ANALYTICAL SIMULATION OF REINFORCED CONCRETE BEAM WITH OPENINGS USING ANSYS SOFTWARE

ANG YEONG LEANG

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

Reinforced concrete beam with openings have commonly used in the construction of structure due to aesthetic pleasant. So, beam is one of the important components for a structure and needed to be analysed and designed properly so that it can satisfy the strength and serviceability limitation. Although there have many existing commercial engineering software such as STAAD PRO, LUSAS, MATLAB and ANSYS, but finite element analysis is also known as ANSYS is become popular and widely used for solving engineering problems. ANSYS is a finite element modelling for numerically solving wide variety of structural analysis problems such as static structural analysis for both linear and nonlinear. This study is about the modelling of reinforced concrete beam with opening using ANSYS and CivilFEM 12.0 to determine the maximum deflection, displacement, stresses and strains under a set of loads. There are six random inputs will be used in this study such as Young Modulus (EX), Poison Ratio, Density, Area, and Fy Loads. The random output will be the graph of maximum deflection, displacement, stresses and strains. This study is going to be basic guidelines on how to implement the civil engineering element using ANSYS software.

ABSTRAK

Rasuk konkrit bertetulang dengan bukaan telah biasa digunakan dalam pembinaan struktur disebabkan estetik yang menyenangkan. Jadi , rasuk merupakan salah satu komponen penting bagi struktur dan perlu dianalisis dan direka dengan teliti supaya ia dapat memenuhi had kekuatan dan had kebolehkhidmatan. Walaupun terdapat mempunyai pelbagai perisian kejuruteraan komersial yang sedia ada seperti STAAD PRO, LUSAS , MATLAB dan ANSYS , tetapi analisis unsur terhingga juga dikenali sebagai ANSYS telah menjadi popular dan digunakan secara meluas untuk menyelesaikan masalah kejuruteraan. ANSYS adalah model unsur terhingga bagi menyelesaikan secara berangka dalam pelbagai masalah analisis struktur seperti analisis struktur statik untuk linear dan tak linear . Kajian ini adalah tentang pemodelan rasuk konkrit bertetulang dengan pembukaan menggunakan ANSYS dan CivilFEM 12.0 untuk menentukan pesongan maksimum, anjakan, tegasan dan terikan di bawah satu set beban. Terdapat enam input rawak akan digunakan dalam kajian ini iaitu soalan semuanya (EX) , Nisbah Racun , Ketumpatan , Kawasan, dan Beban Fy . Output rawak akan graf pesongan maksimum, anjakan, tegasan dan terikan . Kajian ini akan menjadi garis panduan asas bagaimana untuk melaksanakan elemen kejuruteraan awam menggunakan perisian ANSYS .

TABLE OF CONTENTS

CHAPTER 4 RESULT AND ANALYSIS

CHAPTER 5 CONCLUSION AND RECOMMEDATION

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATION

RC	Reinforced Concrete
FEA	Finite Element Analysis
3D	Three Dimensional
GUI	Graphical User Interface

CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION

Transverse opening in reinforced concrete beam is a facility, which allows for the installation ducts and pipes such as water supply, electrical supply, telephone and networking. Besides, the height of structure can be reduced by the designer with this type of design, which leads to an economical design. Furthermore, beam opening are generally located at shear happening area which is the support, where openings come in a range of sizes and shapes. From the range of sizes in beam opening, circular and rectangular openings are the common ones; for service pipes such as plumbing, circular opening will be used while air conditioning duct will be using rectangular openings.

With regard to the openings size, many researches used the terms "small" and "large" without any declaration or drawing a clear-cut demarcation line. The circular openings may be considered as small when its diameter less than 0.25 time the depth of the beam. (Somes & Corley, 1974). Therefore, behaviour of the beam will become more complex due to presence of transverse opening as the openings represent a source of weakness and failure parts such as deflection and cracking always passes through the openings. The provision of openings at beam will undergo stress distribution and cause the cracking around the opening part of the beam. (Ahmad et al., 2012). Therefore, the

additional reinforcement should be provided around the opening region to prevent the cracking and premature failure of the beam.

Among the various numerical methods, finite element analysis is become popular and widely used for solving engineering problems. Finite element analyses is become an integral part of computer aided engineering. ANSYS is a general purpose finite element modelling for numerically solving wide variety of structural analysis problems which include static structural analysis for both linear and nonlinear. In general, a finite element solution may be broken into three stages which are preprocessing, solution, and post processing.

In this study, reinforced concrete beam with opening is modelled by using ANSYS to analyse the deflection and stress of beam. Finite element method is using in this study in order to get more accurate result for the behaviour of beam. Besides, probabilistic analysis is also doing in multiple simulations to get the results of maximum deflection, stress and strain.

1.2 PROBLEM STATEMENT

In Malaysia, beams such as T-beams, precast beams and deep beams with openings were commonly used in building construction especially in high-rise buildings. Although there had many of studies and researches have been investigated the effect of opening on the T-beams, precast beams and deep beams, but there had limited date regarding on rectangular beams with openings. So, it is significant to introduce the opening in rectangular beam due to limited depth of beams. The behaviour of beam will not be affected if the opening is less than half of the depth of beam. (Saksena & Prof Patel, 2013). Therefore, the beam with opening region will not be able sustained design loading and may be undergo cracking and deflection.

Besides, the fabrication of diagonal bars and stirrup on top and bottom of the opening will increase the beam's ultimate strength. (Vaseghi Amiri & Hosseinalibygie, 2004). The presence of the diagonal bars and stirrup on the opening will control the flexural strains, deflection and cracking on the beam's opening. A research stated that as

increased of the diameter of opening in the beams, the pattern of cracks changed and decreased the strength of beam (Saksena & Prof Patel, 2013). So, web opening at the beam may be seriously affected on the behaviour of beams such as its strength and serviceability. Therefore, beam with openings in concrete design are possible and there are still many factor and properties such as flexural strength, life span, pattern cracking at openings and others should be considered.

Engineers need to know some parameter such as loading adopted, deflection and strength for a structure before designing it. However, they required couple of time to calculate same structure and waste a lot of time. By using ANSYS, the same structure can be designed with different loads cases. This enables the engineers to make their work easier and more efficiency.

1.3 RESEARCH OBJECTIVE

The research objectives of this study are:

i. To determine the deflection and ultimate strength of RC beam with circular web opening.
ii. To establish finite element analysis are capable to predict the behaviour of RC beam with circular openings.
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

Scopes of this study are:

i. Beam model (B × H × L) is 200mm × 400mm × 3000mm
ii. Circular opening diameter is 100mm
iii. Main reinforcement and link will calculated based on Eurocode Standard.

iv. Additional reinforcement will be calculated based on opening size (Mansur,.M.A. & Tan,. K.H. (1999). Concrete Beam with Openings Analysis and Design)

1.5 EXPECTED OUTCOME

A verification study from previous researches will prove that there has a good agreement between FEA and researches results. Form the verification study, it will guide the method and apply the same theory in simulate a new model. The RC beam with opening will be simulated and its results such as deflection and cracking will be determined. Besides, the behaviour of RC beam such as stress-strain, deformation, shear force and bending moment will predicted by using ANSYS software.

1.6 SUMMARY OF CHAPTER

In the conclusion, nowadays there are many researches on simulate beam with opening such as deep beam with opening, steel beam with opening, precast beam with opening and RC beam with opening. ANSYS or FEA is a good design and analysis software in engineering field because it able analysis the behaviour of a structure such as cracking, deformation shape, deflection, ultimate strength, stress-strain, shear and bending moment. The model can be modified immediately and simulate again without waste time and money compared to experimental work. For our research is to determine the behaviour of RC beam with opening by using ANSYS software.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

Beam is classified as a structural element which is sustain the loading such as slab loading and wind loading and transferring the loading to the support which is known as column. Nowadays, there have many type of beam opening are commonly used in the construction field such as deep beam with opening, steel beam with opening, and precast beam with opening. However, beam with opening influenced the behaviour of beam in term of its ultimate strength, shear strength, deflection and cracking. So it must undergo proper design and analysis before it constructed. ANSYS and FEA is a design and analysis tools which able to simulate the behaviour of the models.

2.2 RC BEAM WITH OPENINGS

According to Saksena, .N.H & Patel,.P.G. (2013), RC beams with opening are commonly used in the structure building to reduce the height of structure which leads an economical design and allows the utility line pass through the structure. However, there have various shapes of opening such as circular, rectangular, triangular and others irregular shapes but circular and rectangular are the commonly used (Prentzas 1968). The circular openings may be considered as small when its diameter less than 0.25 time the depth of the beam. (Somes & Corley, 1974). So the size of the opening is depend on

the type of loading of building and dimension of beam. Based on previous researches (Prentzas, 1968; Mansur et. al., 1985,1990; Nasser et. al., 1967), it showed that beam-type behaviour transforms into a Vierendeel truss action as the size of opening is increased. The opening size designed is dealing with various types of loading and load combinations.

2.3 BEHAVIOUR OF RC BEAM WITH OPENINGS

The behaviour of RC beam with openings will be effect with various loading act on beam, location of loading located on beam, and size of opening. Based on the researcher (Vaseghi Amiri & Hosseinalibygie, 2004), different diameter size of opening in beam will cause change pattern of cracks and type of failure. However, number of openings and different distance of opening located will affect the ultimate strength and mode of failure of beam (Cheng et al., 2009). So additional reinforcement provided such as longitudinal bars, diagonal bars and stirrups around openings can limits the cracks and increase beam's ultimate strength (Mansur, 2006). Increased in concrete strength will result maximum crack width around openings decreased and ultimate shear strength of deep beam with openings decreased (Yang et al., 2006). Finite element analysis also used to simulate the behaviour of reinforced concrete beam with large transverse openings subjected to flexure and it showed that the predicted load-deflection curves are in good agreement with the experimental results (Al-Shaarbaf et al., 2007).

2.4 ANSYS AND FINITE ELEMENT ANALYSIS (FEA)

ANSYS is comprehensive finite element analysis and design software for civil or structural engineering projects. It combines the general purpose structural analysis features of ANSYS with high-end civil engineering structural analysis capabilities of CivilFEM and makes it become a unique and formidable tool for a wide range of civil engineering field. ANSYS software is the most dominant method for calculation and simulation of computer made model. It also can be used in various types of simulation such as structural, thermal, fluid mechanics and analysis (Niklasson, 2000). In spite of the powerful of FEA, the disadvantage is the stresses are influenced by important problem variable such as material properties and geometrical features, and wrongs in input data can produce incorrect results (Roylance, 2001).

According to Siemens PLM Software (2010), FEA is an engineering computational analysis methodology that helps to determine the strength of a product in respond to loading that might typically experience in its operating environment. It can also help to know the reason why the parts failed. Besides, FEA simulates the behaviour of a model which including the physical conditions in which it operates. FEA solver analyse the model and determine the results that reflect the design behaviour to the applied boundary condition, and can help to identify weaknesses or potential failures of the model.

However, Kurowski,.P.M. (2004) said that FEA is a design tool which change the design process from iterative cycles of "design, prototype, test" into a streamlined process where prototype only used in final design verification. With the use of FEA, design iteration are transformed from physical space of prototyping and testing to virtual space of computer-based simulation. It also can be used for computational fluid dynamics and motion analysis which known as tolls of computer-aided engineering (CAE).

FEA is applied in this research and RC beam with opening will be modelled using ANSYS software. Three dimensional modelling will be done in this research to produce more accurate results.

CHAPTER 3

RESEARCH METHODOLOGY

3.1 INTRODUCTION

This chapter will discuss the method used to analyse and model RC beam with opening. Finite element analysis method will be applied in this research to analyse the behaviour of RC beam with opening. FEA method is used to prove that it is suitable to complement the experiment on RC beam with opening and it is less time consuming. ANSYS software is used to analyse and simulate the behaviour of RC beam with opening. In ANSYS, the analysis can be categories into three stages which are preprocessing, solution, and post processing. Besides, probabilistic method is doing in order to get maximum deflection, maximum stress, and maximum strain value in samples values graphs, histogram plots graph, cumulative graph, and pie chart.

Verification between ANSYS results and manual calculation is important as this will served as a platform to superimpose that the current study can apply the same basic theory and method with different condition to fulfil the research objectives and match with the research title. Therefore, 3D model of RC beam with opening results will be compared with the manual calculation to prove that the results can be applied in the engineering field.

3.2 ANSYS MODELLER SOFTWARE

Finite element analysis is applicable to matrix analysis because the analysis is fast, accurate and its suitability for computer application. Due to large displacement that caused by load, finite element is able to take account on the geometrical non-linear factors such as large deflection and stress stiffening. ANSYS is used by the product designers and engineers around the world. It is also one of the world's leading developers of engineering simulation software. This modern technology has advantages like quick, efficient and cost effective product. However, the analysis in ANSYS consist three stages which are pre-processing, solution and post processing. The models can be creating using command prompt line input or the Graphical User Interface (GUI). The GUI was utilized to create the model.

The last stage of the process which is post processing considers a very important part of analysis because it will achieve the objectives if run smoothly and succeed. So it must be handling carefully and concentrated while interpreting the results. The outcome from the finite element method analysis is fully dependent on the input in the programme. Hence, the input must be very carefully processed before key in into computer and make an equilibrium check before the results from computer can be accepted.

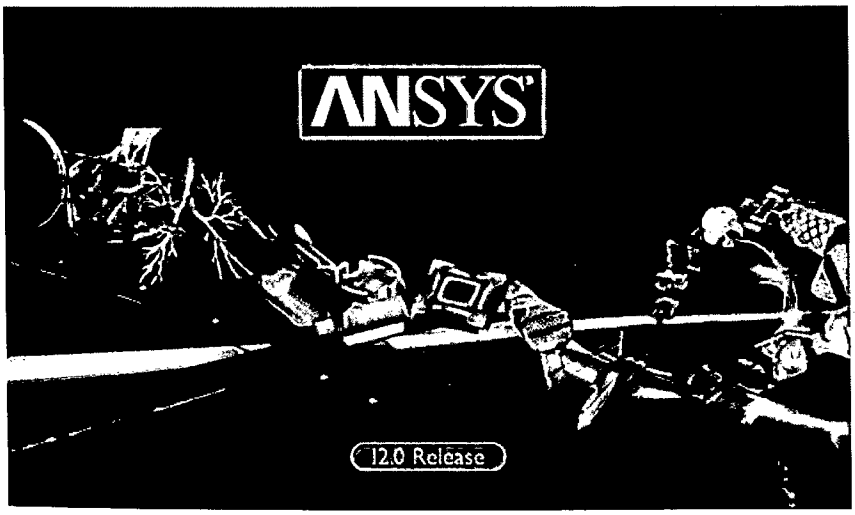

Figure 3 1: ANSYS Modeller

3.2.1 PRE-PROCESSING

The first stage is pre-processing which is defining the problem. The pre-processing include element type, real constant, material properties, modelling, and meshing,

3.2.1.1 ELEMENT TYPES

The element type for this model is solid 65 which is a three-dimensional eight noded solid isoparametric element was used to model the concrete. This element has eight nodes with three degrees of freedom at each node- translations in the nodal x, y, and z directions and also capable of plastic deformation, cracking and crushing.

From the pre-processor menu, select: Element type > Add/ Edit/ Delete. Click on the 'Add' button in the Element Type window. Select 'Solid' in the left hand box and 'concret 65' in the right at Library of Element Type. Click 'OK' after select the element and close the window.

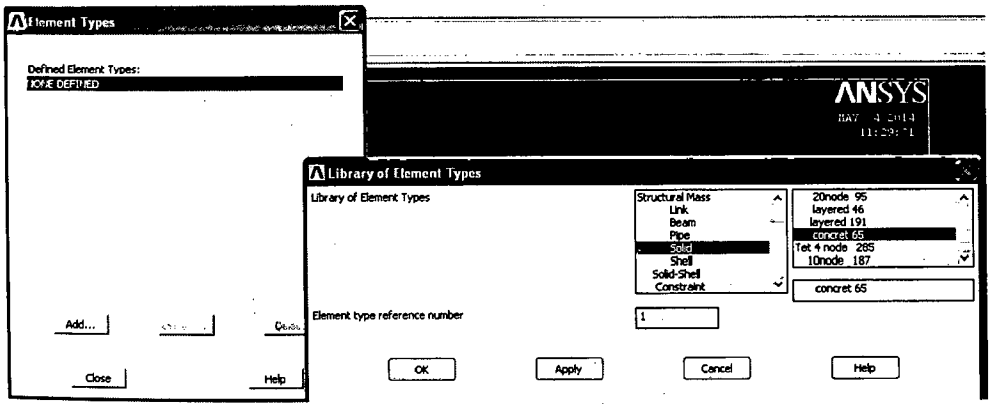

Figure 3.2: Element Type of Model

3.2.1.2 Material Properties

In order to generate the model, there have some parameter and material needed to identify. In this study, the concrete grade of the model is grade 25 while the reinforcement is S500 which is steel grade 500 N/mm^2 that select through Civil Preprocessor > Materials. However, properties of each material can be adjusted such as elastic modulus, poisson's ratio, density and others in material model.

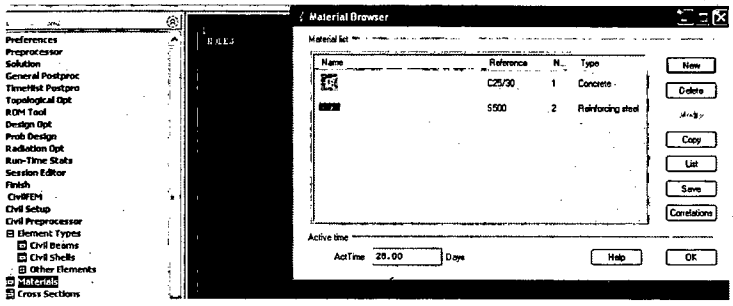

Figure 3.3: Material Brower in ANSYS

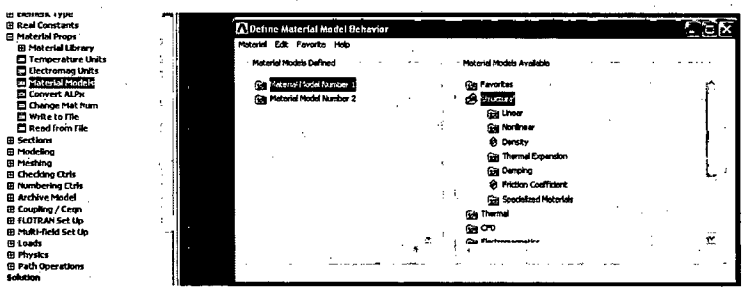

Figure 3.4: Materials Model in ANSYS

3.2.1.3 MODELLING

The overall geometry is defined in ANSYS using keypoints which specify various principal coordinates to define the body. The overall geometry is defined in ANSYS using keypoints which specify varous principal coordinates to define the body. From the ANSYS main menu, select Preprocessor > Modeling > Create > Keypoints > In Active CS. Key in keypoint number and X, Y, Z location in active CS properly.

Same steps are repeated for the remaining keypoints. Table 3.1 below shows the keypoint of the model.

Table 3.1: Keypoint of Model

Keypoints	X	Y	Z
1	0	0	0
2	0.1	0	0
2	0	0.4	0
4	0.1	0.4	0

After create the keypoint, connect the kypoint by using lines. In the main meu select Preprocessor > Modeling > Create > Lines > Lines > Straight Lines. Pick keypoint 1 and then move toward keypoint 2, the line will be form and repeat same method to form others lines. After all the lines are connecting, mirror the area in x axis. In main menu, select Preprocessor > Modeling > Reflect > Area > OK > Select Plane of Symmetry in yz plane > OK as shown in figure. After that, model the line that connect to become area through Proprecessor > Modeling > Operate > Areas > Select the both parts > OK as shown in figure 3.5.

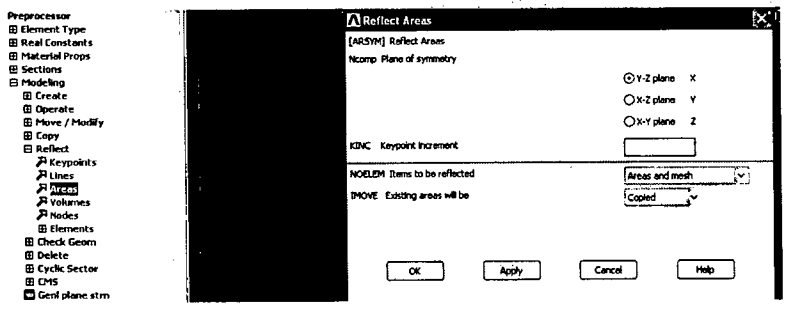

Figure 3.5: Area Reflect in YZ Plane

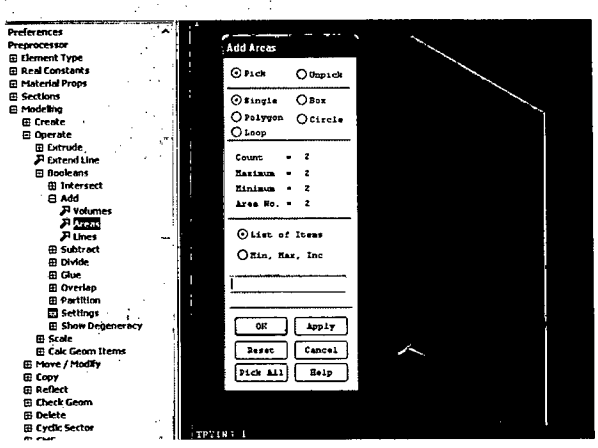

Figure 3.6: Model Line to Area

After mirror the area, a solid beam will be form by offset the area along the z axis. In Preprocessor > Modeling > Operate > Extrude > Areas > Along Normal > Length of extrusion. Key in 3 in length of extrusion which is mean the beam is 3 metre length.

Subtracting of Boolean operator will be used to create the holes of the beam. The model is 200mm width × 400mm height × 3000mm long while the opening of the beam is 100mm diameter. The dimensions of solid cylinder volume are shown in table 3.2. The beam with opening model is shown in figure 3.7.

Table 3.2: Dimension of Solid Cylinder Volumes (Opening)

ANSYS	Left Solid Cylinder, m	Right Solid Cylinder, m
WP X	0.6	2.4
WP Y	0.2	0.2
Radius	0.05	0.05
Depth	0.2	0.2