Parameters	59
4.2.3.6.6	Spearman Rank Order Correlation Coefficients between Output Parameters	59

LIST OF FIGURES

Figure No.	Title	Page
2.2	typical types of loading condition standards that must be Follow when design process.	6
3.1	Summarize research methodologies for the study	11
3.2.1	Title for the steel frame model.	12
3.2.2	Activate CivilFEM	13
3.2.3	CivilFEM Setup	13
3.2.4	CivilFEM code setup.	14
3.2.5	CivilFEM unit setup.	14
3.2.6	Structural steel material	15
3.2.7	Steel material	15
3.2.8	Material specification browser	16
3.2.9	Beam element types for code checking	16
3.2.10	Steel cross section	17
3.2.11	Steel cross section	17
3.2.12	I-beam steel cross section	18
3.2.13	Steel cross section detail	18
3.2.14	Steel beam and shell properties	19
3.2.15	Window to enter the nodes coordinates	20
3.2.16	nodes in active coordinate	20

3.2.17	Element joints in active coordinate	21
3.2.18	Applied for all DOF	22
3.2.19	Force apply on FY direction	22
3.2.20	Force apply on FX direction	23
3.2.21	All force applied on model	23
3.2.22	solution is done	24
3.2.23	Plot Deformed Shape	24
3.2.24	Deformed Shape	25
3.2.25	Contour Nodal Solution Data	25
3.2.26	Contour Plot of Deflection	26
3.2.27	Read result by the load step	26
3.2.28	Force and moments graph result	27
3.2.29	stress and strain graph result	27
3.2.30	Tension checking	28
3.2.31	Tension graph steel result selection	28
3.2.32	Graph for tension steel result	29
3.2.33	Graph for compression steel result	29
3.2.34	Graph steel result for compression buckling	29
3.2.35	Graph result for bending moment	30
3.2.36	Graph result for shear	30
3.2.37	Graph result for bending and shear	30

3.2.38	Graph result for bending and axial	31
3.2.39	Graph result for bending axial and shear.	31
3.2.40	Graph result for bending buckling	32
3.2.41	Graph result for bending and compression buckling	32
4.2.2.1	PDF & CDF of input random variable applied load	37
4.2.2.2	PDF & CDF of input random variable wind load	37
4.2.2.3	PDF & CDF of input random variable density	38
4.2.2.4	PDF & CDF of input random variable Poisson	39
4.2.2.5	PDF & CDF of input random variable temperature	39
4.2.3.2.1	Sampled Values for Output Parameter MAXIMUMDEFLECTION	42
4.2.3.2.2	Mean Values for Output Parameter MAXIMUMDEFLECTION	43
4.2.3.2.3	Standard Deviation History for Output Parameter MAXIMUMDEFLECTION	43
4.2.3.2.4	Minimum Value History for Output Parameter MAXIMUMDEFLECTION	44
4.2.3.2.5	Maximum Value History for Output Parameter MAXIMUMDEFLECTION	44
4.2.3.3.1	Histogram of Input Variable APPLIEDLOAD	45
4.2.3.3.2	Histogram of Input Variable WINDLOAD	46
4.2.3.3.3	Histogram of Input Variable DENSITY	46
4.2.3.3.4	Histogram of Input Variable POISSON	47
4.2.3.3.5	Histogram of Input Variable TEMPERATURE	47

4.2.3.3.6	Histogram of Input Variable ELASTIC	48
4.2.3.3.7	Histogram output Parameter MAXIMUMDEFLECTION	48
4.2.3.3.8	Histogram output Parameter MAX_DEFLECTION	49
4.2.3.4.1	CDF of Input Variable APPLIEDLOAD MAX_DEFLECTION	50
4.2.3.4.2	CDF of Input Variable WINDLOAD	50
4.2.3.4.3	CDF of Input Variable DENSITY	51
4.2.3.4.4	CDF of Input Variable POISSON	51
4.2.3.4.5	CDF of Input Variable TEMPERATURE	5
4.2.3.4.6	CDF of Output Parameter MAXIMUMDEFLECTION	52
4.2.3.4.7	CDF of Output Parameter MAX_DEFLECTION	53
4.2.3.5.1	Linear correlation Sensitivities for MAXIMUMDEFLECTION.	54
4.2.3.5.2	Rank-Order correlation Sensitivities for MAXIMUMDEFLECTION	54
4.2.3.5.3	Linear correlation Sensitivities for MAX_DEFLECTION	55
4.2.3.5.4	Rank-Order correlation Sensitivities for MAX_DEFLECTION	55

LIST OF SYMBOLS

tw	Web thickness
tf	Flange thickness
hw	Height of web
i	Height
b	Width
d	Depth
A	Area of section
I	Moment of inertia
W_{pl}	Plastic modulus
i	Radius of gyration
N	Axial load
V	Shear force
M	Moment
α	Imperfection factor
$\gamma M0$	Partial factor for resistance of cross-sections whatever the class is
$\gamma M1$	Partial factor for resistance of members to instability assessed by member checks
λ	Slenderness value
\emptyset	Value to determine the reduction factor
X	Reduction factor
L_{cr}	Buckling Length
K_{zy}	Interaction factor
[K]	Global-coordinate structure stiffness matrix

LIST OF ABBREVIATIONS

2D	Two Dimensional
3D	Three Dimensional
CIVIFEM	Civil Finite Element Method
LatBuck	Lateral Buckling
ChckAxis	Check Axis
BMSHPRO	Beam and Shell Properties
CS	Coordinate System
LS	Load Step
DOF	Degree of Freedom
PRES	Pressure
GAUS	Gaussian
DENS	Density
ELASTIC	Elastic modulus
POISON	Poison ratio
LOAD	Point load
WINDLOAD	Wind load
TEMP	Temperature
PDF	Probabilistic density function
CDF	Cumulative distribution function
MAXIMUMDEFLECTION /MAX_DEFLECTION	Maximum Deflection

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.

Explore how to conduct the ANSYS software by using the tutorial from the class and internet. Practicing the tutorials and understanding the function can help to solve a problem when the software runs the real model that is proposed. The results from the analysis can produce the data for the steel frame structures that are mentioned in the objectives.

1.5 EXPECTED OUTCOME

This research is more about the steel frame structure behavior. It aims to analyze the behaviors and pass code checking. The behavior that is of interest for this research is deformation, deflection, stress and strain tension checking, compression checking and lateral torsion buckling checking according to standard Eurocode 3.

1.6 SIGNIFICANCE OF STUDY

This research will be a significant endeavor in providing the information about the steps and results of analyzing steel frame structure by using ANSYS. Deflection, axial stress and strain are able to be determined. It is relevant to use and practice to analyze the steel frame structure to ensure it is safe before the construction phase continues.

Besides that, this research study can help in the term to save the time and also the cost by determining the relevant size of the section that follows the standard. It is important to study and to learn of the steel structure design today because a lot of the buildings use the steel frame structure. In Malaysia, as we can see, the steel frame structure has become famous. In a nutshell, this study is very significant to our country, building that totally uses the steel structure and the field of civil engineering.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

Numbers of study have been made for many years in designing the steel frame structure and still continue until today. There are lot types of steel frame including braced, unbraced and portal frame. As we know many software that used by researcher to present the simplest way such as ANSYS, Visual Basic and many more for determination of the cross section and size with follow the standard principle and limitations as stated in Eurocode 3.

Apart from the several methods introduced to fulfill the criteria such as Mechant-Rankine method and Wind Moment method. But the efficiencies and the effectiveness of these both method still not complete because the second order equations effect due to sway does not taking into account. The sway deflection gave the major effect in the construction of steel frames. It is not relevant to produce the final design of steel frames that exceeds the sway limitation because the structure does not effect by the sway. Therefore in the present of the Direct Design Method it can simplify the sway deflection and introduce the criterion.

ANSYS software is the supporting software that used by the researcher to reduced the difficulty when they designing the steel frame structures. The result that produced by this software can be compared with the manual calculation to get the true results. Besides that the establishment of this software can help in the term of modify the structure.

2.2 LOADS ON STRUCTURE

Load can be classified into a few groups. It can be affect the main structures especially in their behavior. Before we want to start the design process we must follow the criteria of the loading that must be taken by follow the standards (Brussels,1994). It is because if it is not consider the structure automatically will be fail. There are several types of loads that must be considered.

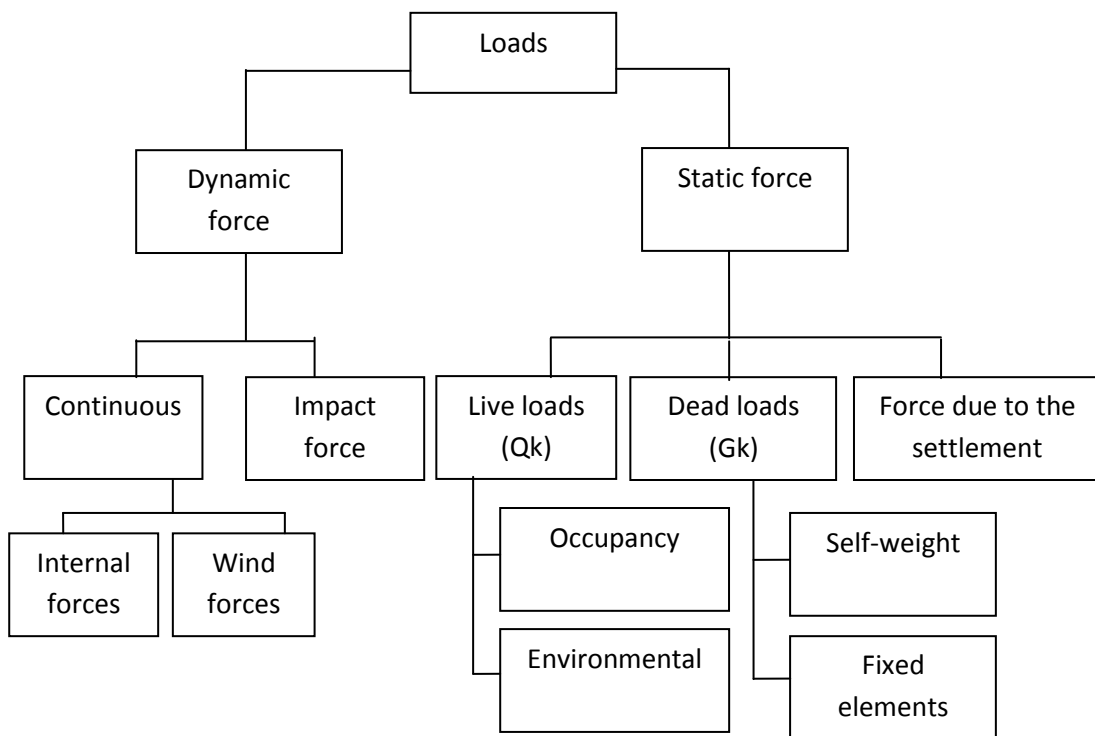


Figure 2.2 show the typical types of loading condition standards that must be follow when design process.

2.2.1 Static Force

There have a few types of loads that must be considered when the design process starts. The loads that must consist are live loads, dead loads and load due to the settlement. This several loads must be considered to ensure the structures follow the criteria in designing (National Research Council of Canada 1995).

The dead loads is the load that classified as non-movable load and it's consider as fixed or permanent structures such as column and beam. These types of structures are considered as dead loads because it has their own self-weight and the loading never change. For the live loads, it is the force that acting vertically or in the other words the forces that acting downward onto the structure. The value of the live loads not static it will be changes when the loads move. Occupancy and environmental can be described as live loads that including the furniture, stored material, human and the other structures that acts as movable structure.

2.2.2 Wind Loads

Wind loads is the loads that must be consider when the structure that construct more that 12 meter. It is important to calculate because it can cause the sway to the structure. When the wind move it will deflect when something such as high rise building blocking its path. It can produce the wind loads to the structure of the buildings. This type of phenomenon will cause the kinematic energy transform into potential energy. The loading for the wind load can be the major design parameter due to fact steel structure are strongly affect by lateral loads (Stathopoulos, 2007). So, for the buildings that want to construct for high rise building its must be design properly in order to resist this wind loads.

2.3 FRAME

Frame is the structure that combination of two permanent structures that are columns and beams. The structure was connected to each other by using bolt and nuts connection. Frame can be classified into two types that are braced frame and unbraced frame. Both of this frame are classified into one specific structure that is steel frame.

2.3.1 Steel Frame

The uses of this designing steel frame structures have reduce the use of the steel material in the frame construction. Many approaches have been made to study the

feasibility of steel frame structure since 1856 by Besemer's process. As we know many tall and high rise building used this types of steel frame structures in their buildings construction.

The uses of this designing steel frame structures have reduce the use of the steel material in the frame construction. Many approaches have been made to study the feasibility of steel frame structure since 1856 by Besemer's process such as the analysis consider parameter stiffness and strength for the steel frame based on Eurocode 3 (Design of steel structures, 1997). As we know many tall and high rise building used this types of steel frame structures in their buildings construction.

2.4 FINITE ELEMENTS ANALYSIS

Finite element analysis (FEA) is the subject that consists of software model of material and design that analyzed for the specified results. This FEA is used for many product design and existing product research. FEA is defined as a numerical method to find out approximate solution for variables which difficult to obtained analytically. The solution for calculation of the potential design changes such as buckling and deflection are usually complicated to solve. So, in the present of this FEA the problem will be solve (Banardo and Maicon, 2004).

The concept of this FEA solution is basically by solving a discrete model. It is solve by dividing the problem into small several elements. For each element is produced in simple geometry make it easier to analyze than the actual structure. All the elements that produced is applied with the known physical laws. The equation that formed by all the elements will be combined to form a global equation and it can be used to solve the field variables such as displacement and buckling. As we know this finite elements had started used since 1970. It is used in the aeronautics and nuclear industries and after a few years the finite element is carried out into modeling and solution (Reddy, 1993). This FEA becomes more famous today because with present of this analysis many engineering problems will simplify and solves easily.

2.4.1 Finite Element Analysis Function.

As we know FEA is the analysis that used a complex system of points called nodes which make a grid that called a mesh. This grid is programmed to contain the material and structural properties which shows how the structure reacts with certain loading conditions. Nodes are assigned at certain density throughout the material depending on the anticipated stress levels of particular area. For regions it will function as receive large amounts of stress that usually have a higher node density than those. Point of interest is the point may consist of the fracture point of previously tested materials and high stress areas. This mesh functions like the spider web from each node and extends a mesh element to each of the adjacent nodes. This web of vectors produced the material properties to the object and create element. (Peter Widas, 1997).

2.4.2 Advantage Of Parametric Design

Parametric design is one of the analysis that can produced the accurate information about the influence of all the parameters on the design objectives such as system performance with respect to loading, deflection of structures and respect to the stress. With this analysis people or team that wants to propose their product can get the decision more easily. The design teams can get the info more effectively to any modification due the external constraints such as manufacturing. (Thieffry, 2008).

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

All of the analysis that wants to conduct must have their own method. For this chapter it will discuss about the method that used to analyze the steel frame structure model. By using ANSYS software the structure will analyze and the result that produce will collect to get the related results with the Finite element method (FEM) calculation. The analysis processing for the ANSYS was divided into three main stages that are civil preprocessor, solution and postprocessor. In the first stage civil preprocessor the model materials and the cross section were selected. Secondly, in preprocessor part the model is built using the nodes and the elements to show the figure of the models and to confirm the elements is joints with the true nodes. After finish the modeling process the solution stage was continued. In this stage the types of support and the loading vertical and horizontal load was specified. Lastly the post processing will continue to checking the models to know the models that design is not failed in each member.

For this study the food court steel frame structure is modeling and analyzed. The suitable material and the cross section were defined. The loading that consider are vertical load (Q_k and G_k) and horizontal load (wind load) is applied to the model. This

types of loading is important to consider to ensure the model can support the without failure.

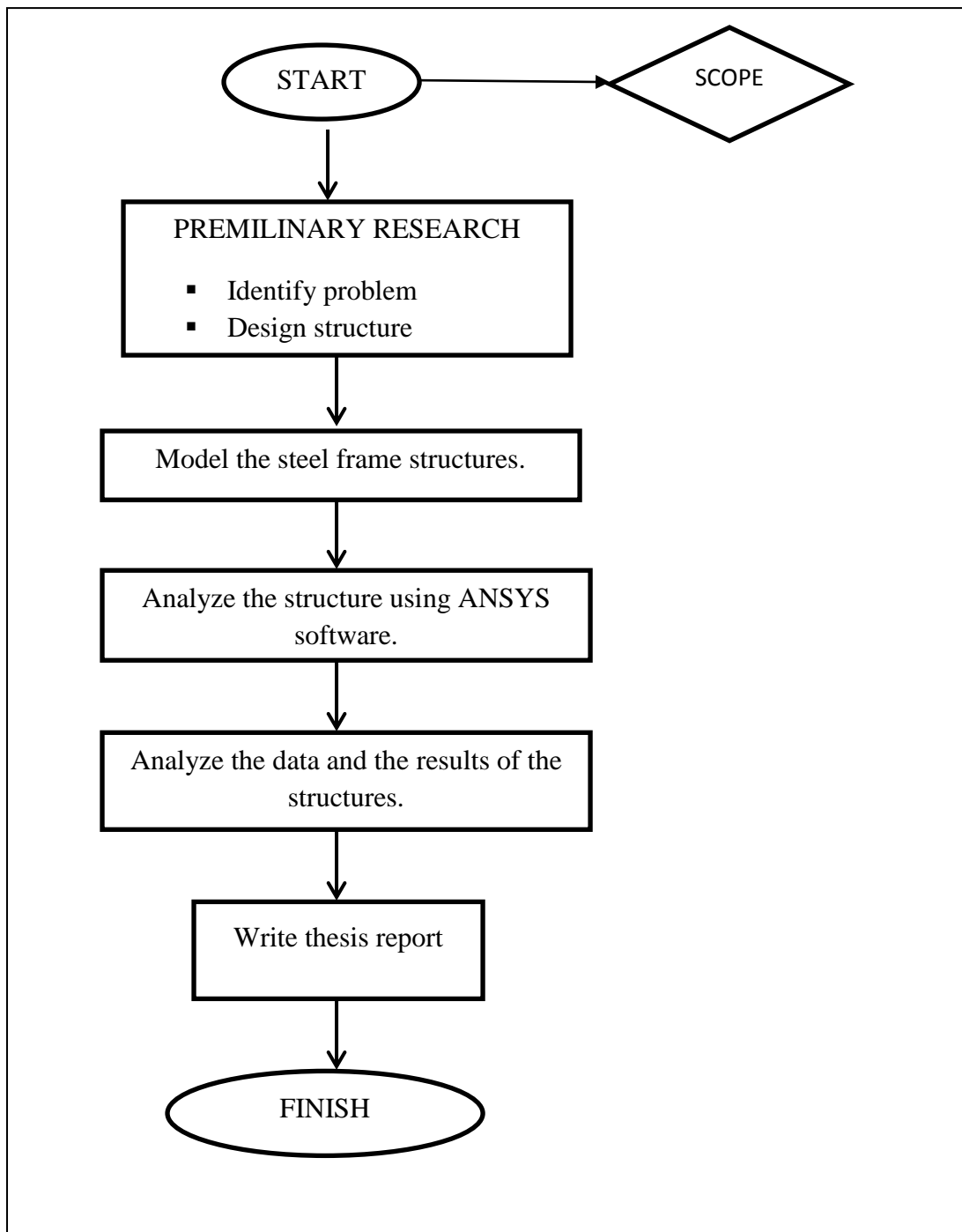


Figure 3.1 Summarize research methodologies for the study