

FINITE ELEMENT ANALYSIS OF 3D
TRANSMISSION TOWER USING ANSYS

LEE JIA WEI

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 2015

ABSTRACT

This thesis presents a probabilistic analysis approach applied in finite element analysis for modelling of 3D transmission tower with different parameters. The aim of this study analyse the transmission tower using Probabilistic Design System (PDS) to obtain a transmission tower that is sustainable and long term life. Furthermore, the transmission tower is also conducted under Eurocode 3 checking that is steel design. In probabilistic analysis, the results of probabilistic density function, cumulative distribution function, sample history plot, histogram plot and sensitivity plot for any input and output can be obtained. The transmission tower is analysed at 45 and 10000 simulations. The scope for this study is by using ANSYS as a software modelling process based on Finite Element Analysis and the selection of a 3D transmission tower structure based on the existing dimensions. From the Eurocode 3 checking, the structure is examined under tension, compression, bending moment resistance, shear resistance, compression buckling and lateral buckling. The green colour represents that all the members in the structure pass the checking while red colour indicates failure in the particular member. In addition, the results extracted from the simulation of 3D transmission tower model using ANSYS software are gathered and analysed to form a conclusion. There are two sources of data which are input and output parameters. There are eight input parameters which are DEADLOAD1, DEADLOAD2, DENS, ELASTIC, POISSON, TEMP, WINDLOAD1 and WINDLOAD2. In this thesis, Monte Carlo Simulation and Response Surface simulation was used to analyse the effect of parameter on the transmission tower structure. From the corresponding graphs, the values for mean, standard deviation, skewness, kurtosis, minimum and maximum relative frequency can be obtained. In conclusion, there are some limitations in this study where the literature review that is related is difficult to be found. However, it is proved that ANSYS software is capable to get results for reaction forces, deformation, axial forces, and maximum deflection by probabilistic analysis for transmission tower structures.

ABSTRAK

Tesis ini membentangkan pendekatan analisis kebarangkalian digunakan dalam analisis unsur terhingga untuk pemodelan menara penghantaran 3D dengan parameter yang berbeza. Tujuan kajian ini menganalisis menara penghantaran menggunakan Sistem reka bentuk kebarangkalian (PDS) untuk mendapatkan sebuah menara penghantaran yang hidup jangka mampan dan panjang. Tambahan pula, menara penghantaran juga dijalankan di bawah Eurocode 3 pemeriksaan yang reka bentuk keluli. Dalam analisis kebarangkalian, keputusan fungsi kebarangkalian ketumpatan, fungsi taburan kumulatif, sampel sejarah plot, histogram plot dan plot sensitiviti bagi apa-apa input dan output boleh diperolehi. Menara penghantaran dianalisis pada 45 dan 10000 simulasi. Skop kajian ini adalah dengan menggunakan ANSYS sebagai proses pemodelan perisian berdasarkan Terhingga Analisis Unsur dan pemilihan struktur menara penghantaran 3D berdasarkan dimensi yang sedia ada. Dari Eurocode 3 memeriksa, struktur itu diperiksa di bawah ketegangan, mampatan, lenturan rintangan masa, rintangan ricih, lengkokan mampatan dan lengkokan sisi. Warna hijau melambangkan bahawa semua ahli-ahli dalam struktur lulus pemeriksaan manakala warna merah menunjukkan kegagalan ahli tertentu. Di samping itu, keputusan yang diekstrak daripada simulasi 3D model menara penghantaran menggunakan perisian ANSYS dikumpulkan dan dianalisis untuk membentuk kesimpulan. Terdapat dua sumber data iaitu input dan output parameter. Terdapat lapan parameter input yang DEADLOAD1, DEADLOAD2, liang elastik, POISSON, TEMP, WINDLOAD1 dan WINDLOAD2. Dalam tesis ini, Monte Carlo Simulasi dan Tindak Balas Permukaan simulasi digunakan untuk menganalisis kesan parameter kepada struktur menara penghantaran. Daripada graf sepadan, nilai min, sisihan piawai, kepencongan, kurtosis, minimum dan kekerapan relatif maksimum boleh diperolehi. Kesimpulannya, terdapat beberapa batasan dalam kajian ini di mana kajian literatur yang berkaitan adalah sukar untuk ditemui. Walau bagaimanapun, jika dibuktikan bahawa perisian ANSYS mampu untuk mendapatkan keputusan untuk daya tindakbalas, ubah bentuk, daya paksi, dan pesongan maksimum oleh analisis kebarangkalian untuk struktur menara penghantaran.

TABLE OF CONTENTS

	Page
SUPERVISOR'S DECLARATION	ii
STUDENT'S DECLARATION	iii
ACKNOWLEDGEMENTS	v
ABSTRACT	vi
ABSTRAK	vii
TABLE OF CONTENTS	viii
LIST OF TABLES	xi
LIST OF FIGURES	xii
CHAPTER 1 INTRODUCTION	
1.1 Introduction	1
1.2 Background of Study	1
1.3 Problem Statement	2
1.4 Research Objective	2
1.5 Scope of Study	3
1.6 Significance of Study	3
CHAPTER 2 LITERATURE REVIEW	
2.1 Introduction	4
2.2 Transmission Tower	4
2.3 Steel and Truss	5
2.4 Finite Element Analysis	5
2.5 ANSYS	6

CHAPTER 3 METHODOLOGY

3.1	Introduction	7
3.2	Flow Chart of Methodology	8
3.3	Pre-Processor Process	9
3.3.1	Specify the Title	9
3.3.2	Activate CivilFEM	10
3.3.3	Enter KeyPoints	11
3.3.4	Form the Lines	12
3.3.5	Copy Key Points	13
3.3.6	Form the Lines Again	16
3.3.7	Divide Lines	17
3.3.8	Form the Lines Again	19
3.3.9	Define Element Types	20
3.3.10	Define Materials	21
3.3.11	Define Cross Sections	22
3.3.12	Define Member Properties	23
3.3.13	Define Beam & Shell Properties	25
3.3.14	Mesh Size	26
3.3.15	Mesh	28
3.3.16	Saving Work	29
3.4	Solution Process	30
3.4.1	Apply Constraints	30
3.4.2	Apply Loads	32
3.4.3	Apply Wind Loads	34
3.4.4	Solving the System	36
3.5	Post Processor Process	37
3.5.1	Code Checking & Beam Results	37
3.5.2	Reaction Forces	44
3.5.3	Deformation	45
3.5.4	Deflection	47
3.5.5	Forces & Moments	50

CHAPTER 4 RESULTS AND ANALYSIS

4.1	Introduction	52
4.2	Probabilistic Results from ANSYS	53

4.2.1	Random Input Variables	53
4.2.2	Statistics of the Probabilistic Results	58
4.2.3	Sample History Plots	59
4.2.4	Histogram Plots of Input Variables	66
4.2.5	Histogram Plots of Output Variables	72
4.2.6	Cumulative Distribution Function Plots	73
4.2.7	Sensitivity Plots	80
4.2.8	Linear Correlation Coefficients	81
4.2.9	Spearman Rank Order Correlation Coefficients	82

CHAPTER 5 CONCLUSIONS AND RECOMMENDATIONS

5.1	Summary of the Study	84
5.2	Limitations	85
5.3	Suggestions	85
5.4	Conclusion	86

REFERENCES	88
-------------------	-----------

APPENDICES

A	Log File of 3D Transmission Tower Analysis	90
B	Eurocode 3 Checking of Transmission Tower	100
C	Stiffness Matrix of Transmission Tower	105

LIST OF TABLES

Table No.	Page
3.1 Coordinates of keypoints	11
3.2 Coordinates of keypoints	12
3.3 Z-coordinate of keypoints	14
3.4 Force entered	33
3.5 Force entered	35
4.1 Random Input Variables Specifications	53
4.2 Statistics of the Random Input Variables	59
4.3 Statistics of the Random Output Variables	59
4.4 Linear Correlation Coefficients between Input Variables	82
4.5 Linear Correlation Coefficients between Input and Output Variables	82
4.6 Linear Correlation Coefficients between Output Variables	82
4.7 Spearman Rank Order Correlation Coefficients between Input Variables	83
4.8 Spearman Rank Order Correlation Coefficients between Input and Output Variables	83
4.9 Spearman Rank Order Correlation Coefficients between Output Variables	83

LIST OF FIGURES

Figure No.		Page
3.1	Flow chart of the research process	9
3.2	Change title step	9
3.3	Change title window	9
3.4	Activate CivilFEM window	10
3.5	CivilFEM setup options window	10
3.6	Create keypoint window	11
3.7	Keypoints of transmission tower	12
3.8	Lines in active coord window	13
3.9	Model appears	13
3.10	Copy keypoints window	14
3.11	Copy keypoints window	15
3.12	Model appears after copying	15
3.13	Model appears	16
3.14	Lines in active coord window	16
3.15	Model appears	17
3.16	Divide line steps	17
3.17	Divide line in N lines window	18
3.18	Divide line in N lines window	18
3.19	Divided lines model	19

3.20	Lines in active coor window	19
3.21	Model of transmission tower	20
3.22	Beam element types for code checking window	20
3.23	Material browser window	21
3.24	Material browser window	21
3.25	Cross sections explorer window	22
3.26	Hot rolled shapes library window	22
3.27	Cross sections explorer window	23
3.28	Member properties window	23
3.29	Member property 1 window	24
3.30	Member properties explorer window	24
3.31	Beam and shell properties window	25
3.32	Beam 1 window	25
3.33	Beam and shell properties window	26
3.34	Mesh size steps	26
3.35	Element sizes on all selected lines window	27
3.36	Model appears after mesh size	27
3.37	Mesh line steps	28
3.38	Mesh lines window	28
3.39	Model appears after meshing	29
3.40	Saving work steps	29
3.41	Apply constraint steps	30

3.42	Apply U,ROT on KPs window	30
3.43	Apply U,ROT on KPs window	31
3.44	Applied constrain model	31
3.45	Apply load steps	32
3.46	Apply F/M on KPs window	32
3.47	Apply F/M on KPs window	33
3.48	Model with load	33
3.49	Apply wind load steps	34
3.50	Apply PRES on beams window	34
3.51	Apply PRES on beams window	35
3.52	Model with wind load	35
3.53	Solving steps	36
3.54	Status command window	36
3.55	Note window	37
3.56	Code checking steps	37
3.57	Code checking	38
3.58	Beam result steps	38
3.59	Graph steel results window	39
3.60	Tension result	39
3.61	Compression result	40
3.62	Bending moment result	40
3.63	Shear result	41

3.64	Bending & shear result	41
3.65	Bending & axial force result	42
3.66	Bending, axial & shear result	42
3.67	Compression buckling result	43
3.68	Bending buckling result	43
3.69	Bending & compression buckling result	44
3.70	Reaction solution steps	44
3.71	List reaction solution window	45
3.72	List of reaction forces	45
3.73	Plot deformed shape steps	46
3.74	Plot deformed shape window	46
3.75	Result of deform shape	47
3.76	Plot result steps	47
3.77	Contour nodal solution data window	48
3.78	Deflection result appears	48
3.79	List result steps	49
3.80	List nodal solution window	49
3.81	List nodal solution window	50
3.82	Forces & moments steps	50
3.83	Graph force and moment results window	51
3.84	Forces and moments result window	51
4.1	PDF & CDF of Input Random Variable DEADLOAD1	54

4.2	PDF & CDF of Input Random Variable DEADLOAD2	54
4.3	PDF & CDF of Input Random Variable DENS	55
4.4	PDF & CDF of Input Random Variable ELASTIC	55
4.5	PDF & CDF of Input Random Variable POISSON	56
4.6	PDF & CDF of Input Random Variable TEMP	57
4.7	PDF & CDF of Input Random Variable WINDLOAD1	57
4.8	PDF & CDF of Input Random Variable WINDLOAD2	58
4.9	Sampled Values for Output Parameter MAXIMUMDEFLECTION	60
4.10	Mean Value History for Output Parameter MAXIMUMDEFLECTION	60
4.11	Standard Deviation History for Output Parameter MAXIMUMDEFLECTION	61
4.12	Minimum Value History for Output Parameter MAXIMUMDEFLECTION	62
4.13	Maximum Value History for Output Parameter MAXIMUMDEFLECTION	62
4.14	Sampled Values for Output Parameter MAX_DEFLECTION	63
4.15	Mean Value History for Output Parameter MAX_DEFLECTION	63
4.16	Standard Deviation History for Output Parameter MAX_DEFLECTION	64

4.17	Minimum Value History for Output Parameter MAX_DEFLECTION	64
4.18	Maximum Value History for Output Parameter MAX_DEFLECTION	65
4.19	Sampled Values for Output Parameter MAXIMUMDEFLECTION	66
4.20	Sampled Values for Output Parameter MAX_DEFLECTION	66
4.21	Histogram of Input Variable for DEADLOAD1	67
4.22	Histogram of Input Variable for DEADLOAD2	68
4.23	Histogram of Input Variable for DENS	69
4.24	Histogram of Input Variable for ELASTIC	69
4.25	Histogram of Input Variable for POISSON	70
4.26	Histogram of Input Variable for TEMP	70
4.27	Histogram of Input Variable for WINDLOAD1	71
4.28	Histogram of Input Variable for WINDLOAD2	71
4.29	Histogram of Output Variable for MAXIMUMDEFLECTION	72
4.30	Histogram of Output Variable for MAX_DEFLECTION	73
4.31	CDF of Input Variable DEADLOAD1	74
4.32	CDF of Input Variable DEADLOAD2	75
4.33	CDF of Input Variable DENS	75
4.34	CDF of Input Variable ELASTIC	76
4.35	CDF of Input Variable POISSON	76

4.36	CDF of Input Variable TEMP	77
4.37	CDF of Input Variable WINDLOAD1	78
4.38	CDF of Input Variable WINDLOAD2	78
4.39	CDF of Output Variable MAXIMUMDEFLECTION	79
4.40	CDF of Output Variable MAX_DEFLECTION	79
4.41	Sensitivity Plot for MAXIMUMDEFLECTION	80
4.42	Sensitivity Plot for MAX_DEFLECTION	81

CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION

This chapter will introduce readers to this study. Furthermore, readers will manage to get information on background of study, problem statement as well as objectives, and scope of the study.

1.2 BACKGROUND OF STUDY

Nowadays, our world is becoming more developed and developing as we know. Due to this, power consumption has continued to rise and this in turn had led to the increase in the number of power stations as well as power transmission towers from the generating stations to the load centres. Electrical power demand has been increasing around the world and many large-scale transmission towers have been newly constructed [1].

Generally, transmission tower is a medium to carry power loads from one station to another station. It is usually composed of steel and is able to run at long distances. Transmission tower consist of several types and designed in accordance to the tower height and capacity of tower to support load from the conductor, compression load, wind load, vertical load, longitudinal load and uplift load. Transmission towers are usually used where large amount of electrical current is to be distributed often ranging from 115,000 to 800,000 volts. Transmission towers are a vital component and management to assess reliability is needed to minimise the risk of power supply disruption that may result from in-service tower failure [2]. Towers are widely regarded as one of the most difficult form of lattice structure to analyse [2].

1.3 PROBLEM STATEMENT

The use of electrical power has become an increasingly important part of the economy of industrial countries. Transmission tower supports the phase conductors and earth wires of transmission line [3]. The importance of the transmission tower on national economy and people's living has been well recognized. Unfortunately, natural disasters such as earthquakes and floods come without notice. These natural phenomena caused major damages to the transmission towers. During the attack of the Ji-Ji earthquake, with a size of 7.3 in Richter magnitude, in Taiwan on Sept. 21, 1999, the strong vibration of ground motion has caused the collapse of a main transmission tower located in the central region of the state [4]. Many transmission towers were built in the mountains or crossing rivers under rugged circumstances to overcome the loss electrical supply.

Before designing and planning in a construction, analysis of the structure becomes the main role in the process. Transmission tower structures are generally analysed by linear static analysis methods but it is also necessary to determine the deformation of the structure as well so that any failure can be avoided. In addition, Finite Element Analysis was applied towards the design of the transmission tower. However, using this manual method is less efficient and less practical. Therefore, computer software called ANSYS was used to compute calculation in order to create a safe and optimum design.

1.4 OBJECTIVES

An objective of study is an important aspect to be considered in any research. It might be useful as a guideline in preparing a research in order to get the expected result.

The main objective of this study is to analyse the transmission tower using Probabilistic Design System (PDS). The sub-objectives are:

- i. To determine the reaction forces, deformation, deflection, forces and moments.

- ii. To study the purpose of code checking on the structure.

1.5 SCOPE OF STUDY

During this study, several scopes are set to ensure that the objectives of this study can be achieved and any other variables that are not considered in the scope can be eliminated. The factors that are considered are:

- i. Selection of a transmission tower structure.
- ii. Using ANSYS as a software modelling process based on Finite Element Analysis.

1.6 SIGNIFICANCE OF STUDY

Significance of study is a general outline on the importance of the study which will be carried out. In terms of a research paper, the significance of study refers to the expected outcome from the research. Firstly, this study is important to produce a structure without failures. Failures within a structure can cause incidents such as collapse or breakdown when dealing with other environmental factors. Besides that, by doing this research, it is important to develop a structure that is environmentally viable as well as politically acceptable. If these factors are not accounted in the research, it might bring unwanted occasions such as rebellion from the public.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

Literature review contains four topics that discuss specifically and review on past study. The topics are transmission tower, steel and truss, finite element analysis and ANSYS. These topics are important when we analyse and design a transmission tower structure.

2.2 TRANSMISSION TOWER

In general, transmission tower is defined as a set of steel structure that distributes electrical power through transmission lines. Towers or masts are structures that are built in order to fulfil the need for placing objects or persons at a certain level above the ground [3]. Transmission tower can be modelled by truss or frame element [5]. The result of a field test was compared to those of a finite element analysis. The comparison indicated that tower member can be modelled adequately by truss element; it shows that transmission tower also can be concluded as truss structure.

Recent developments in manufacturing of truss structure appear to greatly extend their application possibilities where that application possibility also involves a transmission tower [6]. Besides that, in the paper titled “The united design method of a transmission tower and the foundation” mentioned that transmission tower is designed as the truss structure [7]. Truss structures are ubiquitous in the industrialized world, appearing as bridges, tower, as roof support and building exoskeletons [8].

2.3 STEEL AND TRUSS

Truss can also act like a beam which involves bending resistance by couple created by force in the top and bottom member. In accordance to researchers who present different design philosophy, where tower are modelled with beam elements [9]. Truss structure can consist of rigid beam, pin connected at joint and also exerting axial force only [8].

Steels are the materials that commonly used for truss structure as well as a transmission tower. It is contended that the steel as a building material has been used extensively by various types of structure which concludes high-rise building skeletons, industrial building, transmission tower, railway bridges, overhead tank, bunkers and silos [10]. Transmission tower structures are generally constructed using symmetric thin-walled angle section members which are eccentrically connected [2] while latticed transmission towers are constructed using eccentrically connected angle section members [11].

2.4 FINITE ELEMENT ANALYSIS

Based on the appearance of the transmission tower itself, we can imagine the difficulty in the time constraint of analysing, designing and also constructing this structure. Consequently, finite element method is used in order to simplify the processes. Moreover, Finite Element Analysis (FEA) is an engineering software with the purposed of accepting input data and determined a structure design to meet the performance criteria. It can be used in analyse and design transmission tower. Finite Element Analysis (FEA) was first developed in 1943 by R.Courant [12]. He has utilized the Ritz method of numerical analysis and minimization of variation calculus in order to obtain approximate solutions to vibration systems. Shortly after that, M. J. Turner, R.W. Clough, H.C. Martin, and L. J. Topp established a broader definition of numerical analysis in a paper published in 1956. “Stiffness and deflection of complex structure” is the aim of this paper.

By using finite element, numerical solutions for complicated stress problem can be solved by using Finite Element Analysis (FEA). Finite element analysis software available are such as LUCAS, MSC Patran, ANSYS, ABAQUS Algor (FEMPRO), STAAD PRO, RISA, PROKON, MIKROSTRAN, BRICSCA and many more. Besides this, AK Tower is also a finite element computer program that used geometric and material nonlinear analysis to stimulate the ultimate structural behaviour of lattice transmission tower as stated by [2].

2.5 ANSYS

Despite all software that has been stated, this study only applies the usage of ANSYS software. ANSYS is a finite element analysis package used widely in industry to simulate the response of a physical system to structural loading, and also thermal and electromagnetic effects. It also uses the finite element method to solve the underlying governing equations and the associated problem-specific boundary conditions like have been stated by Steve Weidner (2012) and edited by Chia-Hsun Hsieh (2012).

Design Optimizing before the construction begins is the best way to ensure no time and cost overruns. It also helps in ensuring that the individual structural components work properly, placement and designing HVAC equipment, to protect buildings occupants and structures from disaster by devising smoke management systems, modelling air flow for occupant comfort, and last but not least analysing environmental-structural effects are all possible through ANSYS engineering simulation software. With its modular structure, ANSYS software gives the opportunity for taking up only needed features. By adding CAD and FEA connection modules, ANSYS can work to integrate with other used engineering software on the desktop. ANSYS have ability to import CAD data and also able to build geometry with its “pre-processing” abilities. Thus, the finite element model is generated. Results can be viewed as numerical and graphical after defining loadings and carrying out analyses. ANSYS too can carry out advanced engineering Element Method where pioneering mathematical formalism of the method is being built.

CHAPTER 3

RESEARCH METHODOLOGY

3.1 INTRODUCTION

In this chapter, readers will get brief information on the methods that are used to get information for this study. ANSYS software is used to run this project in order to achieve the objectives. Generally, ANSYS program is computer program for finite element analysis and design. The analysis process of transmission tower using ANSYS will be conducted by using the following steps which are Pre-processing Process, Solution Process and Post-processing Process.

Pre-processing Process is the step where defining the problem takes part. This process includes defining project title for this project, modelling process which includes creating the key points and also forming lines, defining the element being used, its geometry properties, element material properties, mesh size and meshing process.

Solution Process is the step where load assigned and solving process take part. It includes defining the analysis type where for this project the analysis type is static. Apply constraints and also load to model and last but not least solving the problem

Post-processing Process is the step where results can be viewed. The result can be viewed in two forms which is in a form of figure that can symbolize the result for example a figure that shows a bend transmission tower due to load or numbering forms which shows all the data required such as deflection value.

3.2 FLOW CHART OF METHODOLOGY

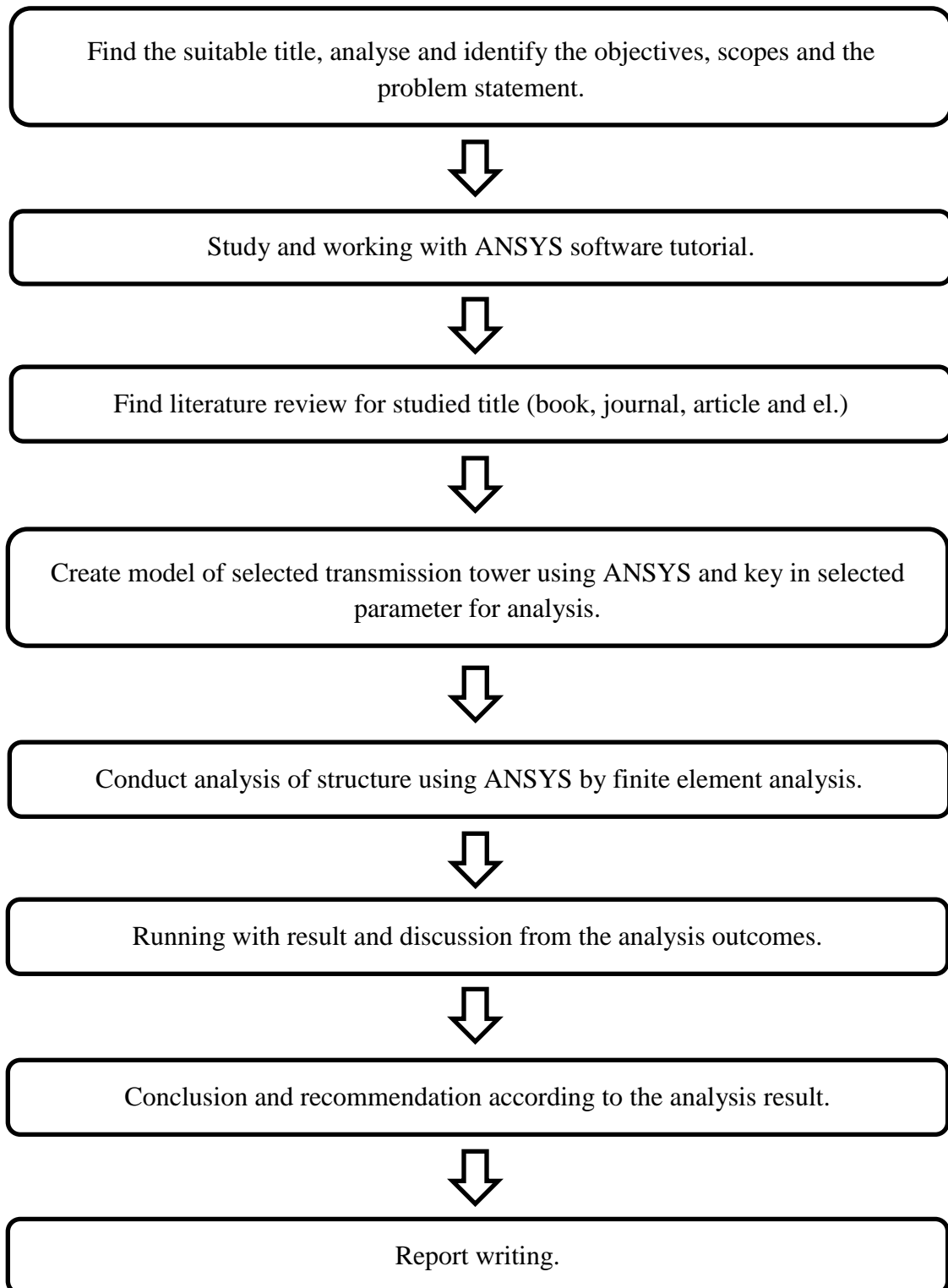


Figure 3.1: Flow chart of the research process

3.3 PRE-PROCESSOR PROCESS

3.3.1 Specify the Title

In the “Utility menu bar”, first select “File” followed by “Change Title”. It has been shown in the figure below.

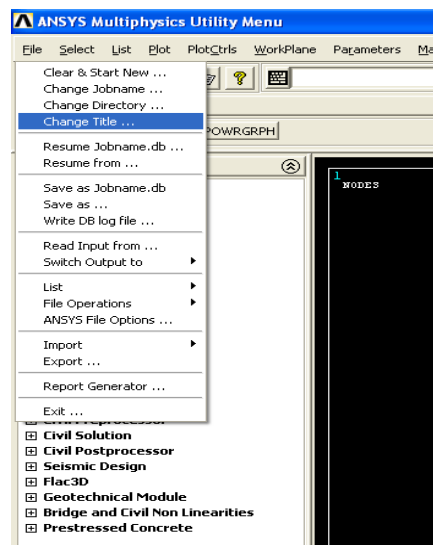


Figure 3.2: Change title step

Then the following window will appear. Enter file name for this project for example “3D Transmission Tower”. Next, click on “OK”.

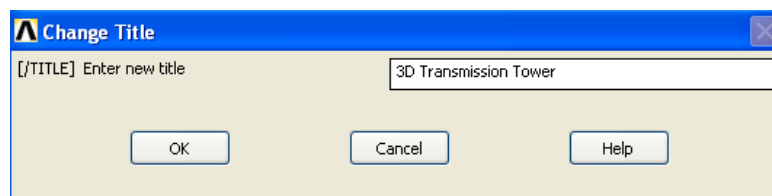


Figure 3.3: Change title window

Although this step is not required for the analysis process, it has been recommended to be done. It will help in order to identify the file that has been saved.

3.3.2 Activate CivilFEM

From “ANSYS main menu”, click on “CivilFEM” and the following window will appear. Next, click on “OK”.

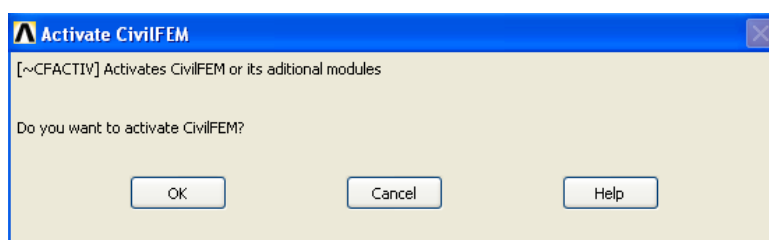


Figure 3.4: Activate CivilFEM window

Then click on “Civil Setup”, and click “OK”. This is where we can specify the codes and unit to be used as shown below.

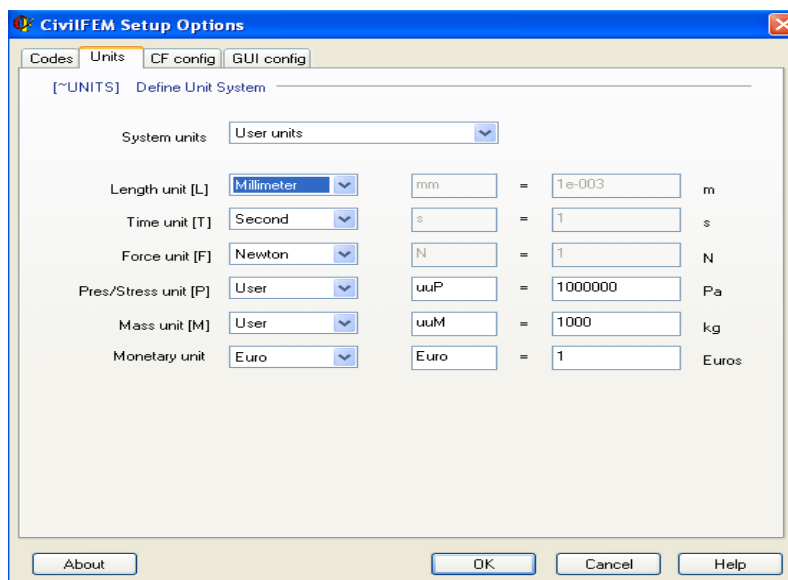


Figure 3.5: CivilFEM setup options window