FINITE ELEMENT ANALYSIS OF REINFORCED CONCRETE BEAM WITH TWO CIRCULAR OPENING AT CONSTANT MOMENT AREA BY USING ANSYS SOFTWARE

BONG KIT HIAN

A 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

UNIVERISITI MALAYSIA PAHANG

JUNE 2014

ABSTRACT

Beam is one of the most important structural element for a building because it can withstanding primarily load by resisting bending. Thus, there is a need to have a proper design so that it have enough strength to be sustain loading acting on it. Here the study was conducted a beam with two circular web opening at constant moment area with the beam dimension 200 x 400 x 3000 mm. In order to analyse these beam, ANSYS software was been used in these study although there are still many existing commercial engineering software such as STAAD PRO, LUSAS, MATLAB and etc. This is because ANSYS is an engineering stimulation software which offer stimulation solution set that a design process requires. Besides, it is widely used in industry to stimulate the response of a physical system to structural loading. By using ANSYS software, it can help to analyse complex structure without simplify the structure and also help to analyse the reaction force, deformation, deflection and axial stress as well as obtain the maximum deflection. With this study, the finding seem has been fulfil to the objectives where the results of simulation of Reinforce concrete beams with two web opening such as maximum deflection, stresses and strains has been analysed with respect to six random inputs such as force, reinforcement diameter, link diameter, stirrups diameter and spacing of link.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

CHAPTER 1

INTRODUCTION

1.1 GENERAL

Nowadays, the use of web opening on beam structures has become most popular in building constructions. No matter it is use in residential building, high rise building, or low rise offices, it could bring a lot of benefit to all of us. For the example, the use of the openings can help to accommodate the essential services such as mechanical and electrical systems like the wiring for air-condition, electricity, telephone or computer network which are usually located around underneath the beam soffit that creating the dead space of false ceilings. The reason behind by passing these ducts through openings is to eliminating these dead spaces in false ceiling of multi-story building and it will results in a compact design. Besides that, the height of a building can be reduced due to the reduction of the dead spaces thus the amount of every material item which runs from floor to floor will be reduced and it indirectly show the significantly reducing construction costs. Therefore in the modern construction, the provision of beam with web openings has become an acceptable engineering to reduce the probability cutting holes in inappropriate locations.

However, finite element analysis has widely use because it can help engineer to analysis for the entire structure member whether it is in a building, bridge or model of transportation. Besides, it can help to solve the problem with certain degree of complication because of the result is more precise and save cost and time. Generally, finite element have two type of analysis used in industry either 2D modeling or 3D modeling. In this study, ANSYS software is used to analyse a 3D modelling of reinforced

concrete beam with two web opening. In this study, ANSYS software is used to analyse a 3D modelling of reinforced concrete beam with two web opening. Furthermore, ANSYS is the most advanced, comprehensive and reputable finite element and design software that based on civil or structural engineering projects such as to produce simulation code for the analysis and it is a pure structural simulation of making it a unique and powerful tool for a wide range of civil engineering projects (Sofialidis, 2000).

1.2   PROBLEM STATEMENT

Most of the studies are about the effects of web opening on the precast beams, deep beams, and T-beams only and there is limited data for the normal reinforcement rectangular concrete beam with web opening. Due to the limited depth of the rectangular reinforcement concrete beams thus the web opening in these beams introducing is very significant. According to Saksena (2013) the design of openings should not be more than half of the depth beam because it may effects on the behaviour of beam if the opening is over than half of the beam depth. Due to unexpected alterations in the sectional configuration or the changing in dimension of cross section of beam, the corner of opening will subjected to the high stress and it lead to the cracking which is unacceptable from aesthetic and durability. Besides, many diagonal cracks can also be developed above and below opening in reinforced concrete beam to high stress concentration at corner.

On the other hand, the reducing of stiffness may cause the deflection to rise to excessive under service load and these will result in a considerable distributing of force and internal moment (Mansur, 2006). Besides that, Mansur's (1998) finding also discussed the source of weakness for the present of opening that will effect to the failure plane that always passes through the opening unless the opening is closes to the support and bypass the potential inclined failure plane. Last but not least, the appropriate design of the web openings can also be critical to the strength and serviceability and help in reducing shear, bending, and torsion capacities of the members otherwise provide special reinforcement to the beam to avoid serious affected (Saksena, 2013).

1.3 RESEACHER OBJECTIVE

The research objectives of this study are:

i. To analyse reinforced concrete beam with two circular web opening by using ANSYS + CIVILFEM software.

ii. To determine the maximum deflection of reinforced concrete beam with two circular web opening by probabilistic design system (PDS)

iii. To study the response and behaviour of RC beam with opening through a series of analysis under different load, reinforcement diameter, link diameter, stirrup diameter and link spacing.

1.4 SCOPE OF STUDY

The study is about the finite element analyses on beam with two circular web opening at constant moment and below are the specimen and detail to be analyse by finite element analysis.

- Dimension of beam is 200 x 400 x 3000 mm
- Steel grade 500 N/mm^2
- Bar using T20, T16 and R8
- Shape of web opening is circular and its diameter is 100mm

The scope of this research is to analyse the beam with opening by using finite element software called ANSYS. Besides, it also used to simulate the bending and shear, behaviour, and effect of reinforcement concrete beams with two circular web opening.

1.5 EXPECTED OUTCOME

The expected outcome is to achieve the objective of this research which are able to analyse reinforced concrete beam with two circular web opening by using ANSYS + CIVILFEM software, able to determine the maximum deflection of reinforced concrete beam with two circular web opening by probabilistic design system (PDS) and also able

to study the response and behaviour of RC beam with opening through a series of analysis under different load, reinforcement diameter, link diameter, stirrup diameter and link spacing.

1.6   SIGNIFICANT OF STUDY

It is a need to have a knowledge about the behaviour of the beam with opening under certain conditions. This is because opening in beam will influence the strength of a beam due to the improper design of an opening. Hence, the main purpose of the study is to simulation the behaviour of the beam web opening by using finite element method, ANSYS. This study may also be a reference for a future study or improvement.

1.7   SUMMARY OF CHAPTER

Beam is one of the important structural element for a building because it can withstanding primarily load by resisting bending. Nowadays, the use of web opening on beam structures has become most popular in building constructions because it can help in reducing the height of the building by eliminating the death space. Thus, a proper design for the beam with opening is very importance to have a safety building. Therefore, an analysis had been done to determine the behaviour of the beam with opening so that it is safe for the design by using ANSYS software for this research.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

For the modern building, the design of reinforced concrete beam with 2 circular web opening has been acceptable by the engineering and this type of structure has been design to encourage the designer to reduce the height of the building. The purpose of designing the beam with opening is because it is a facility which allow the utility line to pass through the structure and at the same time the height of the building will be reducing. For the designing RC beam with opening, it may influence the behaviour of the beam such as the strength, stiffness, deformation and cracks pattern. This influence may difference to the normal beam which beam without any web opening. Thus, an analysis of the beam with opening must be carried out to determine the behaviour of the structure and the result will be compare with a normal beam so that it can shows which beam are more suitable in use for a modern structure design. For the analysis part, a nonlinear finite element analysis has been use in these study as a method for predicting the behaviour of reinforced concrete beams with two circular web openings.

2.2 ANSYS AND FINITE ELEMENT ANALYSIS

A software that analyses the structure for these research are unknown as ANSYS and it is a finite element analysis that has been used widely for the engineer to construct a model of structure. However, ANSYS is the most advanced, comprehensive and reputable finite element and design software that based on civil or structural engineering

projects such as to produce simulation code for the analysis and it is a pure structural

simulation of making it a unique and powerful tool for a wide range of civil engineering

of the beam. This is because the corner of opening will subjected to stress concentration

to deform under service load. Therefore, a proper design of beam should be consider so
that there is a safety structure on there. The finite element method has been used to

reduction in the serviceability of the beam (Amiri & Hosseinalibygie, 2004). It should also be noted that deflections of beams containing large openings having the same area are smaller than those of beams containing a few small openings

. Moreover, they also state that the concept of serviceability is the service load that induced to the deflection in beam and the maximum width of the crack must not be exceed to the allowable limit. Other than that, the presence of diagonal reinforcement will controls cracks and increasing the stiffness in beam too. In a short word, the deflection increased is because of the variation in bending and shear stiffness of the beam due to the contribution of web opening. According to Lawson in year 2006, the influence of the variation of bending stiffness dominates the deflection. For more information, there are several detailed calculation or method to predict the deflection, such as the virtual work method (Ward, 1990), the stiffness method (Benitez, 1998), the typical design table method (Fabsec, 2006) and the simple coefficient method (Arcellarmittal, 2008). However, virtual work method and simple coefficient method are not usually use because of the complexity and potentially inaccurate (Chung, 2001). Therefore, the typical design table method and the simple coefficient method are usually uses to predict the deflection (Panedpojaman & Thepehatri, 2013). This prediction is depending on the geometrical variables of beam and this methods are generally derived based on the empirical study of finite element analysis (FEA)

2.3.2 SHEAR BEHAVIOUR

Shear behaviour can be determine when the experiment has been done and it is going through by investigating the cracks of the beam. According to Amiri & Hosseinalibygie experiment in year 2004, the beams with opening before the beam going through the first crack, it will act in a linear state of the graph of load against strain. But after the first shear crack in area of opening, the gradual curving of the load deflection curve is induced and it show a very large strains on there. This happen because the large strain observed is in the state of failure of the beam due to the crack induced in time. Besides, the assumption is make for a diameter of the opening is exceeding 1/3 depth of beam and it will induce the considerable change in ultimate strength of the beam as well

as the shear and flexural behaviour too (Amiri & Hosseinalibygie, 2004). However, in a position where the opening is located must be near to the force apply or in between the force apply and the support, will result the similar behaviour. In their research, they also expecting the diagonal reinforcement installation will influence the effect of the cracking beam which mean proper installing the diagonal reinforcement can helps in controlling the crack as well as reduce the diagonal strain of cracks. Other than that, it is importance to install the small stirrups together with the diagonal reinforcement which located around

and control or prevent the development of shear cracks. For the beam with opening in the

For bending moment and shear force distribution are uniquely defined by static because it is a statically determinate structure. However, the distribution of internal force and moment is in statically indeterminate structure and it depend on the stiffness of each individual member. The opening on beam also play an important role for effecting the stiffness of the structure and it will cause a redistribution of internal action due to the reduction of stiffness. Thus, all of these aspect are need to be consider before a proper design method for the beam opening is done.

2.5　SUMMARY OF CHAPTER

This chapter emphasizes on the analysis of reinforced concrete beams with two

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

In this chapter will discuss the method used for the analysing beam with two web opening. ANSYS software has been used to analyse and to investigate the behaviour of reinforcement concrete beam with web opening. Thus, a basic model in ANSYS software is developed to determine whether it is an acceptable for the parametric study or not. Besides, ANSYS structural offers a comprehensive product solution for structural linear or non-linear and dynamics analysis. A complete set of element behaviour, material models and equation solvers for a wide range of engineering problems. From previous experimental result, the verification study should be done in order to superimpose in current study which the parameter will be changeable. Hence a manual calculation has been done to compare with the ANSYS software to prove that the result can be applied in ANSYS or not.

3.2   FLOW CHART OF METHODOLOGY

```
    ⬭ START ⬭  ──────────▶  ⬦ SCOPE ⬦
        │
        ▼
┌─────────────────────────────────────────────────────────────────┐
│ OBJECTIVES:                                                       │
│                                                                   │
│  i.   To analyse reinforced concrete beam with two circular web   │
│       opening by using ANSYS + CIVILFEM software.                 │
│                                                                   │
│  ii.  To determine the maximum deflection of reinforced concrete  │
│       beam with two circular web opening by probabilistic design  │
│       system (PDS)                                                │
│                                                                   │
│  iii. To study the response and behaviour of RC beam with opening │
│       through a series of analysis under different load,          │
│       reinforcement diameter, link diameter, stirrup diameter     │
│       and link spacing                                            │
└─────────────────────────────────────────────────────────────────┘
        │
        ▼
┌──────────────────────┐
│  COLLECTING DATA      │
└──────────────────────┘
        │
        ▼
┌──────────────────────┐
│ MODELLING AND ANALYSIS│
└──────────────────────┘
        │
        ▼
┌──────────────────────┐
│      RESULTS          │
└──────────────────────┘
        │
        ▼
┌─────────────────────────────────┐      ┌──────────────────────┐
│ DISCUSSION AND CONCLUSION        │─────▶│  PREPARING FOR        │
└─────────────────────────────────┘      │  COMPLETE THESIS      │
                                         │  REPORT               │
                                         └──────────────────────┘
```

3.3   DESIGN PROCESS

ANSYS is a general purpose for finite element modelling package because it can helps in numerically solving variety of mechanical problems. Those problem are including structural analysis, heat transfer, static or dynamic, fluid problems and electromagnetic problems. Basically this design process can divided into three phases which are pre-processing phase, solution phase, and post processing phase.

3.3.1   PRE-PROCESSING PHASE

It is define geometry and boundary conditions and material properties in this stage which is defining the problem and the major step in this phase are shown below.

- Define key points/lines/area/volumes
- Define element type and material/geometric properties
- Mesh lines/areas/volume as required

The amount of detail required will depend on the dimensionality of the analysis such as 1D, 2D, and axisymmetric or even 3D.

3.3.1.1 SELECTION TITLE

For the title selection, select 'File' at utility menu bar and press the change title after the file selection. After that, the title can be edit and the step has been shown in the figure below.
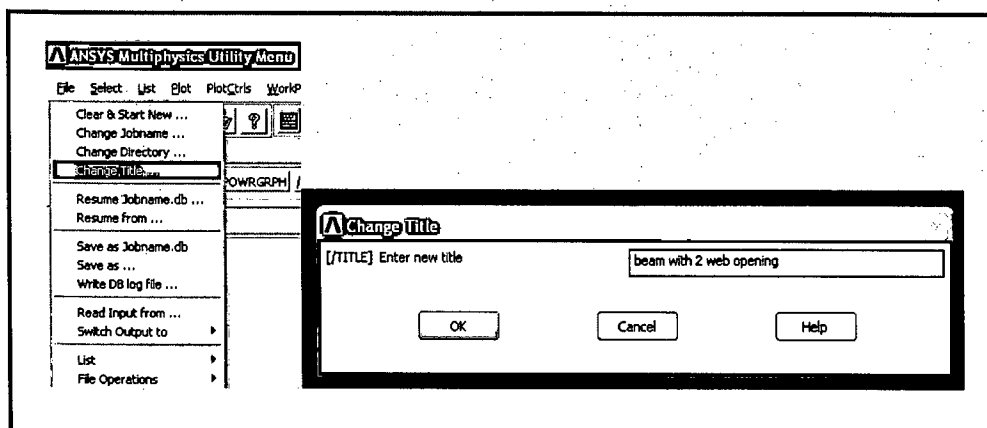

Figure 3.1 : title selection

3.3.1.2 SET UP AND ACTIVATE CIVILFEM

For the activate CivilFEM, select the CivilFEM at ANSYS main menu and ok for the CivilFEM to activate. However, to setting up the CivilFEM, choose the civil setup and the code for reinforced concrete is eurocode 2 and choose the unit by using SI units. Figure below shows the step of setting up and activate CivilFEM.
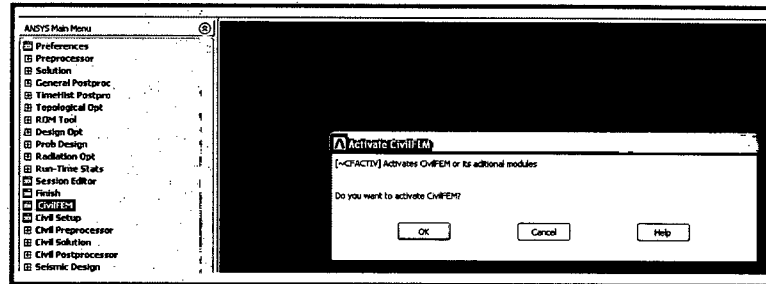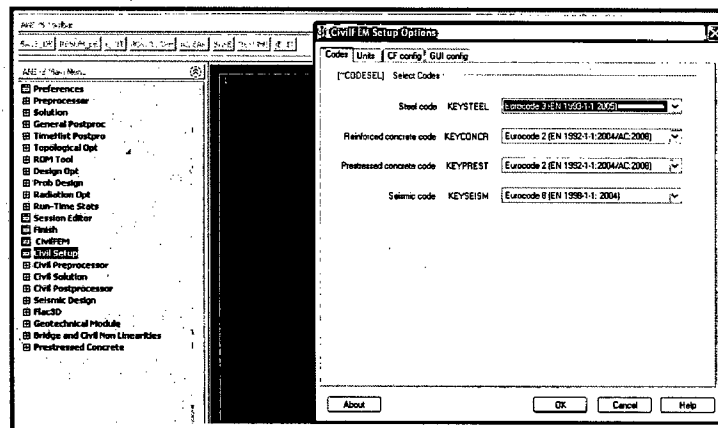

Figure 3.2 : Activate CivilFEM

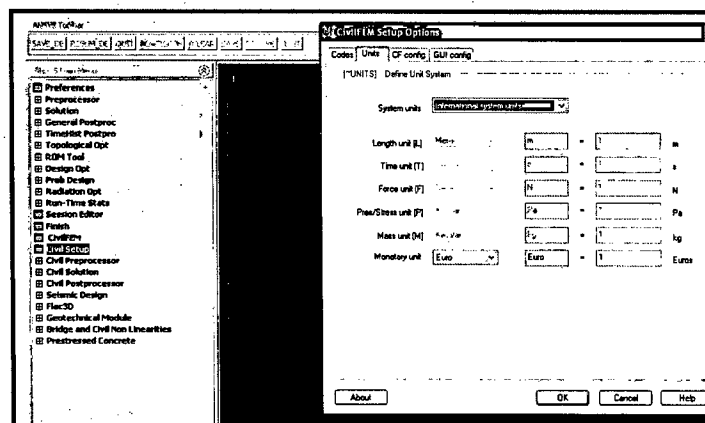**Figure 3.3** : CivilFEM setup for codes

Figure 3.4 : CivilFEM setup for units

3.3.1.3 DEFINE ELEMENT TYPES AND REAL CONSTANTS

There have more than 100 differences element type in ANSYS. Each element types has a unique number and prefix that identifties the element catergory. For this research, the element types that been use for link is link 8 and solid 65 for the modelling concrete. However, it is assumed to be elastic-plastic material and identical in tension and compression. Besides, it is three-dimensional with three degree of freedom at each node.
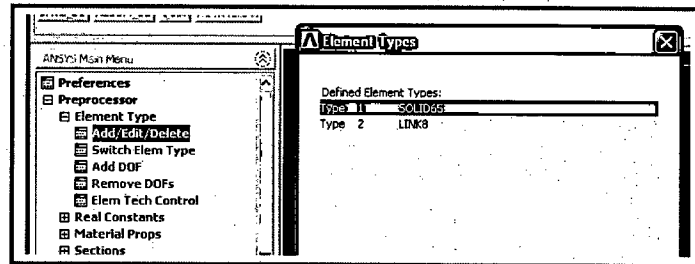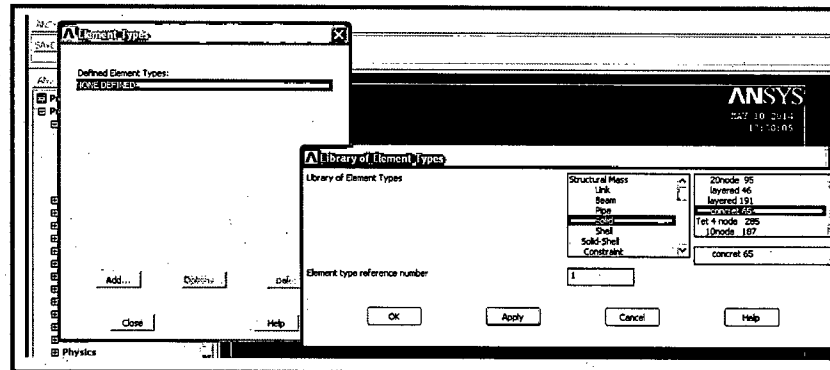
Figure 3.5 : Element types

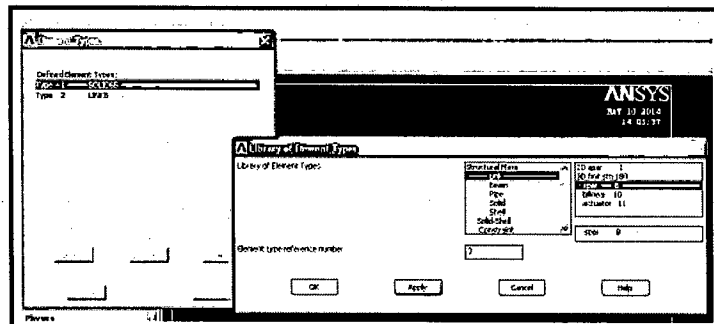**Figure 3.6** : Element type solid concrete 65

Figure 3.7 : Element type link 8

Element real constants are properties that depending on the element type, such as cross-sectional properties of a beam element which mean it is describing the element reference. To define element real constants, first go to main menu follow with