CHAPTER 1

INTRODUCTION

1.1 Background of the Study

Finite element analysis is to replace the complicated problems with relatively simple problems and then solving. It will solve the field as many as by little interconnection finite element subdomains, for each unit there is a suitable approximate solution, and then solve the domain always satisfy condition, so as to get the solution of the problem. This solution is not the exact solution, but approximate solution, because the actual problem is replaced by a simple question. Since most practical problems difficult to get the accurate solution, and the finite element calculation not only high precision, but also can adapt to various kinds of complicated shapes, and therefore become an effective means of engineering analysis (Corley, 2004).

ANSYS software is the combination of filed form of financial structure, fluid mechanics, electric, magnetic, acoustic analysis. It's concentrated in one of the large general-purpose finite element analysis software (Liang, 2003). It is developed by the world's largest ANSYS finite element analysis software company from United States. It can interface with most CAD software, sharing and exchange of data. For specific areas of physics, ANSYS software allows users to delve deeper to solve a wider range of issues to deal with more complex situations, like the field of construction, exploration, geology, water conservancy, transportation, electric power, mapping, land, environment, forestry, metallurgy etc.
The box girder refer to the cross-sectional like a box in the form of beam. When the bridge span is huge, perform box girder is the best, it’s closed thin-walled section of its torsional stiffness beneficial for the bridge and curved bridge of cantilever construction. It has a large area of the roof and floor, can effectively resist the positive and negative moment reinforcement and satisfy the need. It has the good dynamic characteristics and small shrinkage deformation value (Hanson, 2009).

In this research, Finite Element Methods (FEM) models were used to stimulate the characteristic behavior of the steel, concrete and reinforcement steel structure using ANSYS+CIVILFEM 12.0 program. CivilFEM is the ANSYS civil engineering special software package based on the structure of civil engineering for a variety of numerical simulation of design and checking (Watson, 2007).

CivilFEM ANSYS powerful analysis ability and CivilFEM provides for civil engineering special functions and modules together, in order to meet the special needs of the civil engineering industry, provides a powerful tool for the design of a variety of high-end civil engineering analysis (Weng, 2008).

1.2 Problem Statement

In the modern century of bridge construction field, box girder is a significant component of many form of bridge, so, the box girder structures must be designed well with appropriate calculation to make sure it can support the load and it is safe for using in a long period of time (Mansur, 2008). Even though there are several methods which can be used to analysis the behavior of box girder structure, in order to make the procedure easier and specific, finite element method (FEM) is used through ANSYS software.

In the bridge construction process, in order to meet the alignment, driving comfort, people’s aesthetic and other requirements, often perform the bridge in a variety
of curve form and special-shaped bridge structure. But, in modern days, those requirements is not that easily to accomplish. In order to validate it feasibility and reliability, this study test the box girder by load though ANSYS+CIVILFEM software (Salam, 2007).

1.3 Objective

The purpose of this study is to investigate the structural characteristic and behavior of box girder by using finite element software ANSYS.

(i) To modeling the box girder in an ANSYS FEM design.
(ii) To verify box girder pass all the code checking.
(iii) To check the frame structure of box girder in force and moments, axial + bending and stress and strain graph.
(iv) To determine the result based on difference graph.

1.4 Scope of Study

This research is mainly concentrate on generating a 3-Dimension model though the ANSYS+CIVILFEM software. Those focused on the analysis the box girder using the EUROCODE 2 and applying the parameter to generate the results in ANSYS. For the purpose of accomplish the research objectives, there are few researches scope is necessary to be followed. Such as study the parameters of box girder in EURCODE and characteristic of the structure (Warzak, 2001).

This research will applying different size and type of box girder, generating different result by applying different loading on box girder. The result of this research will be comparing for come out the best result due to complete the research study. Explore ANSYS+CIVILFEM software though tutorial to get more and more familiar with operating the software (Warzak, 2001).