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

CHAPTER 3 METHODOLOGY

CHAPTER 4 RESULTS AND ANALYSIS

LIST OF TABLES

LIST OF FIGURES

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 political 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

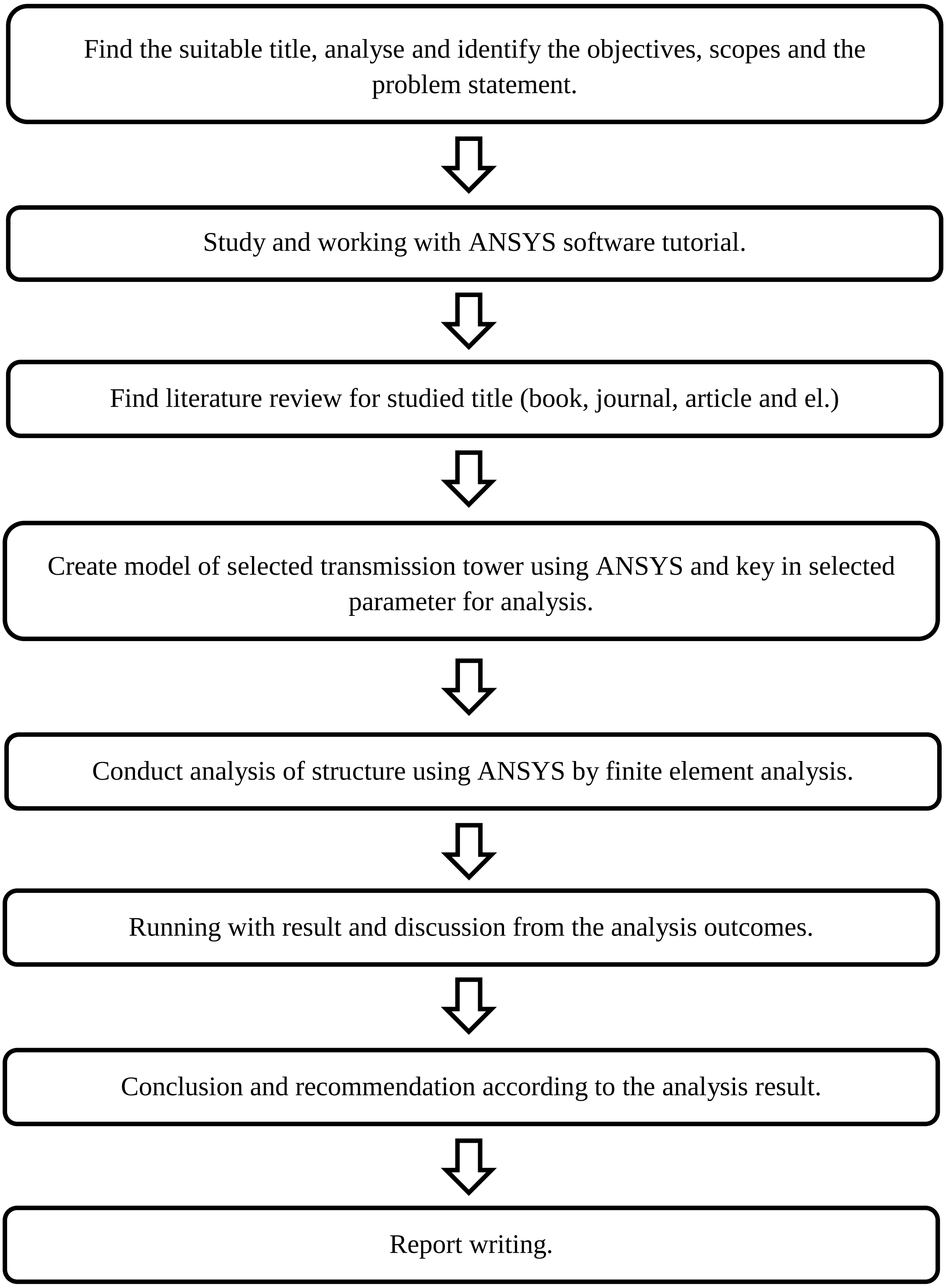

Figure 3.1: Flow chart of the research process

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.

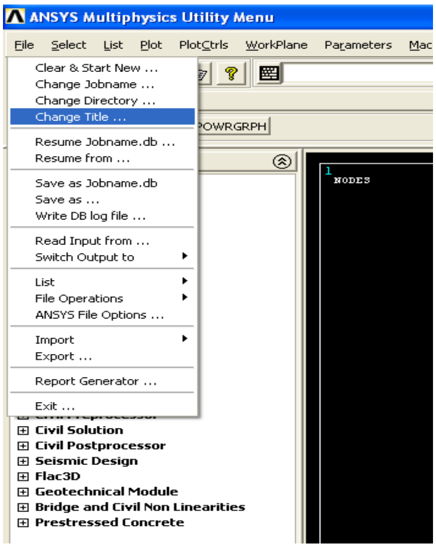

Figure 3.2: Change title step

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

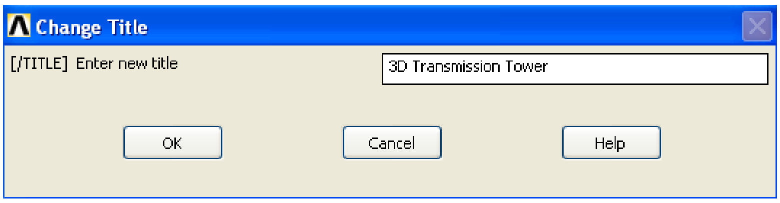

Figure 3.3: Change title window

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.

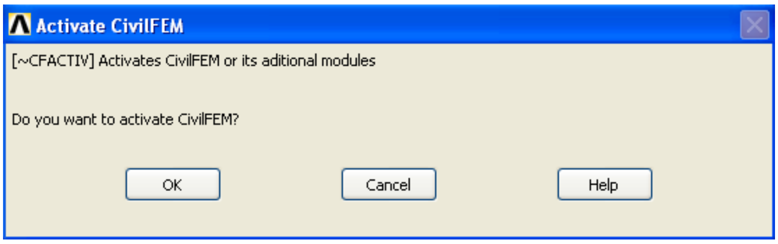

Figure 3.4: Activate CivilFEM window

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

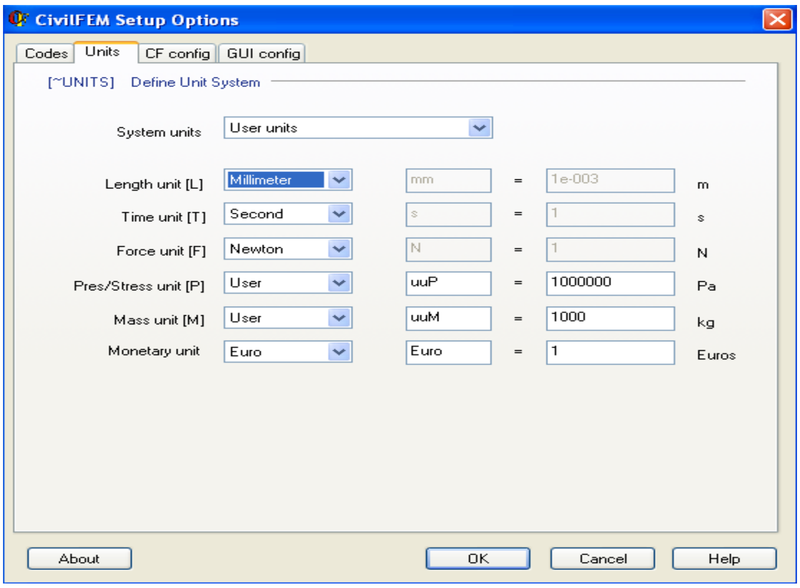

Figure 3.5: CivilFEM setup options window