

FINITE ELEMENT ANALYSIS OF 3D REINFORCED CONCRETE FRAME USING  
ANSYS

MUHAMMAD ADIB BIN EMBONG

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, 3D reinforced concrete frame structure is analyzed by using Probabilistic Method with ANSYS and checked by Finite Element Method and Eurocode 3. This analysis is to determine the behavior of the reinforced concrete frame and to study about the probabilistic response analysis and input sensitivities of the reinforced concrete frame. Nowadays, reinforced concrete frame is widely used in many high rise building construction include shopping mall, condominium, apartment and tower. The material used in this analysis is  $30\text{N/mm}^2$  of concrete and  $500\text{N/mm}^2$  of reinforced steel bar. The material and geometry of the reinforced concrete frame was satisfied for this research since it passed all the design criteria in designing for axial bending, shear & torsion and cracking. The numerical results show very good agreement with manual calculation results. From the results of simulation, we get to know the real behavior of the structure under the applied loads. In probabilistic analysis, we get the results of cumulative distribution function, histogram plot, sensitivity plot and simple history plot by 10000 times of simulation for any input and output.

## ABSTRACT

Dalam kajian ini kerangka konkrit bertetulang 3D telah dianalisis menggunakan teknik kebarangkalian dengan ANSYS dan disemak menggunakan Eurocode 3. Kajian ini bertujuan untuk menentukan tingkah laku kerangka konkrit bertetulang dan mengkaji kebarangkalian berdasarkan sensitiviti input kerangka konkrit bertetulang. Kini, kerangka konkrit bertetulang banyak digunakan dalam pembinaan bangunan yang tinggi seperti pusat membeli belah, kondominium, apartmen, dan menara. Antara bahan yang digunakan dalam kerangka konkrit bertetulang ialah konkrit gred 30N/mm<sup>2</sup> dan tetulang gred 500N/mm<sup>2</sup>. Bahan-bahan serta reka bentuk kerangka tulang bertetulang memenuhi keperluan untuk kajian kerana melepasi kesemua ujian reka bentuk untuk lenturan, ricih dan kilasan serta keretakan. Berdasarkan keputusan berangka dan pengiraan manual, kedua-duanya menunjukkan hasil yang baik dan memuaskan. Dari hasil simulasi, kita dapat mengenali tingkah laku sebenar struktur apabila beban dikenakan. Dalam analisis kebarangkalian, kita akan mendapat hasil fungsi kumulatif pengedaran, histogram plot, plot sensitiviti dan sejarah plot mudah oleh 10000 kali simulasi dari segi input dan output.

**TABLE OF CONTENTS**

<b>SUPERVISOR'S DECLARATION</b>	<b>ii</b>
<b>STUDENT'S DECLARATION</b>	<b>iii</b>
<b>DECICATION</b>	<b>iv</b>
<b>ACKNOWLEDGEMENTS</b>	<b>v</b>
<b>ABSTRACT</b>	<b>vi</b>
<b>ABSTRAK</b>	<b>vii</b>
<b>TABLE OF CONTENTS</b>	<b>viii</b>
<b>LIST OF TABLES</b>	<b>xii</b>
<b>LIST OF FIGURES</b>	<b>xiii</b>
<b>LIST OF SYMBOLS</b>	<b>xvi</b>
<b>LIST OF ABBREVIATIONS</b>	<b>xvii</b>

**CHAPTER 1      INTRODUCTION**

1.1	Background of Study	1
1.2	Problem Statement	2
1.3	Objective	3
1.4	Scope of Study	3

## **CHAPTER 2          LITERATURE REVIEW**

2.1	Introduction	4
2.2	Reinforced Concrete Frame Structure	4
2.3	Advantages of Concrete Structure	5
2.4	Finite Element Method (FEM)	6
2.5	Load on Frame Structure	7
2.5.1	Static Force	7
2.5.2	Wind Load	8
2.5.3	Earthquake	9

## **CHAPTER 3          METHODOLOGY**

3.1	Introduction	10
3.2	Design Process	11
3.2.1	Preprocessing Phase	11
3.2.1.1	Enter the title	12
3.2.1.2	Set code and unit	12
3.2.1.3	Define Material	14
3.2.1.4	Define the Type of Material	16
3.2.1.5	Define Concrete Cross Section	16
3.2.1.6	Beam & Shell Properties	18
3.2.1.7	Create Keypoint	19
3.2.1.8	Form Line	21
3.2.1.9	Meshing	21
3.2.2	Solution Phase	22
3.2.2.1	Apply Displacement	22

3.2.2.2 Apply Force/Moment	22
3.2.2.3 Solving the Model	24

## **CHAPTER 4            RESULTS AND DATA ANALYSIS**

4.1	Introduction	30
4.2	Deterministic Model Information	31
4.3	Probabilistic Model Information	32
4.4	Probabilistic Analysis Summary	36
4.5	Regression Analysis Summary	36
4.6	Probabilistic Analysis Result	36
4.6.1	Statistics of the Probabilistic Results	36
4.6.2	Sample History Plots	38
4.6.3	Histogram Plots	45
4.6.3.1	Histogram Plots of Input Parameters	48
4.6.4	Cumulative Distribution Function Plots	52
4.6.5	Sensitivity Plots	56
4.6.6	Linear Correlation Coefficients	58

## **CHAPTER 5            CONCLUSIONS AND RECOMMENDATIONS**

<b>5.1</b>	<b>Conclusion</b>	<b>61</b>
<b>5.2</b>	<b>Recommendation</b>	<b>62</b>

<b>REFERENCES</b>	<b>63</b>
-------------------	-----------

**APPENDICES**

A1	Eurocode Checking of Reinforced Concrete Frame	64
A2	Stiffness Matrix of Reinforced Concrete Frame	69
A3	Displacements of Reinforced Concrete Frame	72
A4	Element Forces of Reinforced Concrete Frame	73
B1	Log File	78

## LIST OF TABLES

<b>Table No.</b>	<b>Title</b>	<b>Page</b>
3.1	Keypoint of Frame Structure	14
4.1	Details of the Finite Element Model	32
4.2	Material Properties	32
4.3	Random Input Variable Specifications	33
4.4	Specifications for Monte Carlo Simulation Analysis	36
4.5	Statistics of the Random Input Variables	37
4.6	Statistics of the Random Output Parameters	37
4.7	Linear Correlation Coefficients Between Input Variables	59
4.8	Linear Correlation Coefficients between Input and Output Variables	59
4.9	Linear Correlation Coefficients between Output Variables	59
4.10	Spearman Rank Order Correlation Coefficients between Input Variable	59
4.11	Spearman Rank Order Correlation Coefficients between Input and Output Variables	60
4.12	Spearman Rank Order Correlation Coefficients between Output Variables	60



## LIST OF FIGURES

<b>Figure No.</b>	<b>Title</b>	<b>Page</b>
Figure 2.1	Wind load effect to the frame	8
Figure 3.1	Simplified research methodology for this study	11
Figure 3.2	Title for the model	12
Figure 3.3	Activate CivilFEM	12
Figure 3.4	CivilFEM Setup	13
Figure 3.5	CivilFEM setup code	13
Figure 3.6	CivilFEM setup unit	14
Figure 3.7	Concrete material	14
Figure 3.8	Reinforced steel material	15
Figure 3.9	Material browser after choosing material	15
Figure 3.10	Window beam element type for code checking	16
Figure 3.11	Concrete cross-section	17
Figure 3.12	Reinforcement properties	17
Figure 3.13	Reinforcement concrete cross section	18
Figure 3.14	Window beam & shell properties	18
Figure 3.15	Window to enter the keypoint	20
Figure 3.16	keypoint in active coordinate	20
Figure 3.17	Line in active coordinate	21
Figure 3.18	After mesh process	21
Figure 3.19	Nodes for ALL DOF	22
Figure 3.20	Force apply on FY direction	23
Figure 3.21	Force apply on FX direction	23
Figure 3.22	All force applied on model	24
Figure 3.23	Solution is done	24
Figure 3.24	Read result by load step	25
Figure 3.25	Code checking by Eurocode 2	25
Figure 3.26	Shear and torsion checking	26
Figure 3.27	Axial force X graph	26
Figure 3.28	Shear force Y graph	27

Figure 3.29	Bending moment Z graph	27
Figure 3.30	Axial + Biaxial bending checking	28
Figure 3.31	Shear & Torsion checking	28
Figure 3.32	Cracking checking	29
Figure 4.1	Model Geometry and Finite Element Mesh	31
Figure 4.2	PDF & CDF of Input Random Variable APPLIEDLOAD	33
Figure 4.3	PDF & CDF of Input Random Variable DENS	34
Figure 4.5	PDF & CDF of Input Random Variable POISON	34
Figure 4.6	PDF & CDF of Input Random Variable TEMP	35
Figure 4.7	PDF & CDF of Input Random Variable WINDLOAD	35
Figure 4.8	Sampled Values for Output Parameter MAXIMUMDEFLECTION	39
Figure 4.9	Mean Value History for Output Parameter MAXIMUMDEFLECTION	39
Figure 4.10	Standard Deviation History for Output Parameter MAXIMUMDEFLECTION	40
Figure 4.11	Minimum Value for Output Parameter MAXIMUMDEFLECTION	40
Figure 4.12	Maximum Value for Output Parameter of MAXIMUMDEFLECTION	41
Figure 4.13	Sampled Values for Output Parameter MAX_DEFLECTION	41
Figure 4.14	Mean Value History for Output Parameter MAX_DEFLECTION	42
Figure 4.15	Standard Deviation History for Output Parameter MAX_DEFLECTION	42
Figure 4.16	Minimum Value for Output Parameter MAX_DEFLECTION	43
Figure 4.17	Maximum Value for Output Parameter of MAX_DEFLECTION	43

Figure 4.18	Sampled Values for Output Parameter MAXIMUMDEFLECTION	44
Figure 4.19	Sampled Values for Output Parameter MAX_DEFLECTION	45
Figure 4.20	Histogram of Output Parameter MAXIMUMDEFLECTION	46
Figure 4.21	Histogram of Output Parameter MAX_DEFLECTION	46
Figure 4.22	Histogram of Input Variable APPLIEDLOAD	48
Figure 4.23	Histogram of Input Variable DENS	49
Figure 4.24	Histogram of Input Variable ELASTIC	49
Figure 4.25	Histogram of Input Variable POISSON	50
Figure 4.26	Histogram of Input Variable TEMP	51
Figure 4.27	Histogram of Input Variable WINDLOAD	51
Figure 4.28	CDF of Input Variable APPLIEDLOAD	52
Figure 4.29	CDF of Input Variable DENS	53
Figure 4.30	CDF of Input Variable POISSON	54
Figure 4.31	CDF of Input Variable TEMP	54
Figure 4.32	CDF of Input Variable WINDLOAD	55
Figure 4.33	CDF of Output Parameter MAXIMUMDEFLECTION	55
Figure 4.34	CDF of Output Parameter MAX_DEFLECTION	56
Figure 4.35	Sensitivity Plot for MAXIMUMDEFLECTION	57
Figure 4.36	Sensitivity Plot for MAX_DEFLECTION	58

## LIST OF SYMBOLS

$t_w$	Web thickness
$t_f$	Flange thickness
$h_w$	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
$\alpha$	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
$\lambda$	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 Background of Study

Reinforced concrete structure is a common element used in construction. Concrete can be called as artificial stone which is produced by mixing sand, aggregates, cement and water. Fresh concrete can be customized into any shape. This is one of the advantages of concrete compare to the other material. The use of concrete became popular in 19<sup>th</sup> century after Portland cement is introduced. However, concrete is weak against tension as it has limited tension resistance. To solve the problem, steel bars are placed in the concrete to form composite material. This composite material is called reinforced concrete. Reinforced concrete has been used in many engineering application such as buildings, bridges, dams and others.

Reinforced concrete is used by most of the country in the world because of the wide availability of steel bars and concrete ingredients. Compare to steel, concrete production requires cheap manufacturing factory. However concrete construction requires expertise, workmanship, and certain level of technology during the construction work. For reinforced concrete frame, it consists of horizontal and vertical element which is called beam and column. These two elements are connected by fixed joints. To increase the strength of the frame, beam and column are usually cast together on the same time during construction work. There are two types of frame which is braced frame and unbraced frame. Braced frame usually can support higher value if lateral force compare to the unbraced frame. This thing happens because braced frame provide bracing system that prevent the structure from sway when horizontal load is applied. On

the other hand, bracing system is not provided in unbraced frame which not allow them to support high horizontal load.

Frame analysis is performed to know the behavior of reinforced concrete structure when horizontal and vertical load is applied. The calculation just takes a short time for a simple frame analysis. But for the complex frame structure, it takes long time to be finished. Because of that, many structural engineers invented computer software to help the engineers for the calculation. There are many software created for structural engineer such as Esteem, STAAD.Pro, Orion, ANSYS, and others. By using computer software to analyze civil structure, we can reduce error on the calculation and save much time to finish the design.

In this project, we use unbraced frame as our structure. We will study the behavior of reinforced concrete frame under different load condition. Four storey water tank structures are used to analyze the behavior of the frame. Finite element method (FEM) is used for this study to get more accurate result. ANSYS is one of the computer software that used finite element method for the analysis.

## **1.2 Problem Statement**

Reinforce concrete frame is widely used nowadays. Most of the high rise building construction use reinforce concrete frame because reinforced concrete frame structure has many advantages compare to steel frame structure. It is very important to analyze the frame structure before it is constructed to avoid structure failure in the future.

The most problem faced in reinforced concrete design is cracking and deflection of the structure. Because of that, proper design needs to be done before the construction work started. By using ANSYS, we can design and analyze the structure to know the critical part of the frame structure.

### **1.3 Objectives**

The main purpose of this project is to study the strength and behavior of water tank reinforced concrete frame by using a finite element analysis software ANSYS.

- 1) To proposed a frame design for the water tank structure.
- 2) To determine cracking and deflection on reinforced concrete structure.
- 3) To determine shear and bending moment diagram of the concrete structure.

### **1.4 Scope of Study**

In this study, 3D model of reinforced concrete frame structure is analyzed using Eurocode No.2 in ANSYS. Model of the structure is designed in ANSYS and the structure is analyzed using finite element method. Before we can do the model in ANSYS, we need to learn how to use the software very well. Tutorial of the software is provided by the manual of CivilFEM and also we can get the tutorial from University Alberta that is related to study. There are three levels of tutorial provided by University Alberta which is basic, intermediate, and advance.

To achieve the objective, this 3D model will be applied with vertical and horizontal load. We can obtain the result on cracking and deflection of the reinforced concrete structure when the solution is done by the ANSYS software.



## **CHAPTER 2**

### **LITERATURE REVIEW**

#### **2.1 Introduction**

This chapter will discuss about related topics regarding the reinforced concrete frame. In era of globalization, most of the analysis that involves long calculation is done by the computer. Computer software provides accurate result and save much more time compare to manual calculation. With advances in computer systems, many software is produced to solve complicated analysis. As example, Esteem and STAAD.Pro are two another softwares instead of ANSYS that can be used to design and analyzed concrete structure. In this chapter we will discuss about reinforced concrete frame, advantages of reinforced concrete frame, and the behavior of reinforced concrete frame. There are some experiments about the comparison between braced frame and unbraced frame has been made before. So this thing will also be discussed in this chapter.

#### **2.2 Reinforced Concrete Frame Structure**

In Malaysia, reinforced concrete frame structure is applied to most of the high rise building. This thing happen due to much advantages of reinforced concrete structure compare to steel. According to research done by S. Mohr, J.M. Bairán and A.R. Marí (2010), reinforced concrete frame structure failure occur because the structure is subjected to combination of high level of shear and normal stresses. This kind of failure is not a new thing in construction field. We always think that the failure only occur to old structure but it also can happen to the new construction design that used new design codes. Shear stress failure usually occurs when the structure is subjected to earthquake.

For example, we can see what happen in Kobe city and Taiwan, there are many buildings collapse and cannot be used after earthquake disaster happen in their country.

According to F.J. Vecchio (1992), shear deformation in concrete frame structure cannot be predicted. So it is very importance to analysis the behavior of the structure before it is designed. There is a research program that is done by his team. The objective of the research is to predict the respond of beam section when combination shear, moment and axial load are applied to the beam. The method they use for this experiment by creating a model of beam cross section. They use concept and formulation of modified compression field theory for the analysis. The result found that the effects of concrete compression strain-softening and tension-stiffening are critical factors in determining the response of a reinforced concrete beam section against shear. Therefore, additional experimental work needs to be taken to get better understanding about the magnitude and influence of shear deformation in frame structure.

### **2.3 Advantages of Concrete Structure**

According to A.G. Gerosa (2005), both concrete and steel structure has its own advantages. But, when we compare in term of safety, concrete is better that steel. For high rise building, concrete structure provides more safety advantages. Concrete can resist high temperature for long time depends on the cover of the concrete structure. The thicker the cover of concrete, it can exposed to fire for a long period of time. Concrete also provide good protection from explosion. With proper design, concrete structure can resist wind force up to 200 miles per hour which means that concrete structure is good to build at the windstorm area. According to the Portland Cement Association (2007), modulus elasticity for concrete is from 24 GPa to 65 GPa while steel has modulus of elasticity 187 GPa. In this aspect, we can see that steel has higher modulus elasticity. This is why reinforced steel is embedded in concrete structure. This composite material provides high strength for compression and tension force.

## 2.4 Finite Element Method (FEM)

Finite element method is a numerical method to solve the engineering problem. It is a very powerful method in performing linear and non-linear problem in structural analysis. Finite element analysis was first developed in 1943 by R.Courant who utilized the Ritz method of numerical analysis and minimization of variation calculus to obtain appropriate solutions to vibration system. According to Gallagher (1975), for the manual calculation purpose, finite element method is one of easiest method for the analysis. The finite element method is such a widely used for analysis and design technique that it is essential that undergraduate engineering students have a basic knowledge of the theory and applications of the technique. In this study, finite element method is used to analyze the frame structure.

A study on developing a mixed frame finite element formulation of reinforced concrete and FRP composite column is done by C.G. Cho and M. Kwon (2010) in order to give more accuracy not only to predict the global behavior of the structure system but also to predict the local damage in the cross section. The strength enhancement of concrete is determined by the failure surface of concrete in a tri-axial stress state, and its corresponding peak strain was computed by the strain-enhancement factor proposed in this study. The behavior of FRP jacket was modeled using the dimensional classical lamination theories. The flexural behavior of concrete and composite member was defined using non-linear fiber cross-sectional approach. The result obtained by developed mixed finite element formulation was verified with the experiments of concrete composite column and also were compared with a displacement based finite element formulation. It is shown that the proposed formulation gives more accurate result in the global behavior of the column system as well as in the local damage in the column sections.

A research done by Imad Adel Ahmad Al-Qasem, Ibrahim Mohamad Ahmad Arman (2013) to study on crack in concrete frame by finite element modeling. In this research, a simple portal frame is created in ANSYS software and different load condition is applied on the structure to study the behavior of the structure. The main objective of the research is to know the effect of stress and the crack pattern on portal

frame that occur because of the stress. The material used for this research is only concrete and reinforcement is embedded in the concrete. The model of portal frame is created using shell elements and this model is analyzed in three different cases. For the first case, the model structure will be applied with vertical load and the stress and crack will be determined. For the second case, the model structure will be applied with horizontal load. For the third case, the model will be applied with different settlement. The stress and crack will occur on the beam and column.

## **2.5 Loads On Frame Structure**

There are many type of load need to be considered when designing frame structure. Each of this loads give different behavior to the structure. if one load is missing in analysis, the structure will not be safe if additional load is applied to them.

### **2.5.1 Static Force**

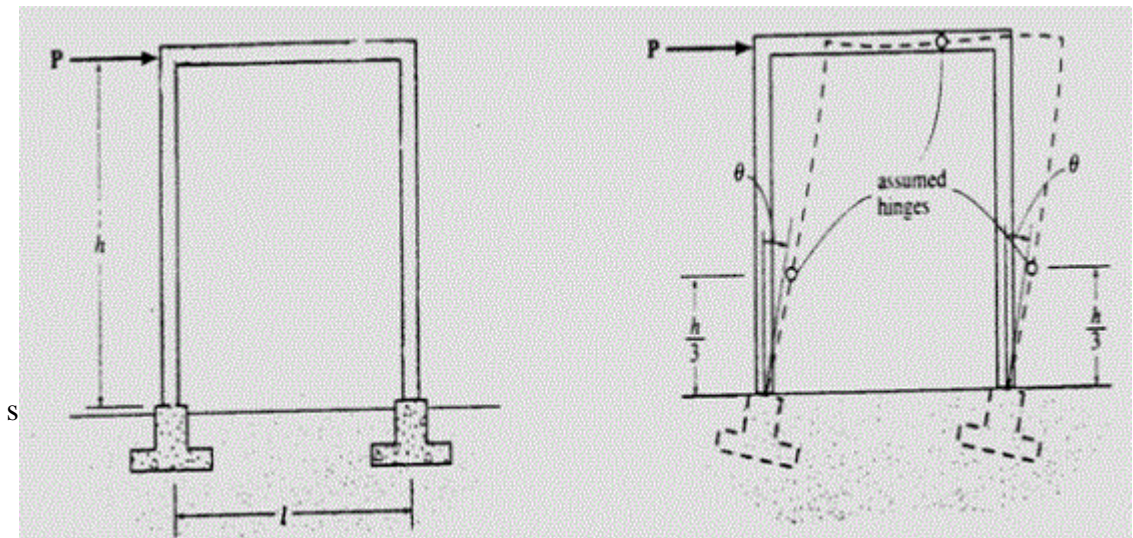
Static force can also be called as vertical load. Vertical load can be divided into two which is dead load and live load. Vertical load is the most important thing to be considered in analysis purpose because it contributes high value of load apply compare to the other load.

According to Myung-Rag Jung, Dong-Ju Min and Moon-Young Kim (2013), dead load is non-movable and fixed load that always apply to the structure. There are several things that subjected to dead load such as self-weight of the structure, brick wall, slab, mechanical equipment, finishing floor, and any building properties. Live load is movable load and not fixed. The value always change depend on the weight of item. Example of live load is furniture, human and other similar.

This dead load and live load value are already provided in Eurocode 2. The value is depends on the place. Different kind of place has different value of vertical load. For the analysis purpose, the value of vertical load need to be determined correctly to make sure the design stage will not have any problem.

## 2.5.2 Wind Load

Wind load can also be called as lateral load. Wind load is applied horizontal to frame structure. According to D. Adhers (2000), wind load design for unbraced frame is quite common in this recent year. We need to consider wind load design when the building is up to two-storey and above. When high wind pressure applied to high rise building, it will produce shear and overturning moment that can cause the building to



**Figure 2.1 :** Wind load effect to the frame

Source : <http://theconstructor.org/>

As we can see from the above figure, the presence of wind load causes horizontal deflection on the frame. Deflection of frame also cause cracking on the structure.

There are several method can be used to avoid horizontal deflection on the frame structure:

- i) Braced frame
- ii) Rigid frame
- iii) Shear wall

For the reinforced concrete frame, the best way to avoid horizontal deflection is by using shear wall. This is because we can avoid using brick wall and can save cost for the

construction. Certain high rise building use combination of that three method and other advance method to make sure the building is in good condition.

### **2.5.3 Earthquake**

According to Widodo Pawirodikromo (2013), earthquake is a natural disaster that happen due to release of energy in earth crust. This energy will produce seismic wave. Earthquake will cause building to vibrate and cause wave motion. Usually the earthquake will cause structure failure at the lower part. This thing happens because of ground motion from the earthquake.

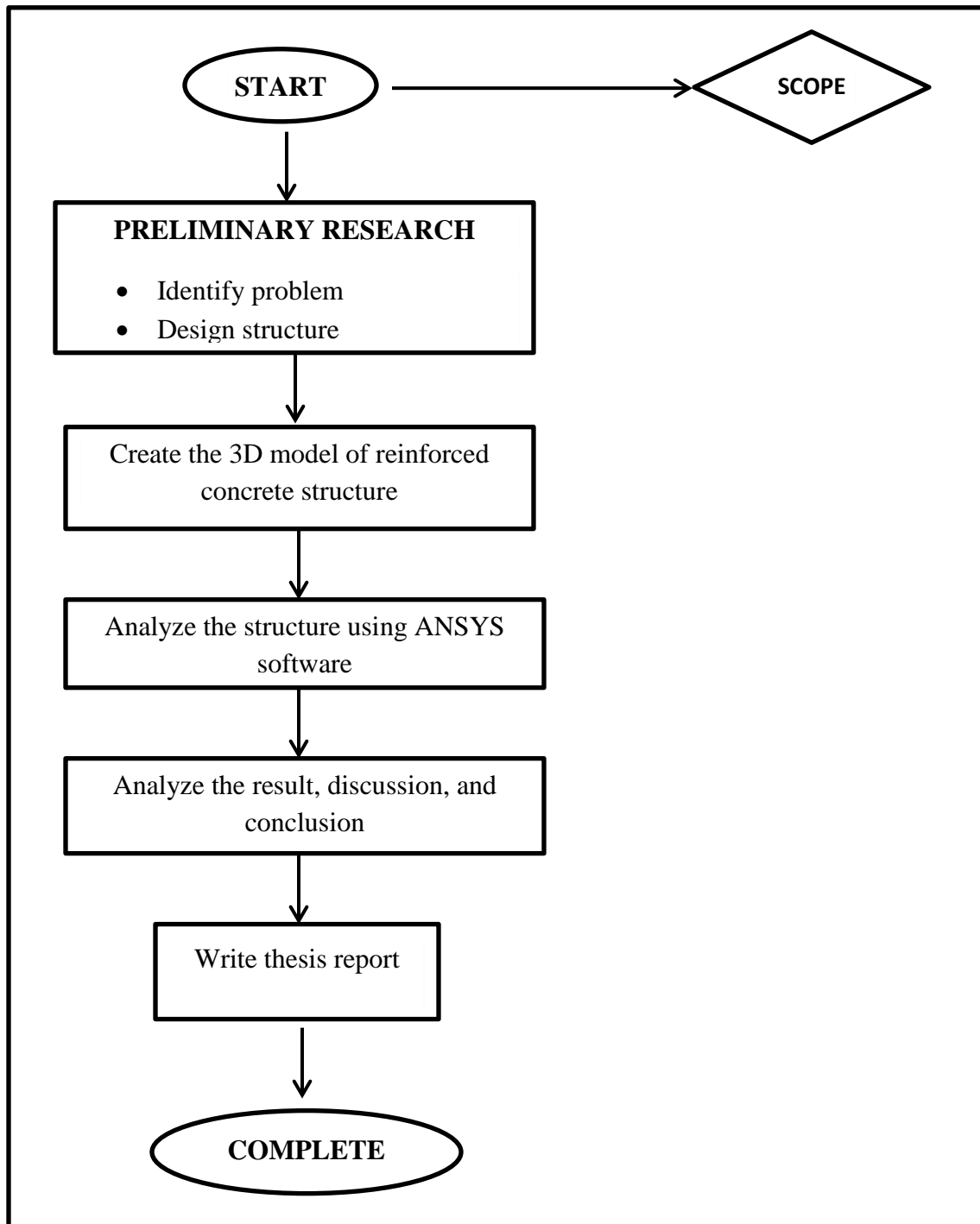
## **CHAPTER 3**

### **METHODOLOGY**

#### **3.1 Introduction**

This chapter will discuss about the method used to analyze the reinforced concrete structure model. Finite element method (FEM) software is used to analyze the unbraced frame model. We use ANSYS software to analyze the structure and to get related information regarding the structure model. ANSYS is a software that used finite element method for numerical solving for both linear and non-linear. In ANSYS, the analysis is divided into three stages which are preprocessing, solution, and post processing. In preprocessing part, the model is built using key point and lines after we define the element type, materials and cross section of the structure. For solution stage, we will specify the type of support, vertical load, and horizontal load for the structure. In post processing stage, all the checking will be applied to the model.

In this study, four storey 3D reinforced concrete frame structure is model and analyzed. Vertical load and horizontal load (wind load) is applied to the model. It is important to consider wind load in our design to make sure the structure will not failed when it exposed to wind storm.



**Figure 3.1 :** Simplified research methodology for this study

The methodology of the project is summarized in figure 3.1. All the work done during two semester is written in the flow chart.