LINEA[illegible] EL ROOF TRUSS

NURULFATIHA BINTI ISMAIL

A report submitted in partial fulfillment of the

Requirement for the award of the degree of

Bachelor of Civil Engineering

Faculty of Civil Engineering and Earth Resources

UNIVERSITI MALAYSIA PAHANG

JUNE 2013

ABSTRACT

This thesis analyzed linear static analysis of a steel roof truss. When loads are applied to a body, the body will deformation and the effect of loads is transferred throughout the body. The external loads induce internal forces and reactions to turn the body into a state of equilibrium. This analysis is determined the stresses and displacements parameter of a roof truss. It also to determine the tensile, compressive and compression buckling in the critical section of a steel roof truss. Therefore, propose the best design of a steel roof truss. This thesis describes the finite element analysis consists of a model of a material or design that stressed and analyzed for specific results. It is used in new design products, and the existing of product refinement. In industry, there are generally had two types of analysis that are used; two dimensional modeling and three dimensional modeling. Usually two dimensional modeling conserves its simplicity and support the analysis to be executed on a relatively normal computer; it tends to produce less accurate results. The finite element model of steel roof truss a linear static was be analyzed. There are 6 model with different steel section is considered in order to get the most optimum result for the construction of a roof truss. All the steel section in equal L section and have the same yield strength which is 275N/mm^2. From the results, it is observed that L angle section 150×150×14 is adequate to applying on a roof truss. The roof truss can be considered as safe since the only design criteria that required in designing is tensile, compression and buckling. Furthermore, if all the steel section can pass all this three forces, the roof truss is considered as safe to be used. All the steel section is in a normal yield strength which is 275N/mm^2.

ABSTRAK

Tesis ini menganalisa linear statik kekuda bumbung keluli. Apabila beban dikenakan kepada struktur, struktur akan berubah dan kesan beban dipindahkan ke seluruh struktur komponen. Beban luar menyebabkan daya dalaman dan tindak balas untuk menjadikan sturktur komponen dalam keadaan seimbang. Analisis ini ditentukan oleh tekanan dan anjakan parameter kekuda bumbung. Ia juga untuk menentukan tegangan, mampatan dan pemampatan lengkokan di bahagian kritikal kekuda bumbung keluli. Tesis ini menerangkan analisis unsur terhingga terdiri daripada bahan model atau reka bentuk yang menekankan untuk mendapatkan hasil tertentu. Ia digunakan dalam produk reka bentuk baru, dan yang sedia ada. Dalam industri, umumnya mempunyai dua jenis analisis yang digunakan, dua model dimensi dan tiga model dimensi. Biasanya dua model dimensi memelihara kesederhanaan dan menyokong analisis yang akan dilaksanakan pada komputer yang agak biasa, ia cenderung untuk menghasilkan keputusan yang kurang tepat. Model unsur terhingga keluli kekuda bumbung statik linear telah dianalisis. Terdapat 6 model seksyen keluli yang berbeza dianggap untuk mendapatkan hasil yang paling optimum untuk pembinaan kekuda bumbung. Semua bahagian keluli dalam seksyen L sama dan mempunyai kekuatan alah yang sama iaitu 275N/mm2. Daripada hasil kajian, didapati bahawa L sudut seksyen 150×150×15 adalah mencukupi dalam menggunakan kekuda bumbung. Kekuda bumbung boleh dianggap sebagai selamat kerana satu-satunya kriteria reka bentuk yang memerlukan dalam bentuk adalah tegangan, mampatan dan lengkokan. Tambahan pula, jika semua bahagian keluli boleh melepasi ketiga-tiga faktor, kekuda bumbung dianggap sebagai selamat untuk digunakan. Semua seksyen keluli adalah dalam kekuatan alah biasa yang 275N/mm^2.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

A	Cross-sectional area
A_{eff}	Effective area of a cross-section
A_{net}	Net area of a cross section
N_{Ed}	Design value of the tensile force
N_{cr}	Elastic critical force for the relevant buckling mode based on the gross sectional properties
$N_{b,\ Rd}$	Design buckling resistance of the compression member
$N_{c,\ Rd}$	Design compression resistance of the cross-section
$N_{net,\ Rd}$	Design plastic resistance of the net cross-section
$N_{pl,\ Rd}$	Design plastic resistance of the gross cross-section
$N_{t,\ Rd}$	Design values of resistance to tension forces
$N_{u,\ Rd}$	Design ultimate resistance
fy	Yield strength
f_u	Ultimate tensile strength
γ_{Mo}	Partial factors for resistance of cross-sections
γ_{M2}	Partial factors for resistance of cross-sections in tension to fracture
λ	Slenderness
L_{cr}	Buckling length (Effective length)
α	Imperfection factor
χ	Reduction factor
φ	Phi
i	Radius of gyration about the relevant axis
σ	Stress
F	Force
ϵ	Strain

CHAPTER 1

INTRODUCTION

1.1 Background of Study

Trusses are one of the most common structures utilized in construction building. A truss is a structure composed of a slender member connected together and consist of straight members. The two and three-dimensional trusses are structures composed of simple one dimensional component that are fitted together which can transfer only axial forces along their axis. All trusses are usually used for long span roofs, bridges, transmission tower and so on. It is typically composed of triangular shape or elements with the bars on the top chord under compression while for the bottom chord, it consists of tension. It is depending on the diagonals of orientation, they can be turned either as compression or tension.

In a roof structure, all trusses are made for the feasible condition from combining live, dead and as well as wind load. The individual truss members are made to restrain the appropriate forces either tension, compression, or combining bending with the both tension and compression forces. Tension is also called as pulling; the member is being pulled to subject to the tension forces are said as in tension. The capability of members to restrain tension forces will depends on its cross-sectional area and also the materials strength of the new member. Conversely, compression is called as pushing, the forces sometimes referred to as a column. The capability of column to restrain compression forces from the combination of the column length, cross-sectional shape of the column and the material strength.

Some sort of triangle shape is the easiest geometric figure that won't transform shape when the measures of the features are usually predetermined. The actual angles and also lengths of a four-sided geometric figure must be predetermined for it in order to retain its shape along with design. Because of the shape of triangles, there are three main reasons to form of trusses. Firstly, the truss has unique geometric properties. This is because of triangle shape that consists of three sides and three angles, which are every angle is held solidly in place by the side opposite it. This means that an angle of triangles is fixed, if the pressure is placed anywhere in a triangle, its angles will not change. Secondly, method of transferring loads. In this method, when a force is applied to a triangle, the pressure that occurs is directed to sideways rather than down. That means the sides of the triangle are in either tension or compression, and that there is no bending movement occurs. Thirdly, the spatial openness; since the central of a triangle does not contribute to its geometric rigidity (it is relative to the stiffness of a material that allows it to resist bending, twisting or deformation under a load), the central of a triangle can remain open. The triangles are an appropriate shape to use as one of the goals in building a truss is to minimizing its weight. There are a large variety of roof trusses in used.

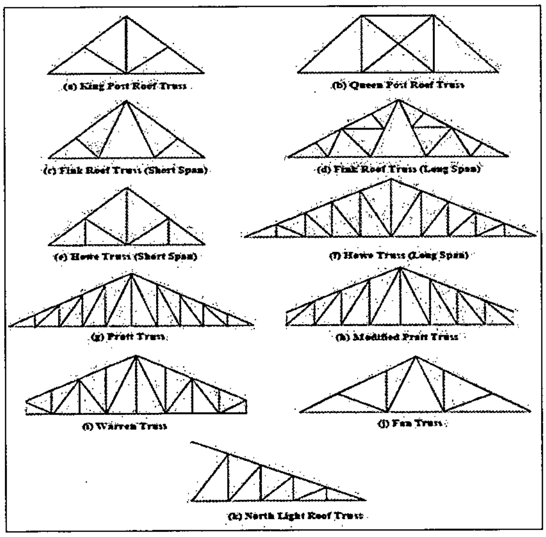

Figure 1.1: Types of a Roof Trusses

In Figure 1.1 shows some of the most common types of a roof trusses. (a) the king-post and (b) and queen-post trusses are some of the oldest forms of roof trusses and were largely used for small span timber roof trusses construction. The Fink (c) and (d), and Howe (e) and (f) trusses are quite suitable for steel construction, both for large and smaller spans. The Pratt truss (g) and (h), Warren truss (i) and Fan truss (j), are also quite common types of roof trusses. Figure 2.1 (k) shows a north light roof truss (unsymmetrical), which is normally used in factories.

Finite Element Analysis (FEA) is for structural stress analysis; it is necessary to determine the stress and strains (deformation) through entire design which can be in equilibrium and it is subjected to applied loads. Commonly, the finite element method consists of modeling from the structure using small units plus a displacement function will be associated with every single of finite element. ANSYS is often a standard purpose of Finite Element Analysis (FEA) computer software. This kind of software utilizes the particular equations that govern the particular actions of these elements and fix all making a detailed justification associated with the way the program functions in general. These results might be presented in table or maybe graphs. This analysis is often used by the design and also optimization of a system too complex to analyze by manually. Systems that will fit into this class are usually too complex due to the geometry, scale, or perhaps regulating equations.

1.2 Problem Statement

In Malaysia, there are lots of issues related to the actual structural analysis. The inappropriate design of the structural elements is probably the circumstances cause the actual structure fall short as well as collapse. The actual designs that do not have a quality are one of aspects in which lead to the structure unstable and also dangerous for consumers. On the other hand, engineers tend to be usually focused on the structure with significant components of the particular building structures compare towards the truss component designs. Many of them simply ignoring the truss design and change it with the design standard which is commonly used within construction. However, sometimes utilized the design standards can't fulfill the specifications at the time period. Being an engineer, it is important to allow them to analyze and run the design regarding truss which is more suitable.

In some instances, the engineer may develop inaccurate results because they do not have enough information about the associated software program. Incorrect data from the analysis may lead unstable trusses and are unable to perform properly due to the lack of strength. Thus, engineers need to find out the actual displacement, stresses, axial forces, as well as deflection towards construction just before designing it. But, they need to calculate a similar structure for a long time and it would spend a lot of time to do so. Therefore, by using ANSYS, engineers may design the same structure with using different loadings. This may help the engineers to generate their works simpler because they do not have repeating the same calculation if they possess some makes mistakes.

1.3 Objectives

The objectives in this study are:

1. To determine the stress and displacement parameter of a roof truss.
2. To determine the tensile, compressive and compression buckling in the critical section of a steel roof truss.
3. Propose the best design of a roof truss.

1.4 Scope of Study

The analysis will be carried out linearly using ANSYS software meanwhile manual calculation will be calculated according to Finite Element Method. ANSYS was used to design tension force, compression force and compression buckling force while finite element method used to calculated stress and displacement parameter in each member of a roof truss. The loading that consider in this analysis is dead load and live load. Dead load is a permanent force, acting on a structure. Normally, weight of roof cover is $0.12kN/m^2$ for steel roofing. In this analysis, wind load does not consider because of using steel roofing; heavy weight so the wind load is not significant to be considered. The self weight of the truss depends on the type of roof covering material and its weight, the span and the rise of the truss and truss spacing. Moreover, live load is a loading which is not of a permanent nature such as snow, wind, movable concentrated loads, and furniture. A live load is a change, or non-permanent force acting on a structure. This includes the force of the wind and the weight of things that are in or on a structure.

Steel section will be checking accordance with Eurocode3 (Design of Steel Structures). Steel has several key features that influenced the nature of a comprehensive standard for structural design. Firstly, the versatility of steel requires that a large number of different types of structure have to be covered. Secondly, the ductility of steel requires the inclusion of both elastic and plastic design and thirdly, slender components that commonly occur and require the inclusion of comprehensive approaches to the stability of structures and members. Therefore, only three types of forces will be considered in the steel section which is tensile, compressive and compression buckling. There are six model with different type of steel section is considered in order to get the most optimum result for the construction of a steel roof truss.

There are three phases throughout ANSYS software; preprocessing, solution and postprocessor. Preprocessing is the first phase in order to identifying the problem. In this particular procedure, the model will probably be created first and it includes several numbers of steps and generally in the following order; constructs the geometry which consists of creating the lines, area or even the volumes. The nodes as well as elements, however, can be created from the coordinates. After that, define materials and produce the element mesh. The second phase is a solution. Applying the particular loads and boundary conditions are generally used on nodes or even elements. Then, obtain the solution if the whole difficulty has been defined. The third phase is postprocessor. This part of the analysis is to view the results after start on preprocessing and solution phase. In order to visualize the results by the following procedure; one example is, plot the deformed shape geometry or stresses. After that, list the results. The reaction forces, deformation, deflection and axial stress will probably view in this phase.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

Ezeagu and Nwokoye (2009) defined truss being a structure which gives a stable type capable of supporting significant external load more than a large span with the elements stressed mainly inside axial tension or compression. A truss is therefore a triangular arrangement of straight components ;bottom and top chords, and webs which are formed from wood, metal or any other structural materials deriving its strength from the triangulation of its components, to transmit loads through the components to the supports. The truss members are generally assumed to be joined together in order to transfer the axial forces, shear and moment from one member to the adjacent members (they are considered to be a pinned joint). The loads allowed to be act only at the nodes on the trusses. The particular trusses can be provided on a simply supported on the two end supports plus a single span, in that case they're usually statically determinate. The analysis regarding trusses might be manually carried out by the method of sections or the method of joints.

In the analysis according to pinned joint assumption, one obtains simply the axial forces within the different members of the trusses. Nevertheless, in the actual design, the particular truss members joined up together by several bolt or welding, either directly or through the end of the gusset larger size. Moreover, several members, specifically members of the cord, may porsibbly remain over numerous nodes. Usually this kind of joints parts enforce not only compatibility associated with rotation of members getting together at the joint. Because of this, the members of the trusses experience bending moment a long with axial force. This isn't negligible, especially at the eaves points of pitched roof trusses, in which the depth is small and the also the trusses having members with a smaller slenderness ratio. Analysis of trusses for the following moment and the secondary pressure can be carried out simply by analysis of secondary indeterminate structure, commonly using ANSYS or maybe other software applications.

2.2 Finite Element Method

The Finite Element Method (FEM) is based on the idea of complex objects with simple blocks, or even, separating a complex item into smaller pieces. The finite element method is a numerical technique for solving models within differential form. For the given design, it requires the overall geometry, such as the surrounding areas, which are modeled along with finite elements. "The finite element method would be the most effective numerical techniques actually created for resolving differential (and integral) equation and also boundary-value difficulties within geometrically areas." (Reddy, 1988).

This method provides approximate ways of differential equation that design the problems. It requires a problem identified in geometrical area to become subdivided into a smaller regions. The start of finite element had been due to the frustration in the attempt to utilize a various method on harder geometrical unpredictable problems. The first use of finite elements put in the use of such techniques for structurally similar problems. (Pepper, D. W and Heinrich J. C, 2006). The fundamental concept of finite element method can be dealing with the procession by a discrete design. It's carried out by simply dividing the region associated with significance to areas that have linked sub parts or elements. Every element will probably be in simpler geometry and also easier to be end up being analyzed. A new acknowledged physical law is usually then applied to each element in addition to global equation is actually formed which in turn can be used to solve the particular variables. Through splitting up the particular into a large number of small parts of elements along with utilizing suitable compatibility and equilibrium equations to assemble these elements, it is possible to get a nearly accurate precise value of variables such as stress and displacement of body. Therefore, the smaller the elements are divided, the more accurate solution but a cost of increased computation time.

2.3 History of Finite Element Analysis

In 1943, Finite Element Analysis (FEA) was created by R. Courant, who's applied the particular Ritz method of numerical evaluation in addition to minimization of variation calculus to have approximate methods to vibration systems. And then, the item changed with 1956 using the presentation made by M. J. Turner, R. W. Clough, H. C. Martin and L. J. Topp. Since the Finite Element Analysis observed improvements and after this the supercomputers provide results accurate for all kinds of parameters.

In the early 1960s, engineers used the strategy for approximate the problem solving throughout fluid flow, heat transfer, and stress analysis. The primary book on Finite Element Method produced by Zienkiewicz and Chung has been published throughout 1967. In the late 1960s and early 1970s, the Finite Element Method has been useful for several times for engineering problems. Most of commercial FEM software programs originated in the 1970s. (Abaqus, Adina, Ansys, etc.) Through the early 1970s, Finite Element Analysis has been limited to high price mainframe personal computers usually had with the automotive, aeronautics, defense, and nuclear industries. Since the rapid downturn in the cost of computers and the remarkable increase in computing power, Finite Element Analysis has been developed to an incredible precision. The particular develop regarding computer programs through the period 1957 to 1970 at Berkeley were freely distributed worldwide allowing training engineers to solve a few problems with structural mechanics. (Ray W. Clough & Edward L. Wilson, 1999).

2.4 Linear Analysis

In this study, linear analysis (first-order analysis) is also known as linear elastic analysis. The term of elastic means that when the structure is unloaded it follows the same deformation path as when loaded. Meanwhile, linear is imply that load and displacement vary proportionally. As result, the relationship between the loads versus displacements for a linear analysis is always in a straight line shown in Figure 2.1. (MacLeod, 2005)

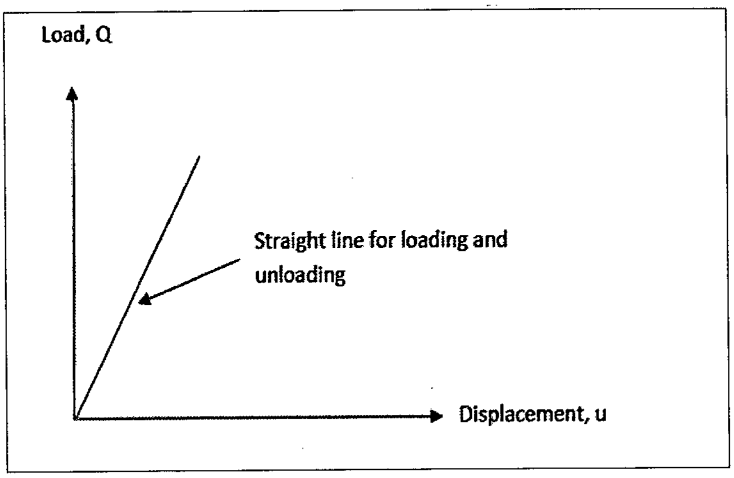

Figure 2.1: Load-Displacement Relationship from Linear Elastic Analysis (MacLeod, 2005)

Basically in linear analysis of a tall truss design, the particular react to various load techniques can obtain through superposition principle. Some sort of classification involving linear static analysis refers to static difficulties when it reacts will be linear in the cause and as well as affect sense for instances when the applied forces tend to be doubled, the displacements as well as internal stresses also double. In the paper of Direct Stiffness Analysis of Lateral Buckling produced by Vacharajittiphan and Trahair (1974) stated that in linear analysis the direct stiffness method is one of the popular and common technique. It is also known as displacement method. This method has been widely used or applied in structural analysis due to it provides a good and systematic algorithm of computation. It is much easier to formulate the necessary matrices for the computer operations by using matrix form if compare to the other method such as flexibility method.

The relationship between the actions and deformations at the ends of a member are obtained directly from the governing differential equation can be solved exactly on the direct stiffness method, as in the first-order or second-order elastic structure analysis. Most important, the direct stiffness method can be used to analyze both statically determinate and indeterminate truss structure in the same manner.

2.5 Linear versus Non-Linear Finite Element Analysis

Linear finite element analysis is usually carried out if a structure is usually likely to behave linearly; for example, obeys the particular Hooke's Law (linear relationship among stress and strain). The linearity prediction when the displacements usually are smaller enough to be able to ignore the change in stiffness due to loading. The boundary conditions do not change during the effective use of loading. Loads should be constant within direction, magnitude, and also distribution. The structure can be load bearing member and can normally end up being categorized as beam, column, shaft or bar. The stress can be proportional towards the strain, and the structure will return to its original configuration once the load has been removed. On the other hand, the non-linear finite element analysis is used to predict the behavior of a structure that is loaded beyond the elastic limit of the material. The actual structure experiences plastic deformation and definitely not return to their original shape.

2.6 Stability of trusses with linear side-supports

This research is to study of a lateral buckling of truss with linear elastic side-supports. It presents to calculate the elastic support reaction in relation to force in compressed chord and coefficient of buckling length that is related to side-support distance. This study is to consider the effect of slope of side-support on limit force. The nonlinear analysis of two-trusses and design sensitivity analysis limit load due to side-support stiffness were carried out in this research. It also wants to calculate a sensitivity of limit load due to side-support localization. The main purpose of this research is to perform non-linear static analysis of two trusses with elastic side-supports. For the differences stiffness of elastic side supports the limit load of the truss, the support reaction and coefficient of out of plane buckling length of the truss chords is calculated.

From this research, the buckling length of the truss compressed chord is greater than side-support spacing. It has shown that the relation between side-support reaction and normal forces in compressed chord is non-linear. Due to introducing a side-support of unit stiffness, it may helpful to find the influence line of the variation of the limit load of the truss. In other cases, additional side-supports may cause a decrease of limit load of the truss. Therefore, the design of these stiffeners should be supported by non-linear analysis or sensitivity analysis. The numerical results of non-linear elastic analysis of trusses with elastic side supports have shown that the buckling length of the truss compressed chord is greater than side supports spacing. (P. Iwicki, 2007).

2.7 Static and dynamic post-buckling behavior of truss structures

In most dynamic analyses of steel frame-structures, mass are assigned only at the ends of members. Such vibration models have been adopted not only for a moment-resistant structure, but also for truss structures whose members are likely to buckle. This paper proposes the structures analyzed are a plane parallel chord truss structure and a double-layer space truss structure. The load deformation relationship was obtained by static and dynamic analyses are compared. In this paper, it has found that the dynamic load-displacement relationship oscillates around the static relationship after buckling. The buckling mode and the post-buckling behavior can be remarkably different between the static and dynamic analysis. This difference may lead to a large difference in post-buckling strength, although it has no effect on buckling strength. (M. Tada, A.Suito, 1998)

2.8 Tension roofs and bridges

A tension structure is typically used for those structures in which one or more load main bearing elements in tension. This structure includes roofing membranes, roofing cable, cable bridges, guyed mast or tower, cooling towers, although some are less common. The tension elements consists of ropes or membranes, strands, which have to combine with structural elements that carrying flexural, compression or a combination thereof, such as, column, arches, towers, girder, beam and truss. In order to achieve adequately stiff and stable tension roof structures from the cable or membrane elements, there are the following possibilities; use gravity stiffness, introduction of pretension, introduction of air pressure support and combination with rigid elements. An alternative concept is to design 'pretensioned' or 'counterstressed' tension as shown as in Figure 2.2. These are both planar or three dimensional, and form the basis of a roof system combined with a membrane or cladding of varying type. (P.Krishna, 2001)

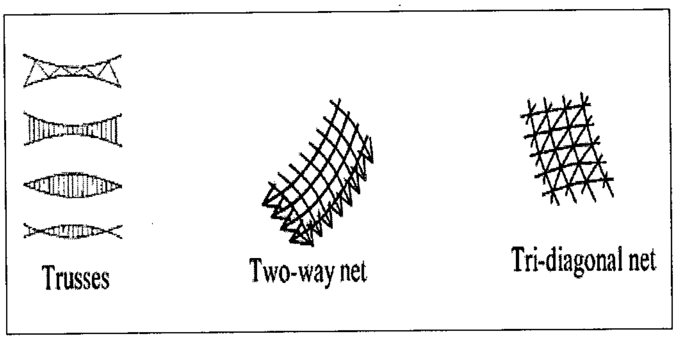

Figure 2.2: Typical tension roof structures

CHAPTER 3

METHODOLOGY

3.1 Introduction

In this particular research, ANSYS used to resolve two-dimensional truss structures based on the Finite Element Method. This kind of method used to analyzes truss structures such as deformation, load or even stress distribution that's applicable to roof supports, bridges, and pylons. It is also used in line programming and solving linear systems. ANSYS is general purpose software, used to simulate the interactions of all disciplines of structural, vibration and heat transfer for Engineers. So it enables to simulate test in a virtual environment before manufacturing prototypes of products. Therefore, ANSYS can carry out advanced engineering quickly, safely and practically by its variety of contact algorithms; time based loading features and nonlinear material models.

The usage of the Finite Element Method as a tool to solve engineering problems commercially in industrial applications is quite new. It was used in the late 1950's and early 60's, but not in the same way as it is today. The calculations were at that time carried out by hand and the method was force based, not displacement based as we use it today. In the mid 60's, very specialized computer programs were used to perform the analysis. The 1970's was the time when commercial programs started to emerge. At first, FEM was limited to expensive mainframe computers owned by the aeronautics, automotive, defense and nuclear industries. However, in the late 70's more companies started to use the FEM,