Applied Element Method - Application to Linear Static Analysis

BY

Vikas B. Gohel

10MCLC03

DEPARTMENT OF CIVIL ENGINEERING

AHMEDABAD-382481

May 2012

Applied Element Method - Application to Linear Static Analysis

Major Project

Submitted in partial fulfillment of the requirements

For the degree of

Master of Technology in Civil Engineering

(Computer Aided Structural Analysis and Design)

By

Vikas B. Gohel

10MCLC03

DEPARTMENT OF CIVIL ENGINEERING

AHMEDABAD-382481

May 2012

Declaration

This is to certify that

a. The thesis comprises my original work towards the Degree of Master of Technology in Civil Engineering (Computer Aided Structural Analysis and Design) at Nirma University and has not been submitted elsewhere for a degree.

b. Due acknowledgement has been made in the text to all other material used.

Vikas B. Gohel

Certificate

This is to certify that the Major Project entitled "APPLIED ELEMENT METHOD - APPLICATION TO LINEAR STATIC ANALYSIS" submitted by Mr. Vikas B. Gohel (10MCLC03), Towards the partial fulfillment of the requirement for the degree of Master of Technology in civil Engineering (Computer Aided Structural Analysis and Design) of Nirma University, Ahmedabad, is the record of work carried out by him under my supervision and guidance. In my opinion, the submitted work has reached a level required for being accepted for examination. The results embodied in this major project, to the best of my knowledge, haven't been submitted to any other university or institution for award of any degree or diploma.

Dr. P. V. Patel	Dr. P. H. Shah
Guide and Professor,	Professor and Head,
Department of Civil Engineering,	Department of Civil Engineering,
Institute of Technology,	Institute of Technology,
Nirma University, Ahmedabad.	Nirma University, Ahmedabad.

Dr K Kotecha	____________________
Director,	Examiner
Institute of Technology,	
Nirma University,	____________________
Ahmedabad.	Date of Examination

Abstract

The Finite element method and other numerical methods are very effectively implemented in static as well as dynamic, linear and nonlinear analysis of structure. Recently, a new method has been developed called Applied Element Method (AEM). This method is more applicable for failure modelling of structure. Applied element method is used for modelling of structure subjected to large displacement and when there is possibility of separation of structural elements. FEM is helpful in predicting failure of structure but can not model separation of elements. The main advantage of applied element method is that it can track the structural collapse behaviour passing through all stages of the application of loads, like elastic stage, crack initiation and propagation in tension-weak materials, reinforcement yielding, element separation, element collision (contact), and collision with the ground and with adjacent structures.

The objective of present study is to understand application of AEM for linear static analysis of structure. In applied element method structure is assumed to be divided into number of rigid elements connected by springs. Springs represent force-displacement relationship of structural elements. So, derivation of stiffness matrix in AEM is different than that of FEM. The present study deals with derivation of stiffness matrix for one dimensional and two dimensional elements.

The study also includes basic introduction of applied element method and fundamental difference between FEM and AEM. Methodology used for applied element analysis is presented. Computer program for meshing of various types of structure is prepared and is used for preparing input data for main applied element analysis program. One dimensional problems like axially loaded column subjected to point load and uniformly distributed load is studied. Two dimensional problems like cantilever beam, deep beam with and without opening, portal frame are solved using computer program developed for Applied Element Analysis. Displacement results are obtained by varying size of elements and varying number of springs used for connectivity of

elements. For comparison of analysis results of AEM, finite element analysis of structure is carried out using ANSYS software. From the application of AEM to various problems it is observed that it can analyse various structure with similar accuracy as finite element analysis. Further the program developed in this study can be modified for dynamic analysis as well as for nonlinear analysis.

Acknowledgement

I would like to thank my guide **Dr. Paresh V. Patel**, whose keen interest and knowledge base helped me to carry out the major project work. His constant support and guidance during my project work equipped me with a great understanding of different aspects of the project work. He has shown keen interest in this work right from beginning and has been a great motivating factor in outlining the flow of my work.

I further extend my thanks to **Dr. P. H. Shah**, Head, Department of Civil Engineering and **Dr K Kotecha**, Director, Institute of Technology, Nirma University,Ahmedabad for providing all kind of required resources during my study.

Finally, I would like to thank The Almighty, my family and all my friends, for supporting and encouraging me in all possible ways throughout the dissertation work.

- Vikas B. Gohel

10MCLC03

Abbreviation, Notation and Nomenclature

AEM .... Applied Element Method

AEA .... Applied Element Analaysis

FEM .... Finite Element Method

FEA .... Finite Element Analysis

DCM .... Discrete Crack Methods

EDEM .... Extended Distinct Element Method

RBSM .... Rigid Body and Spring Model

K_{Normal} or K_n .... Normal spring stiffness

K_{Shear} or K_s .... Shear spring stiffness

E .... Young's Modulus

G .... Shear modulus

d .... Distance covered by springs

a .... Length of the representative area

t .... Thickness of an element

$\{F\}$ .... Applied load vector

Δ .... Displacement vector

K_G .... Global stiffness matrix

1D Element .... One dimensional element

2D Element .... Two dimensional element

P .... Axial force in spring

u_1 or u_4 .... Horizontal degree of freedom at centroid of element

u_2 or u_5 .... Vertical degree of freedom at centroid of element

u_3 or u_6 .... Rotational degree of freedom at centroid of element

K_{11}, K_{12}..etc .... Spring stiffness coefficients

α and θ .... Inclination of spring at first node

α_1 and θ_1 .... Inclination of spring at second node

Contents

List of Tables

List of Figures

Chapter 1

Introduction

1.1 General

Earthquake of last decades like Bhuj earthquake in 2001 caused heavy damage to multistorey buildings[7]. Many buildings were completely collapsed during earthquake. These damages have clearly shown behavior of the structure to given ground motion. Strength of a structure mainly depends on deformation capacities of the individual component of the structure. In order to determine capacity of any structure beyond the elastic limits some form of nonlinear analysis such as the pushover procedure are required to be performed. Usually seismic demands are computed by nonlinear static analysis of the structure, which is subjected to monotonically increasing lateral forces with a constant distribution of the forces throughout height until a target displacement is reached. However, clear understanding about the performance under critical dynamic loading is difficult to understand by following this procedure. For this purpose, a highly efficient numerical modeling procedures is required.

Simulation of such a behavior is not an easy task using currently available numerical techniques. Currently available numerical methods for structural analysis can be classified into two categories. In the first category, model is based on continuum material equations. The finite element method (FEM) is typical example of this category. The mathematical model of the structure is modified to account for reduced resistance of yielding components. Computer programs are available that directly model nonlinear behavior efficiently in static way and to some reasonable level of accuracy in dynamic way. However, to perform collapse behavior of the structures that exceeds beyond their elastic limit is a difficult task to solve by presently available numerical methods. Currently there are several limitations in adopting this approach. For example in case of highly nonlinear case where crack has initiated and element is not detached from the structure, smeared crack approach is followed. However, smeared Crack approach is difficult to use in zones where separation occurs between adjacent structural elements. While, Discrete Crack Methods (DCM) assumes that the location and direction of crack propagation are predefined before the analysis.[7]

The second category of methods uses the discrete element approaches like the Rigid Body and Spring Model (RBSM) and Modified or Extended Distinct Element Method (MDEM or EDEM). The main advantage of these methods is that they are efficient in modeling of cracks present in the structure. While the main advantage of these method is they can easily track the crack propagation path. Analysis using the RBSM could not be performed up to complete collapse of the structure. On the other hand, the EDEM can follow the structural behavior from zero loading and up to complete collapse of the structure. But the required accuracy in results cannot obtain using EDEM for small deformation range as compared with FEM. Hence, the failure behavior obtained by repeated many calculations is affected due to cumulative errors and cannot be predicted accurately using the EDEM. However, Applied Element Method (AEM) has the capability of simulating behavior of structures from zero loading to collapse can be followed with reliable accuracy, reasonable CPU time and with relatively simple material models.[7]

Hence numerical simulation can be performed with Applied Element Method (AEM). The major advantages of the Applied Element Method (AEM) are simple modeling and programming and high accuracy of the results with relatively short CPU time. Using the AEM, highly nonlinear behavior, i.e. crack initiation, crack propagation, separation of structural elements, rigid body motion of failed elements and collapse process of the structure can be followed with high accuracy.

The Applied Element Method (AEM) is a numerical analysis used in predicting the continuum and discrete behavior of structures. the purpose of AEM is to Bridge the gap between continuum models and discrete element models. The AEM provides techniques that allow a structure to be analyzed within one model from the unloaded stage, through small displacement loading, through large displacement loading, and up to collapse. Therefore, with the AEM, crack initiation, crack propagation, separation of structural elements, rigid body motion of element, and the collapse process of whole structure, can be modeled.

AEM is based on dividing the structure into small parts called Elements. Elements can be of various shapes like cuboid, trapezoidal ..etc. Each of which could be analyzed by relatively straight forward method. Then these elements are assembled into a complete structure for which solution can be obtained by simple mathematics.

The modeling method in AEM adopts the concept of discrete cracking allowing it to automatically track structural collapse behavior passing through all stages of loading: elastic, crack initiation and propagation in tension-weak materials, reinforcement yield, element separation, element contact and collision, as well as collision with the ground and adjacent structures.

1.2 Historical Background

The numerical analysis methods available at the time, were not adequate to model the full-range of loading that is seen during an earthquake or potential collapse event. The AEM was established at the University of Tokyo by Hatem Tagel-Din during his doctoral studies during the late 1990's (Tagel-Din, 1998). Motivation for the methodology arose from catastrophic earthquakes in the region (Kobe 1995) which brought about the realization that engineered structures could collapse. Acknowledging that engineered structures could collapse created an interest in analytical modeling for such events. Thus, the AEM was created, designed to overcome the limitations of current methods and allow for the complete collapse analysis of structures.

The term "Applied Element Method" was first coined in 2000 in a paper called "Applied element method for structural analysis: Theory and application for linear Materials." Since then AEM has been the subject of research by a number of academic institutions and the driving factor in real-world applications. Research has verified its accuracy for: elastic analysis, crack initiation and propagation, estimation of failure loads at reinforced concrete structures, reinforced concrete structures under cyclic loading, buckling and post-buckling behavior, nonlinear dynamic analysis of structures subjected to severe earthquakes, fault-rupture propagation, nonlinear behavior of brick structures, and the analysis of glass reinforced polymers (GFRP) walls under blast loads.

1.3 Introduction to Applied Element Method (AEM)

With the AEM, a structure is modeled by virtually dividing it into an assembly of small Elements. Adjacent elements are connected through series of normal and shear springs located at contact points that are distributed over the surface of each element. At each contact point, there is one normal spring and two shear springs in orthogonal directions. Fig.1.1 and 1.2 illustrates the division of a structure into elements and an example of the connectivity between elements. The area of influence on each element for a set of springs is highlighted in bold.

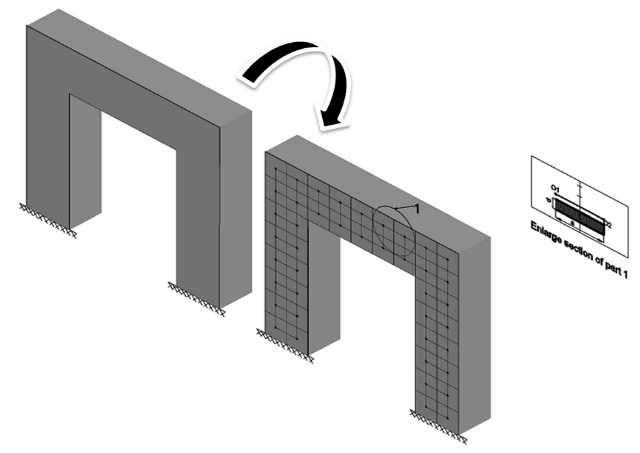


Figure 1.1: Division of structure into small element

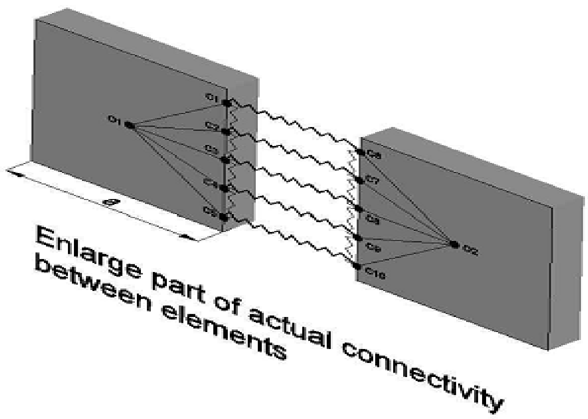


Figure 1.2: Spring distribution and area of influence for each pair of springs

The spring stiffness equations are:

$$k_{normal} = \frac{E \times d \times t}{a} \qquad and \qquad k_{shear} = \frac{G \times d \times t}{a} \tag{1.1}$$

Where, “E” and “G” are Young’s Modulus and Shear modulus, respectively, “d” is the distance covered by springs, “t” is the thickness of element, and “a” is the length of the representative area. This is the equation for axial stiffness.

In two dimensional problem each element has three degrees of freedom located at its centroid representing the rigid body motion of the element. Each set of springs on an elements surface can be geometrically related to the three degrees of freedom at the centroid, thus creating a stiffness matrix for that set of springs. The element stiffness matrix is then created by summing up the stiffness matrices of each individual set of springs. The model can then be analyzed by utilizing the following equation:

$$[K_G][\Delta] = [F] \tag{1.2}$$

Where,$\{F\}$ is applied load vector,$[\Delta]$ is the displacement vector and $[K_G]$ is the global stiffness matrix.

For large displacement analysis, modifications must be made to the force side of the governing equation to account for incompatibility between spring strains and stresses and the geometrical changes in the structure. Formulation of this equation is presented by Meguro and Tagel-Din in ”Applied Element Method Used for Large Displacement Structural Analysis”[12].

Utilizing these equations and techniques to model structures differentiate the AEM from other methodologies. The AEM has been proven to be as accurate as the FEM in modeling linear structures in the small deformation range, but through implementation of its principles, the AEM is able to surpass the capabilities of the FEM in large displacement and collapse events. The AEM offers four distinct advantages in structural modeling: automatic crack generation, element separation, element re-contact, and element impact.

Each of the advantages within the AEM arises from the use of springs to connect adjacent elements. For each set of normal and shear springs, stress and the corresponding strain is calculated throughout the loading. Referencing default or user-defined material properties, once the criteria is reached, springs are cut. This can occur anywhere within the model, therefore no pre-conceived notion of where the cracks will initiate is necessary. Crack propagation follows the same principles.

If all of the springs connecting an element are cut, the element is allowed to separate from the structure. In a dynamic analysis, the element has an assigned mass and generates inertial forces. As the element falls, if it contacts another element, the contact is automatically detected and contact springs are generated between the colliding elements.

These contact springs model the inertial forces transferred between the elements, as well as forces due to bearing and friction. The ability to cover this vast range of structural behavior in a single model is what distinguishes the AEM from other analysis methods.

1.3.1 Assumptions

Following assumptions are made in AEM.

- Elements are assumed to be rigid (i.e. Shape and size of element doesn't change under applied loading.).
- Elements are assumed to be connected on face with large number of springs.
- Assembly of rigid mass and spring behaves as Rigid body Spring Mass model(RBSM).
- Deformation of an element are equal to deformation of springs.

- Direction of loading are assumed to be constant for analysis of problem.

1.3.2 Various aspects of Applied Element Analysis

The knowledge of Applied Element Analysis (AEA) makes a good structural design engineer better while just user without the knowledge of AEA may produce more dangerous results. To use the AEA properly, the user must know the following points clearly:

a. Types of elements to be used for solving the problem in hand.

b. Modeling a Structure by using spring mass elements.

c. Discretization of structure.

d. Development of element properties and their limitations.

e. Introduction of boundary conditions.

f. Interpretation of results produced by AEA

g. Understanding the difficulties involved in the development of AEA programs. Hence there is a need for checking the commercially available package with the results of standard cases.

1.4 Comparison of AEM and FEM

Various points are discussed below to understand distinct idea of applied element method compared to Finite Element Method.

- Finite element method is based on continuum mechanics for numerical analysis of various problems. Whereas in case of applied element method it useful in both continuum as well as discrete/fracture mechanics.
- Finite element method is not much efficient beyond crack initiation, whereas AEM has no such limitation.
- In FEM one cannot model deteriorated structure where AEM is very easily used for such kind of modeling.

Now some points which describes difference in modeling of structural member.

- In case of FEM structure, lines and dummy plans are used for discretization of structure. Whereas, In AEM structure is an assembly of small elements.
- In FEM elements are compulsorily connected at nodes only. Whereas in AEM elements are connected along faces of elements.
- In FEM, Nodes are used for connection of an element. In AEM springs are used to connect elements at faces.
- Many types of finite elements are used for meshing of structure. Generally Cuboid is only element used for meshing of structure in AEM.
- The number and the types of degrees of freedom of the model depend on the type of finite elements used for modeling in FEM. In AEM two elements are connected through a series of contact points. At each points there are three contact springs: a normal spring and two shear springs.
- In FEM, transition elements are needed to switch from large sized elements to smaller elements as elements are connected at nodes only (i.e. No partial

connectivity is allowed). In case of AEM, No need for transition element as element are connected at faces (i.e. partial connectivity between elements is allowed), as shown in Fig1.3.

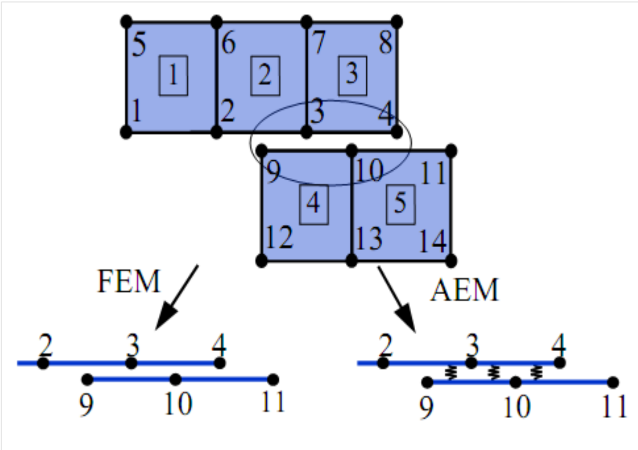


Figure 1.3: Partial connectivity of elements

- In case of FEM, the Global stiffness matrix [K] is determined based of contribution of each elements. In AEM, the Global stiffness matrix [K] is determined as sum of contributions of all springs.

1.5 Objectives of study

The main objective of present work is to understand applications of applied element method for linear static analysis of structure under applied loading conditions. From engineering point of view, development of efficient computer is more important for solving complex problems. The choice of element, organization of computational process and solution of equations are, therefore, important. Many times computer programs developed, suitable for specific need, are very useful for handling typical problems.

Looking to above need the objectives of present study are:

- To understand theory and applications of Applied Element Method for linear static analysis.
- To formulate element properties for one and two dimensional elements
- To develop computer program for Applied Element Method (AEM).
- To illustrate application of AEM to 1D and 2D problems.

1.6 Scope of Work

To achieve above objectives the scope of work is decided as follows:

- Understanding AEM and its difference with FEM.
- Formulation of properties of 1D and 2D elements.
- Development of computer program for analysis using AEM
- Linear static analysis of axially loaded column, cantilever beam, deep beam ..etc using AEM and comparison of results with other exact method. These problems are solved considering with varying no. of elements and varying no. of springs. Comparison of analysis results is made for better understanding of AEM

1.7 Organization of Report

The content of major project report is divided into different chapters as follows:

Chapter 1, presents the introduction and overview of the major project work, Historical background, Introduction to Applied Element Method (AEM), Comparison of AEM and FEM are included. It also includes objectives of study and scope of work.

Chapter 2, presents brief literature review on AEM. Literature survey is carried out to understand theory and applications of Applied Element Method (AEM).

Chapter 3, gives basics of Applied Element Method which includes discretization, methodology adopted,factors affecting analysis results.

Chapter 4, presents procedure followed for development of computer program for Applied Element Method (AEM). It also includes, computer implementation of AEM.

The application of applied element method are covered in **Chapter 5**. Application of AEM is explained by solving various types of problems like analysis of axially loaded column, cantilever beam, portal frame and deep beam.

Summary, conclusions and future scope of work are discussed in **Chapter 6**.

The source code of computer programmes, sample input file and output file are included in Appendix.

Chapter 2

Literature Review

2.1 General

For the objectives of major project discussed in Chapter 1, an extensive Literature review relevant to Applied element method is carried out. This chapter presents review of literature related to theoretical aspect and applications of applied element method.

2.2 Literature Review

2.2.1 Theoretical aspects

Meguro[7] discussed basic concept and advantages of the applied element method for structural analysis. Separate material models were used for plain concrete and reinforcement bars. Internal stresses and strains were calculated for reinforcement bars or concrete at any location. They had also given application of AEM for studying the nonlinear dynamic behavior of structures. Behavior of material under effects of cracking, concrete crushing, yield of reinforcement had been introduced. The applications of AEM for static and dynamic cases, monotonic and cyclic loading conditions and also for small and large deformation ranges were specified. Various topics based on fracture mechanics of structure like, collapse mechanism of structures, additional effects like buckling of reinforcement bars and spalling of concrete were also explained.

Tagel-din and Meguro[11] presented formulation and verification of numerical technique of elastic-static loading condition. The effect due to element size, number of connecting springs between elements and poisson's ratio had been clearly mentioned. They concluded that, increasing the number of connecting springs lead to much accurate results for crack propagation in nonlinear analysis. Increasing the number of connecting springs had no effect on stiffness matrix of an element as decreasing the number of spring lead to increase in area represented by each spring.

Meguro and Tagel-din[8] introduced the elements formulation of the applied-element method. They also discussed the effects of the element size and arrangement. The accuracy of the applied element method in a nonlinear material case was verified by studying the behavior of RC structures under cyclic loading. Effects caused by the size and arrangement of the elements were studied. They showed that accurate results of stresses and strains could be obtained by using small size elements. Crack initiation and Failure behavior simulation of RC structures had also been illustrated explicitly by authors. The calculated load-displacement relationship and the failure load results showed reliable accuracy.

Meguro and Tagel-Din[12] presented further applications of AEM where FEM doesn't perform efficiently. A brief overview of the method's formulation was presented. Modifications needed to analyze the behavior of structures subjected to large displacements under static loading were introduced. They also added some idea on simplicity and applicability of AEM. No geometric stiffness matrix was needed as it requires in FEM for simulation of large deformation of a member. So the formulation was simple, general and applicable to any type of structural configuration or material. Limited applications of the AEM were presented. The direction of the applied forces is assumed to be constant. Because of this, when load conditions changed its results were not accurate, and as in case if a member buckle. A series of examples that verify the applicability of the proposed technique were also presented.

2.2.2 Applications

In this section applications of AEM is reviewed.

Dessousky[1] discussed special algorithms to model the interface between the blocks and added to an Applied Element-Based solver. These algorithm predicted the strength and stiffness at interfaces when cracks opened and closed. In addition to interface springs, contact springs were added automatically when collision occur and springs were removed when elements separate. Through the use of interface and contact springs, contact, re-contact and dislocation phenomena were traced. The program results were verified first for small problems and then a case study was simulated. Results of response of a minaret consisting of stone blocks placed on top of one another, and expected damage patterns for sub-assemblages were also discussed.

Tagel-din[5] presented simulation of collapse processes of scaled reinforced concrete structure by using AEM and compared with results obtained by shake table experiments. Paper presented results of experiment performed using eleven storied RC building model subjected to a series of base excitations. The numerical simulation was performed with two-dimensional Applied Element Method (AEM). The simulated structural response had shown good agreement with experimental results. Due to the limitations of capacity of the shaking table used, the experiment was performed only up to the start of collapse. However, the numerical analysis using the AEM was extended to simulate the detailed collapse behavior under magnified base excitations.

Raparla[3] considered a set of four bare frames designed as per Indian Standards for studying performance of building up to collapse. All bare frames were subjected to Northridge earthquake ground motion (freq. 1-4 Hz). Applied Element Method (AEM) was used as numerical tool for analytical solution of bare frames. They observed that initial crack were appeared first in concrete as the height of structure were increasing. They also observed that, as the structure vibrates more bending stresses were likely to develop in taller structure due to same drift. All the structures had predominantly behaved in bending mode. However, as the steel bars failed, then the progressive collapse phase started in the frames. They found that progressive collapse started in single storey frame at 6.7 sec, in three storey frames at 5.8 sec, in five storey frame at 7 sec and in ten storey frame at 6.8 sec.

An effective application of Applied element method was presented by **Salem**[2]. Failure modeling of several reinforced concrete structures collapsed due to floods in Sinai and Aswan, Egypt had performed. Scouring of soil beneath foundations caused excessive differential settlements, leading to failure of main structural members and complete structural collapse. A three-dimensional nonlinear dynamic analysis of a multi-storey reinforced concrete framed structure with induced soil scour under its foundation was carried out. The analysis of the structure followed until its complete collapse. The numerical analysis were then used to propose a safe design against collapse. Three different alternatives were proposed for preventing progressive collapse by varying dimensions of floor beams, tie beams connecting footings, and diagonal bracings. They did not found much effectiveness of increasing the size of the floor beams on mitigating progressive collapse, while the use of diagonal bracings in the ground floor was found to be effective in preventing progressive collapse. The tie beam reinforcement was found to have a significant effect on the structural behavior during such an extreme loading case.

Tagel din[9] discussed the large deformation of elastic structure under static load condition. Formulation for stiffness matrix used was simple and general computational process. They specified simplicity and applicability of stiffness matrix to the various structural configuration and material types. Technique used was based on determination of residual forces due to geometrical changes of the structure during simulation. They checked the accuracy of proposed technique by comparing with theoretical results. By using this technique they solved problems of the buckling of column and post buckling behavior of structure.

2.3 Summary

In this chapter, review of literature related to Applied element method is carried out. Literature related to theoretical background of method and application of AEM is reviewed in this chapter.

Chapter 3

Basics of Applied Element Method

3.1 General

Applied Element Method (AEM) is numerical technique for analyzing structural systems. A structure can be defined as a skeletal or a continuum. Examples of skeletal structure are:

- Multistorey building frame.
- Truss...etc

Examples of continuum structure are:

- Dam
- Deep Beam...etc

The objective of structural analysis is to find deflection and stresses magnitude at a desired location of skeletal or continuum structure.

3.2 Types of elements

Following are the various types of elements used in structural analysis using AEM.

a. One dimensional element

b. Two dimensional element

c. Three dimensional element

3.2.1 One dimensional element

If the geometry, material properties, and the field variable of the problem can be described in terms of only one spatial coordinate then one dimensional element can be used. In case of one dimensional element one normal spring is considered for connection between elements at contact points located at the faces of each element as shown in Fig3.1. This normal spring will be totally responsible for the axial deformation of an element. Although these elements have cross-sectional area, they are generally shown schematically as a cube. In some case depending upon typical geometry of a structure cross section of a cube element may vary.

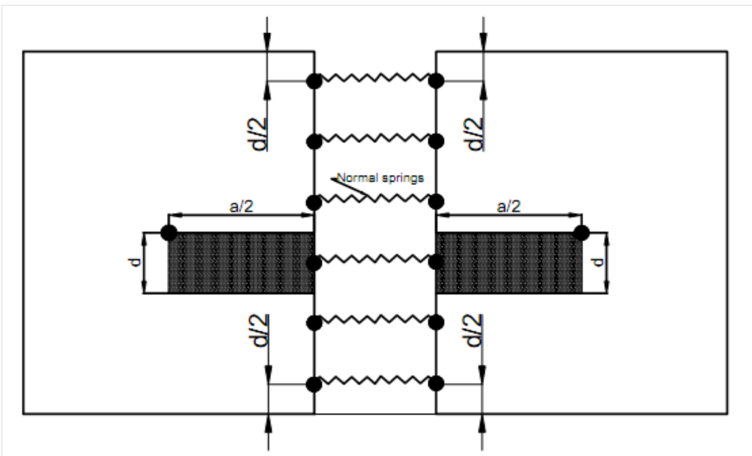


Figure 3.1: One dimensional element connectivity

3.2.2 Two dimensional element

When the configuration and field variables of the problem can be described in terms of two independent spatial coordinates, the two-dimensional elements can be used. The

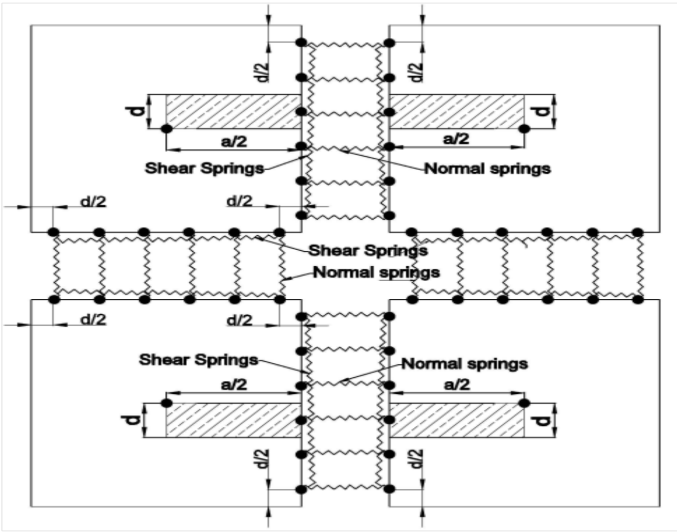


Figure 3.2: Two dimensional element connectivity.

basic element useful for two-dimensional analysis is cube. In case of two dimensional element a pair of one normal spring and one transverse spring are considered for connection between elements at contact points located at the faces of each element shown in Fig3.2. Although these elements have cross-sectional area, they are generally shown schematically as a cube. In some case depending upon typical geometry of a structure cross section of a cube element may vary.
Same way in case of three dimensional element (not considered for project work) a pair of three springs i.e. one normal to the face and two springs in transverse direction to the normal is used at each contact point.

3.3 Basic element shape

The shapes, sizes, number, and configurations of the elements have to be chosen carefully such that the original geometry of body is simulated as closely as possible without increasing the computational effort needed for the solution. Mostly the type of element considered for analysis of all kind of geometries is cube. All neighbouring element have node located at the centroid of each element. When we consider two adjacent elements then it will represent two node element having length equal to the center to center dimension of elements. Element connectivity is assumed on faces of each element using a pair of springs.

3.4 Discretization

The basic concept of AEM is discretization. Discretization (Discrete means separate) is a process in which the system is divided into many distinct smaller portions. These portions are known as an element. And the opposite of discretization is assembly or composing. AEM consists of both discretization and assembling of an element. In process of assembling, the elements are placed back and assembled. In short, in AEM first the system is discretized and then finally assembled. In order to derive the force deformation relationship of structure discretization is carried out. It is difficult to find force deformation relation of entire structure so, the structure is divided into small elements known as discretization. Based on force deformation characteristic properties i.e. stiffness matrix of element is developed. After finding properties of individual elements they are assembled together to represent properties of entire structure. Assembling of element properties should ensure equilibrium and compatibility.

3.5 Node generation

In applied element analysis structure is meshed with the small element by dividing structure virtually. Generating nodes is preliminary task to perform which will define the geometry of a member. Node in member represents small element in discretized structure. In applied element analysis nodes are generated manually or with automated meshing tool which depends on type of geometry to be analyzed. Number of nodes to be used for analysis depends on discretization of any member. Smaller the size of element discretization higher will be total number of nodes. For 1D analysis of structural element Nodes are distributes along major axis of an element. For example, if 1D bar is considered, bar element is used for analysis of member subjected to axial force only. These members are having one dimensional (length) considerably large as compared to cross sectional dimensions. Bar in tension and axially loaded column falls under these category. In case of 2D and 3D analysis nodes are generated along all principle directions.

3.6 Element connectivity and contact point

In AEM, meshing is performed to discretize the geometry of structure into small elements. The effect of discretization of member can be explained in simplest way using varying element sizes. Effect of discretization has been discussed in Chapter 6 by solving various problems using varying meshing patterns. However, In FEM the problem domain can be divided (meshed) into small elements using a set of nodes that are connected in a predefined manner using nodal lines, the solution within each element can be approximated very easily using simple functions such as polynomials, which are termed shape functions. But in case of AEM small elements are connected to each of its neighbour element on their side faces. Elements are connected using pair of springs at their contact points as shown in Fig3.3. In this method each side faces are also messed to define the location of contact points which mainly depends on the numbers of springs to be used for connectivity of an element. Distribution of pair of springs at contact point on face of element is as shown in Fig3.3. In case of 1D element connectivity the spring is considered in one direction only using linear spring. Same way a pair of one normal and one shear spring is to be taken into account for 2D element. Connectivity for 2D element is performed in two principle direction. Information, such as the element connectivity must also be created during the meshing for simulation.

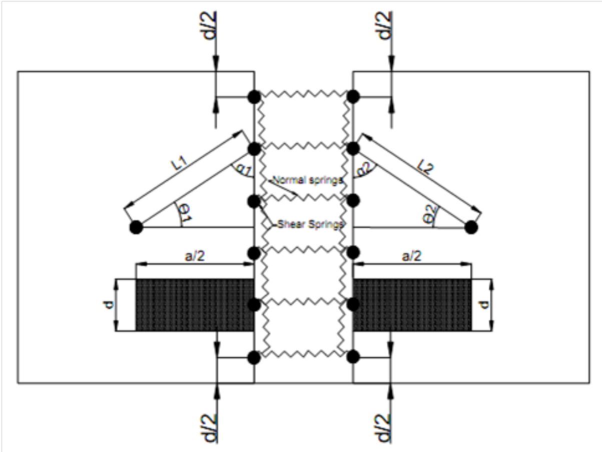


Figure 3.3: Arrangement of contact points and influence of each springs.

3.7 Formulation of elemental stiffness matrix

Formulation of elemental stiffness matrix for one dimensional and two dimensional elements are explained in section 3.7.1 and 3.7.2. Remaining steps of analysis i.e. assembly of stiffness matrix, incorporation of boundary conditions, solution of equations to obtain displacements are same as finite element method or other displacement based methods.

3.7.1 One dimensional element

Consider a linear spring as shown in Fig3.4 the displacements of the two end points of the spring are u1 and u2 and the two points are subjected to axial forces f1 and f2 respectively. Both displacements and forces are assumed in the right-hand side direction which is assumed to be positive in the present applied element formulation. If the spring is in equilibrium, the sum of forces becomes zero. That is,

$$f_1 + f_2 = 0 \tag{3.1}$$

As a result, f_2 = - f_1 and Fig.3.4 shows the equilibrated linear spring. The spring is compressed by these forces and the contraction of the spring is proportional to them.

Using the spring constant k, the force and displacement relationship becomes,

$$k(u_1 - u_2) = f_1 \tag{3.2}$$

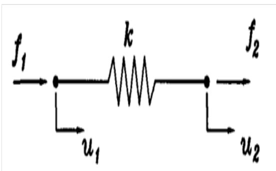
From Eqs (3.1) and (3.2), we obtain

$$k(-u_1 + u_2) = f_2 \tag{3.3}$$

Rewriting Eqs (3.2) and (3.3) in matrix form yields

$$\begin{bmatrix} k & -k \\ -k & k \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} f_1 \\ f_2 \end{Bmatrix} \tag{3.4}$$

This is the matrix equation for a linear spring. A spring is like a linear finite element. As a result, the matrix is called the element stiffness matrix and the right-hand side vector is called the element force vector. A system consisting of series of linear springs can be analysed.



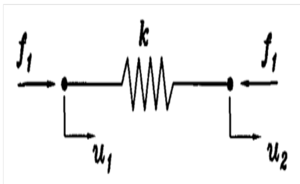


Figure 3.4: Linear spring

The linear spring can represent various systems in engineering applications. One direct application is the axial member. Consider an axial member with length L, uniform cross- section A and elastic modulus E. The elongation δ of the axial member subjected to an axial force P is computed from,

$$\delta = \frac{PL}{AE} \tag{3.5}$$

Rewriting above equation,

$$P = \frac{AE}{L}\delta \tag{3.6}$$

As a result, the equivalent spring constant for the axial member is.

$$k_{eq} = \frac{AE}{L} \tag{3.7}$$

3.7.2 Two dimensional element

Consider two elements connected by pairs of normal and shear springs located at contact points which are distributed around the element edges as shown in Fig3.5. Each pair of springs totally represents stresses and deformations of a certain area of the specified elements. The spring stiffness is determined as equation [3.8]:

$$K_{normal} = \frac{E \times d \times t}{a} \qquad K_{shear} = \frac{G \times d \times t}{a} \tag{3.8}$$

Where, "d" is the distance between springs, "t" is the thickness of the element and "a" is the length of the representative area, E and G are Young's and shear modulus of the material, respectively. The above equation indicates that each spring represents the stiffness of an area (d×t) with length "a" of the specified material. In case of reinforcement, this area is replaced by that of the reinforcement bar. The above equation indicates that the spring stiffness is calculated as if the spring connects the element centerlines.

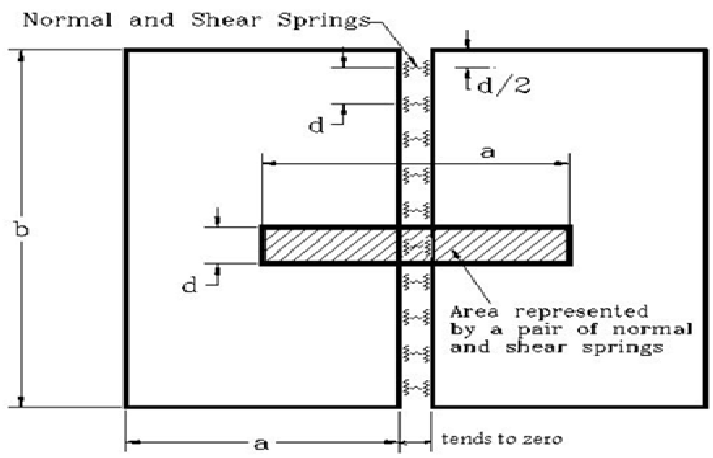


Figure 3.5: Spring distribution and area of influence for each springs

Now, Fig3.6 shows two elements which are connected by pair of springs at their contact points. Each element is having 3 DOF, u_1, u_2 and u_3 respectively at centroid of each element. Therefore, total 6 DOF are considered for each adjacent element. To calculate stiffness matrix of an element shown in Fig3.6 unit displacement in the direction of each DOF, is applied.

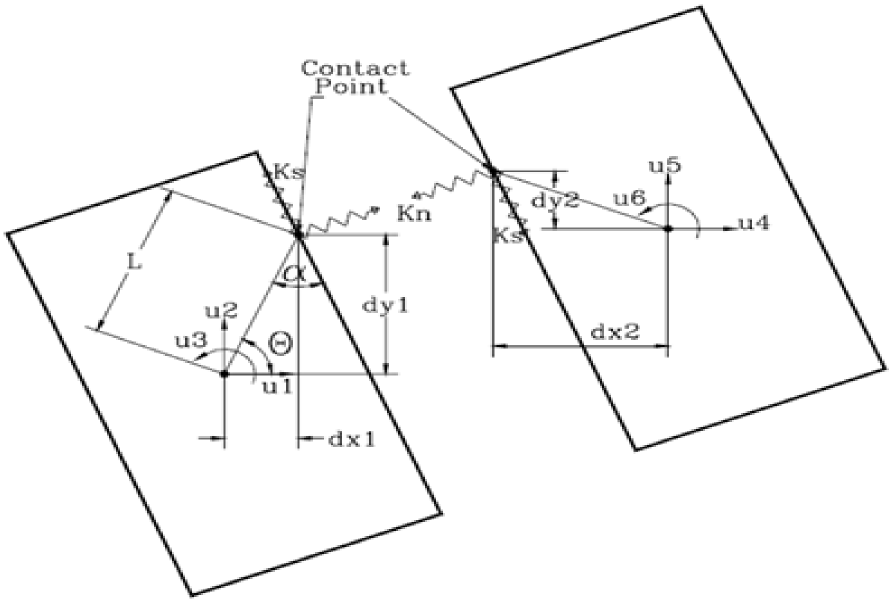


Figure 3.6: General position of a deformed element

In case of 2-D analysis, three degrees of freedom are assumed for each element while six degrees of freedom are used for 3-D problems. These degrees of freedom represent the rigid body motion of the element in 2-D or 3-D. Although the element motion is as a rigid body, its internal deformations are represented by the spring deformation around each element.

To get a general stiffness matrix, the element and contact springs locations are assumed in a general position. The stiffness matrix components corresponding to each degree of freedom are determined by applying a unit displacement in that degree of freedom direction and by determining forces at the centroid of each element. The element stiffness matrix size is only (6 x 6) in case of 2-D analysis and it is (12 x 12) in case of 3-D analysis. It is clear that the stiffness matrix depends on the contact spring stiffness and the spring location.

To calculate components of stiffness matrix, unit displacement in direction of u_1 is considered and the action corresponding to u_1, u_2, u_3, u_4, u_5, u_6 are obtained as shown in Fig3.7. By this we get,

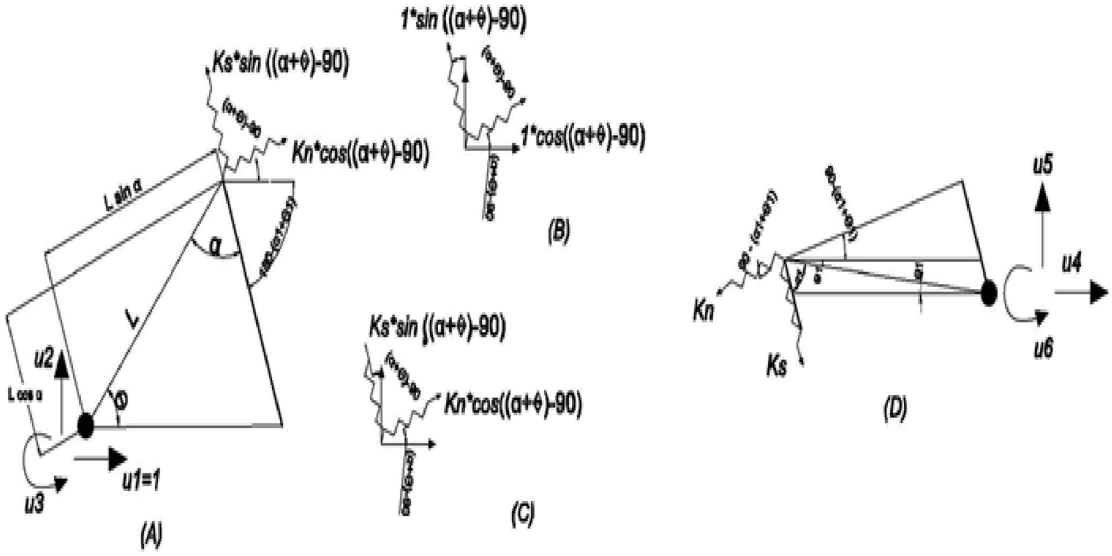


Figure 3.7: Unit displacement in direction of u_1 i.e $u_1 = 1$

$$K_{11} = K_n \sin(\alpha+\theta)\sin(\alpha+\theta) + K_s \cos(\alpha+\theta)\cos(\alpha+\theta) \tag{3.9}$$

$$K_{12} = -K_n \sin(\alpha+\theta)\cos(\alpha+\theta) + K_s \cos(\alpha+\theta)\sin(\alpha+\theta) \tag{3.10}$$

$$K_{13} = -K_n \sin(\alpha+\theta) L \cos(\alpha) + K_s \cos(\alpha+\theta) L \sin(\alpha) \tag{3.11}$$

$$K_{14} = -K_n \sin(\alpha_1+\theta_1)\sin(\alpha+\theta) + K_s \cos(\alpha_1+\theta_1)\cos(\alpha+\theta) \tag{3.12}$$

$$K_{15} = -K_n \cos(\alpha_1+\theta_1)\sin(\alpha+\theta) - K_s \sin(\alpha_1+\theta_1)\cos(\alpha+\theta) \tag{3.13}$$

$$K_{16} = K_n \sin(\alpha+\theta) L_1 \cos(\alpha 1) + K_s \cos(\alpha+\theta) L_1 \sin(\alpha_1) \tag{3.14}$$

Where α and θ corresponds to inclination of spring at first node while α_1 and θ_1 corresponds to inclination of spring at second node as 3.6.

Similarly, applying unit displacement in direction of u_2 (i.e. u_2=1) as shown in Fig3.8.

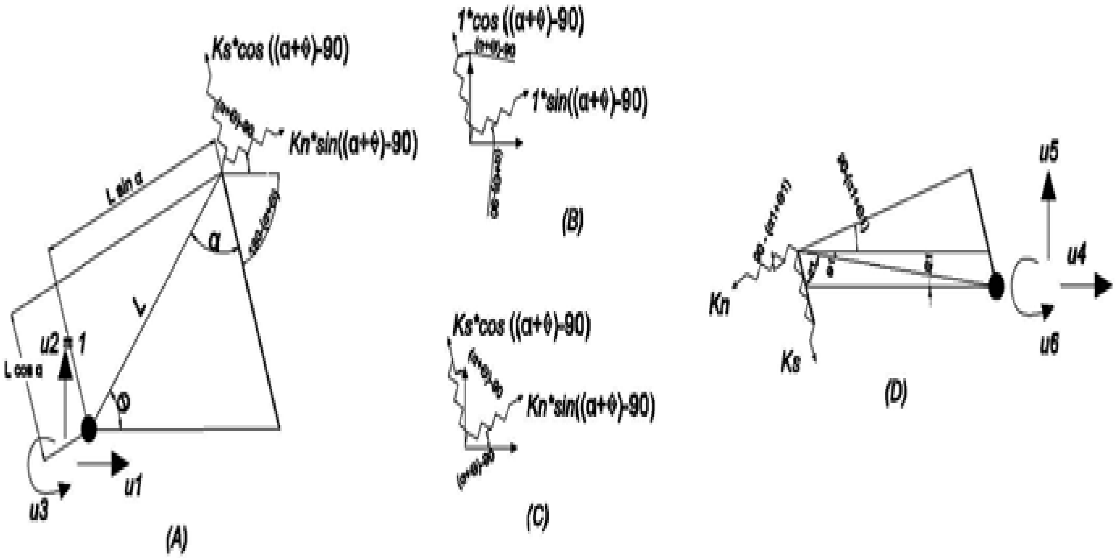


Figure 3.8: Unit displacement in direction of u_2 i.e $u_2 = 1$

$$K_{21} = -K_n \sin(\alpha+\theta)\cos(\alpha+\theta) + K_s \cos(\alpha+\theta)\sin(\alpha+\theta) \tag{3.15}$$

$$K_{22} = K_n \cos(\alpha+\theta)\cos(\alpha+\theta) + K_s \sin(\alpha+\theta)\sin(\alpha+\theta) \tag{3.16}$$

$$K_{23} = K_n \cos(\alpha+\theta)L\cos(\alpha) + K_s \sin(\alpha+\theta)L\sin(\alpha) \tag{3.17}$$

$$K_{24} = K_n \sin(\alpha_1+\theta_1)\cos(\alpha+\theta) + K_s \cos(\alpha_1+\theta_1)\sin(\alpha+\theta) \tag{3.18}$$

$$K_{25} = K_n \cos(\alpha_1+\theta_1)\cos(\alpha+\theta) - K_s \sin(\alpha_1+\theta_1)\sin(\alpha+\theta) \tag{3.19}$$

$$K26 = -K_n \cos(\alpha+\theta)L_1\cos(\alpha_1) + K_s \sin(\alpha+\theta)L_1\sin(\alpha_1) \tag{3.20}$$

Applying unit rotation in direction of u_3 (i.e. $u_3 = 1$) as shown in Fig3.9.

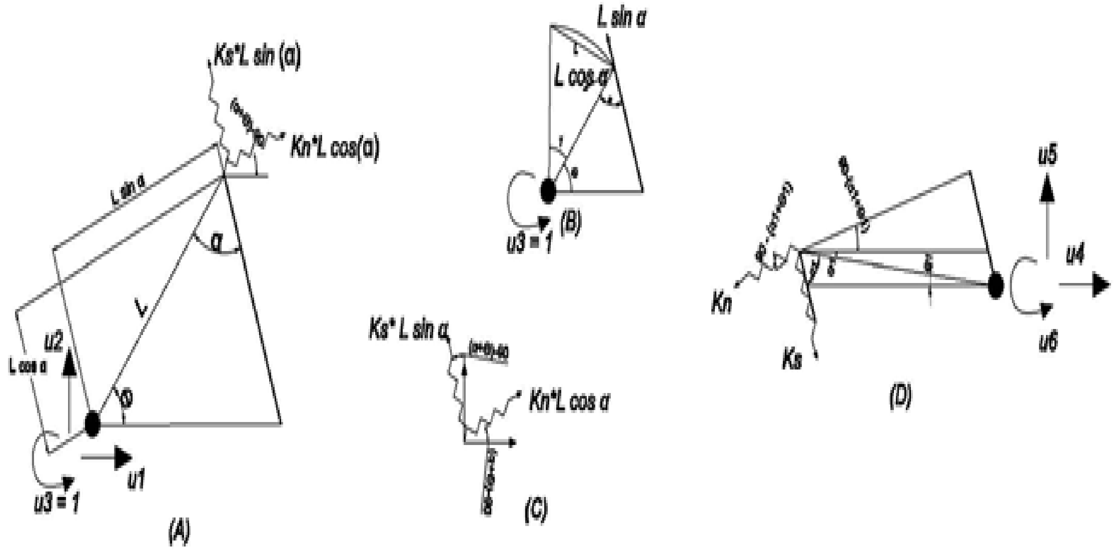


Figure 3.9: Unit rotation in direction of u_3 i.e $u_3 = 1$

.

$$K_{31} = -K_n \sin(\alpha + \theta) L \cos(\alpha) + K_s \cos(\alpha + \theta) L \sin(\alpha) \tag{3.21}$$

$$K_{32} = K_n \cos(\alpha + \theta) L \cos(\alpha) + K_s \sin(\alpha + \theta) L \sin(\alpha) \tag{3.22}$$

$$K_{33} = K_n L \cos(\alpha) L \cos(\alpha) + K_s L \sin(\alpha) L \sin(\alpha) \tag{3.23}$$

$$K_{34} = K_n \sin(\alpha_1 + \theta_1) L \cos(\alpha) + K_s \cos(\alpha_1 + \theta_1) L \sin(\alpha) \tag{3.24}$$

$$K_{35} = K_n \cos(\alpha_1 + \theta_1) L \cos(\alpha) - K_s \sin(\alpha_1 + \theta_1) L \sin(\alpha) \tag{3.25}$$

$$K_{36} = -K_n L \cos(\alpha) L_1 \cos(\alpha_1) + K_s L \sin(\alpha) L_1 \sin(\alpha_1) \tag{3.26}$$

Applying unit displacement in direction of u_4 (i.e. $u_4 = 1$) as shown in Fig3.10, and considering action corresponding to other DOF, stiffness coefficients are obtained as follows:

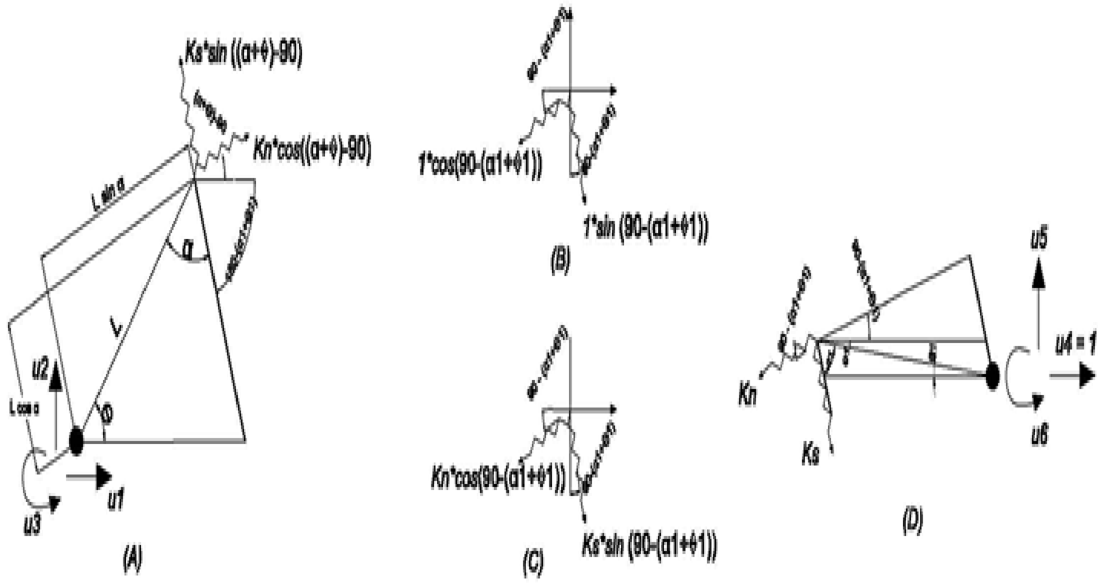


Figure 3.10: Unit displacement in direction of u_4 i.e $u_4 = 1$

$$K_{41} = -K_n \sin(\alpha 1 + \theta 1)\sin(\alpha + \theta) + K_s \cos(\alpha 1 + \theta 1)\cos(\alpha + \theta) \tag{3.27}$$

$$K_{42} = K_n \sin(\alpha_1 + \theta_1)\cos(\alpha + \theta) + K_s \cos(\alpha_1 + \theta_1)\sin(\alpha + \theta) \tag{3.28}$$

$$K_{43} = K_n \sin(\alpha_1 + \theta_1)L\cos(\alpha) + K_s \cos(\alpha_1 + \theta_1)L\sin(\alpha) \tag{3.29}$$

$$K_{44} = K_n \sin(\alpha_1 + \theta_1)\sin(\alpha_1 + \theta_1) + K_s \cos(\alpha_1 + \theta_1)\cos(\alpha_1 + \theta_1) \tag{3.30}$$

$$K_{45} = K_n \cos(\alpha_1 + \theta_1)\sin(\alpha_1 + \theta_1) - K_s \sin(\alpha_1 + \theta_1)\cos(\alpha_1 + \theta_1) \tag{3.31}$$

$$K_{46} = -K_n \sin(\alpha_1 + \theta_1)L_1\cos(\alpha_1) + K_s \cos(\alpha_1 + \theta_1)L_1\sin(\alpha_1) \tag{3.32}$$

By applying unit displacement in direction of u_5 (i.e. $u_5 = 1$) as shown in Fig3.11.

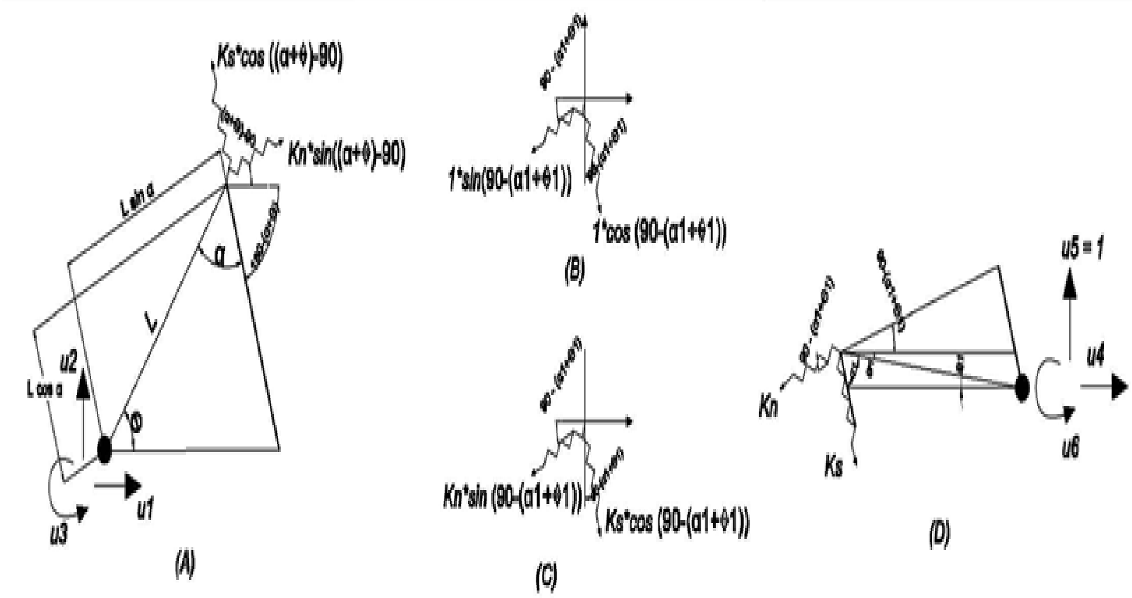


Figure 3.11: Unit displacement in direction of u_5 i.e $u_5 = 1$

$$K_{51} = -K_n \sin(\alpha + \theta) \cos(\alpha_1 + \theta_1) - K_s \cos(\alpha + \theta) \sin(\alpha_1 + \theta_1) \tag{3.33}$$

$$K_{52} = K_n \cos(\alpha + \theta) \cos(\alpha_1 + \theta_1) - K_s \sin(\alpha + \theta) \sin(\alpha_1 + \theta_1) \tag{3.34}$$

$$K_{53} = K_n \cos(\alpha_1 + \theta_1) L \cos(\alpha) - K_s \sin(\alpha_1 + \theta_1) L \sin(\alpha) \tag{3.35}$$

$$K_{54} = K_n \sin(\alpha_1 + \theta_1) \cos(\alpha_1 + \theta_1) - K_s \cos(\alpha_1 + \theta_1) \sin(\alpha_1 + \theta_1) \tag{3.36}$$

$$K_{55} = K_n \cos(\alpha_1 + \theta_1) \cos(\alpha_1 + \theta_1) - K_s \sin(\alpha_1 + \theta_1) \sin(\alpha_1 + \theta_1) \tag{3.37}$$

$$K_{56} = -K_n \cos(\alpha_1 + \theta_1) L_1 \cos(\alpha_1) + K_s \sin(\alpha_1 + \theta_1) L_1 \sin(\alpha_1) \tag{3.38}$$

Applying unit rotation in direction of u_6 (i.e. $u_6 = 1$) as shown in Fig3.12.

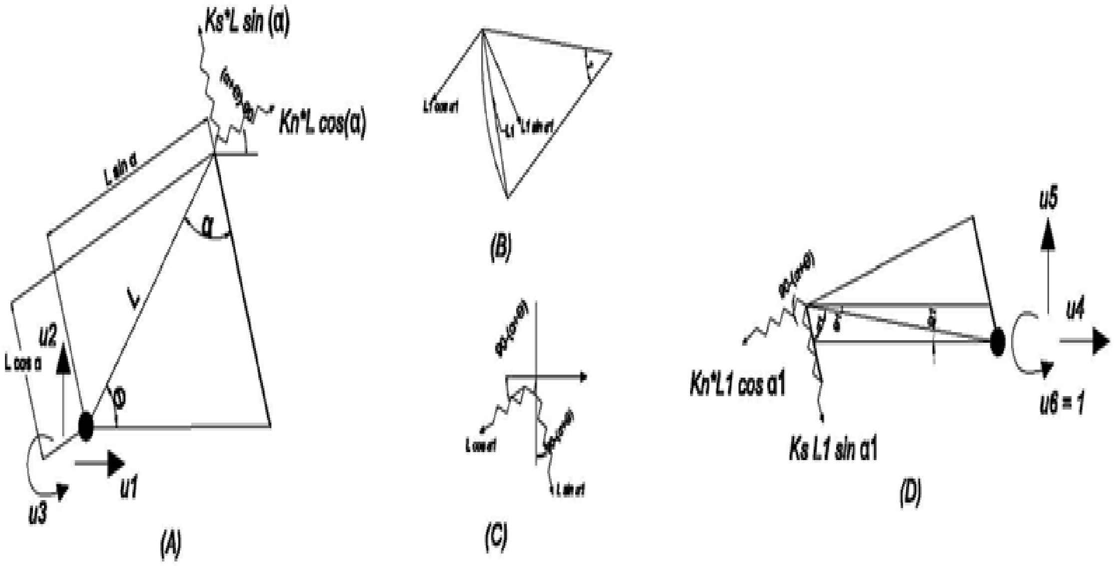


Figure 3.12: Unit rotation in direction of u_6 i.e $u_6 = 1$

$$K_{61} = K_n \sin(\alpha + \theta)L_1 \cos(\alpha_1) + K_s \cos(\alpha + \theta)L_1 \sin(\alpha_1) \tag{3.39}$$

$$K_{62} = -K_n \cos(\alpha + \theta)L_1 \cos(\alpha_1) + K_s \sin(\alpha + \theta)L_1 \sin(\alpha_1) \tag{3.40}$$

$$K_{63} = K_n L \cos(\alpha)L_1 \cos(\alpha_1) + K_s L \sin(\alpha)L_1 \sin(\alpha_1) \tag{3.41}$$

$$K_{64} = -K_n \sin(\alpha_1 + \theta_1)L_1 \cos(\alpha_1) - K_s \cos(\alpha_1 + \theta_1)L_1 \sin(\alpha_1) \tag{3.42}$$

$$K_{65} = -K_n \cos(\alpha_1 + \theta_1)L_1 \cos(\alpha_1) - K_s \sin(\alpha_1 + \theta_1)L_1 \sin(\alpha_1) \tag{3.43}$$

$$K_{66} = K_n L_1 \cos(\alpha_1)L_1 \cos(\alpha_1) + K_s L_1 \sin(\alpha_1)L_1 \sin(\alpha_1) \tag{3.44}$$

Combining stiffness matrix corresponding to all degree of freedom, stiffness matrix of an element is obtained as follows.

$$
\begin{bmatrix}
\begin{matrix} K_n \sin(\theta+\alpha)\sin(\theta+\alpha) \\ +K_s \cos(\theta+\alpha)\cos(\theta+\alpha) \end{matrix} &
\begin{matrix} -K_n \sin(\theta+\alpha)\cos(\theta+\alpha) \\ +K_s \cos(\theta+\alpha)\sin(\theta+\alpha) \end{matrix} &
\begin{matrix} K_s \cos(\theta+\alpha)L\sin(\alpha) \\ -K_n \sin(\theta+\alpha)L\cos(\alpha) \end{matrix} &
\begin{matrix} -K_n \sin(\alpha_1+\theta_1)\sin(\alpha+\theta) \\ +K_s \cos(\alpha_1+\theta_1)\cos(\alpha+\theta) \end{matrix} &
\begin{matrix} -K_n \cos(\alpha_1+\theta_1)\sin(\alpha+\theta) \\ -K_s \sin(\alpha_1+\theta_1)\cos(\alpha+\theta) \end{matrix} &
\begin{matrix} K_n \sin(\alpha+\theta)L_1\cos(\alpha_1) \\ +K_s \cos(\alpha+\theta)L_1\sin(\alpha_1) \end{matrix} \\[2ex]
\begin{matrix} -K_n \sin(\theta+\alpha)\cos(\theta+\alpha) \\ +K_s \cos(\theta+\alpha)\sin(\theta+\alpha) \end{matrix} &
\begin{matrix} K_n \sin(\theta+\alpha)\sin(\theta+\alpha) \\ +K_s \cos(\theta+\alpha)\cos(\theta+\alpha) \end{matrix} &
\begin{matrix} K_s \sin(\theta+\alpha)L\sin(\alpha) \\ +K_n \cos(\theta+\alpha)L\cos(\alpha) \end{matrix} &
\begin{matrix} K_n \sin(\alpha_1+\theta_1)\cos(\alpha+\theta) \\ +K_s \cos(\alpha_1+\theta_1)\sin(\alpha+\theta) \end{matrix} &
\begin{matrix} K_n \cos(\alpha_1+\theta_1)\cos(\alpha+\theta) \\ -K_s \sin(\alpha_1+\theta_1)\sin(\alpha+\theta) \end{matrix} &
\begin{matrix} -K_n \cos(\alpha+\theta)L_1\cos(\alpha_1) \\ +K_s \sin(\alpha+\theta)L_1\sin(\alpha_1) \end{matrix} \\[2ex]
\begin{matrix} K_s \cos(\theta+\alpha)L\sin(\alpha) \\ -K_n \sin(\theta+\alpha)L\cos(\alpha) \end{matrix} &
\begin{matrix} K_s \sin(\theta+\alpha)L\sin(\alpha) \\ +K_n \cos(\theta+\alpha)L\cos(\alpha) \end{matrix} &
\begin{matrix} K_s L\sin(\alpha)L\sin(\alpha) \\ +K_n L\cos(\alpha)L\cos(\alpha) \end{matrix} &
\begin{matrix} K_n \sin(\alpha_1+\theta_1)L\cos(\alpha) \\ +K_s \cos(\alpha_1+\theta_1)L\sin(\alpha) \end{matrix} &
\begin{matrix} K_n \cos(\alpha_1+\theta_1)L\cos(\alpha) \\ -K_s \sin(\alpha_1+\theta_1)L\sin(\alpha) \end{matrix} &
\begin{matrix} -K_n L\cos(\alpha)L_1\cos(\alpha_1) \\ +K_s L\sin(\alpha)L_1\sin(\alpha_1) \end{matrix} \\[2ex]
\begin{matrix} -K_n \sin(\alpha_1+\theta_1)\sin(\alpha+\theta) \\ +K_s \cos(\alpha_1+\theta_1)\cos(\alpha+\theta) \end{matrix} &
\begin{matrix} K_n \sin(\alpha_1+\theta_1)\cos(\alpha+\theta) \\ +K_s \cos(\alpha_1+\theta_1)\sin(\alpha+\theta) \end{matrix} &
\begin{matrix} K_n \sin(\alpha_1+\theta_1)L\cos(\alpha) \\ +K_s \cos(\alpha_1+\theta_1)L\sin(\alpha) \end{matrix} &
\begin{matrix} K_n \sin(\alpha_1+\theta_1)\sin(\alpha_1+\theta_1) \\ +K_s \cos(\alpha_1+\theta_1)\cos(\alpha_1+\theta_1) \end{matrix} &
\begin{matrix} K_n \cos(\alpha_1+\theta_1)\sin(\alpha_1+\theta_1) \\ -K_s \sin(\alpha_1+\theta_1)\cos(\alpha_1+\theta_1) \end{matrix} &
\begin{matrix} -K_n \sin(\alpha_1+\theta_1)L_1\cos(\alpha_1) \\ +K_s \cos(\alpha_1+\theta_1)L_1\sin(\alpha_1) \end{matrix} \\[2ex]
\begin{matrix} -K_n \sin(\alpha+\theta)\cos(\alpha_1+\theta_1) \\ -K_s \cos(\alpha+\theta)\sin(\alpha_1+\theta_1) \end{matrix} &
\begin{matrix} K_n \cos(\alpha+\theta)\cos(\alpha_1+\theta_1) \\ -K_s \sin(\alpha+\theta)\sin(\alpha_1+\theta_1) \end{matrix} &
\begin{matrix} K_n \cos(\alpha_1+\theta_1)L\cos(\alpha) \\ -K_s \sin(\alpha_1+\theta_1)L\sin(\alpha) \end{matrix} &
\begin{matrix} K_n \sin(\alpha_1+\theta_1)\cos(\alpha_1+\theta_1) \\ -K_s \cos(\alpha_1+\theta_1)\sin(\alpha_1+\theta_1) \end{matrix} &
\begin{matrix} K_n \cos(\alpha_1+\theta_1)\cos(\alpha_1+\theta_1) \\ -K_s \sin(\alpha_1+\theta_1)\sin(\alpha_1+\theta_1) \end{matrix} &
\begin{matrix} -K_n \cos(\alpha_1+\theta_1)L_1\cos(\alpha_1) \\ +K_s \sin(\alpha_1+\theta_1)L_1\sin(\alpha_1) \end{matrix} \\[2ex]
\begin{matrix} K_n \sin(\alpha+\theta)L_1\cos(\alpha_1) \\ +K_s \cos(\alpha+\theta)L_1\sin(\alpha_1) \end{matrix} &
\begin{matrix} -K_n \cos(\alpha+\theta)L_1\cos(\alpha_1) \\ +K_s \sin(\alpha+\theta)L_1\sin(\alpha_1) \end{matrix} &
\begin{matrix} K_n L\cos(\alpha)L_1\cos(\alpha_1) \\ +K_s L\sin(\alpha)L_1\sin(\alpha_1) \end{matrix} &
\begin{matrix} -K_n \sin(\alpha_1+\theta_1)L_1\cos(\alpha_1) \\ -K_s \cos(\alpha_1+\theta_1)L_1\sin(\alpha_1) \end{matrix} &
\begin{matrix} -K_n \cos(\alpha_1+\theta_1)L_1\cos(\alpha_1) \\ -K_s \sin(\alpha_1+\theta_1)L_1\sin(\alpha_1) \end{matrix} &
\begin{matrix} K_n L_1\cos(\alpha_1)L_1\cos(\alpha_1) \\ +K_s L_1\sin(\alpha_1)L_1\sin(\alpha_1) \end{matrix}
\end{bmatrix}
\tag{3.45}
$$

3.8 Factors affecting analysis results

To obtain realistic design of any kind of structure, analysis results should be more accurate. Results obtain from numerical technique have some dependent variables which affects the analysis results. Following are the some factors which have considerable importance for Applied Element Analysis (AEA) used in this work.

3.8.1 Effect of number of connecting springs

The effect of number of connecting spring between elements has been explained by Meguro[7]. The number of connecting spring between elements is the key factor that is to be considered in applied element analysis. Effect of connecting springs can be more precisely understood by performing nonlinear analysis of structure. This section gives idea for selecting number of connecting spring in elastic analysis. Referring to

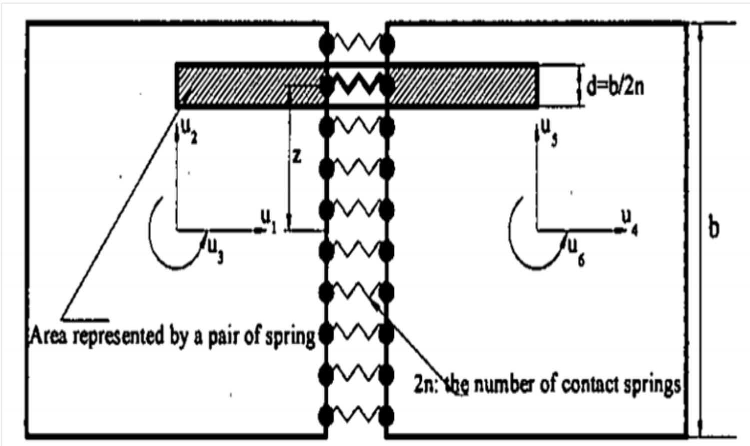


Figure 3.13: Effect of numbers of connecting springs

Fig.3.13, it is assumed that “2n” springs are connecting two elements together. Each spring represents the stiffness of a distance of (b/2n). Number of connecting springs has no effect on translation degree of freedom but it’s effect is very well seen in case of rotational degree of freedom.

Decreasing the number of connecting spring leads to increasing area to be represented by each spring. Ultimately the total area represented by springs becomes equal to the complete area of an element. It means that one spring can represent totally translational degrees of freedom of an element but cannot contribute to rotational degree of freedom. Rotation of an element has effect due to the number of springs. For static analysis effect of number of spring is discussed in chapter 5 by solving different types of problems.

3.8.2 Effect of Element Size

Adjustment of element size in the analysis is very important. Simulation of structures using elements of large size leads to increasing the structure stiffness and failure load. This means that the calculated displacements become smaller and by this failure load will be get larger than the actual one.

3.9 Summary

Basics of applied element method (AEM) are discussed in this chapter. One dimensional and two dimensional elements are presented. Derivation of stiffness matrix for one and two dimensional applied element id presented in this chapter.

Chapter 4

Development of computer program

4.1 General

Theoretical formulation of Applied Element Method is presented in Chapter 3. In this chapter computer implementation of AEM is discussed. The computer program is prepared in C-language. The computer program is useful in solving number of problems with varying geometry, discretization and number of springs.

Applied element analysis involves three stages of activity: preprocessing, processing, and postprocessing. Preprocessing involves the preparation of data, such as member data, element data, nodal coordinates, connectivity, and number of springs, number of element, boundary conditions, and loading and material information. The processing stage involves stiffness matrix generation, global stiffness matrix formulation and solution of equations to obtain displacement in elements are evaluated at this stage. The postprocessing stage deals with the presentation of results. A complete applied element analysis is a logical interaction of the three stages. The preparation of data and postprocessing requires considerable effort if all data are to be handled manually. Computer program for analysis of 1-D problems and 2-D problems are developed. Programming of 1D element involves axially loaded column. The program gives nodal displacements as output. Likewise 2D element program is able to solve problems of cantilever beam, deep beam, 2D frame..etc.

4.2 Structure of Applied Element Analysis Program

The program starts with INPUT information to define the problem. Then the data given is processed and required result is obtained. The flow chart of the program is as shown in Fig4.1. The application of program in solving various problems is discussed in Chapter 5. The steps to be followed are the following:

4.2.1 Input Information

In Applied Element Method (AEM) elements are connected by means of springs at contact faces of element. Connectivity between elements are at centroid of both adjacent elements hence joint coordinates are generated at centroid of element. Information pertaining to the structure itself must be defined clearly.
The various input required to define a problem may be grouped into the following:

- **Geometric Data**:

  a. General information like total number of elements, total number of nodal points, type of element (number of nodes, degrees of freedom for each node), are to be supplied.

  b. Coordinates of each node to be supplied or generated.

  c. For each element nodal connectivity is to be supplied.

- **Material Properties**: The total number of materials used, and for each material material properties like Young's Modulus, Poisson's ratio...etc are defined.

- **Load Data**: It consists of total types of loads and for each load its magnitude, point of application (coordinate or line or surface of application) etc.

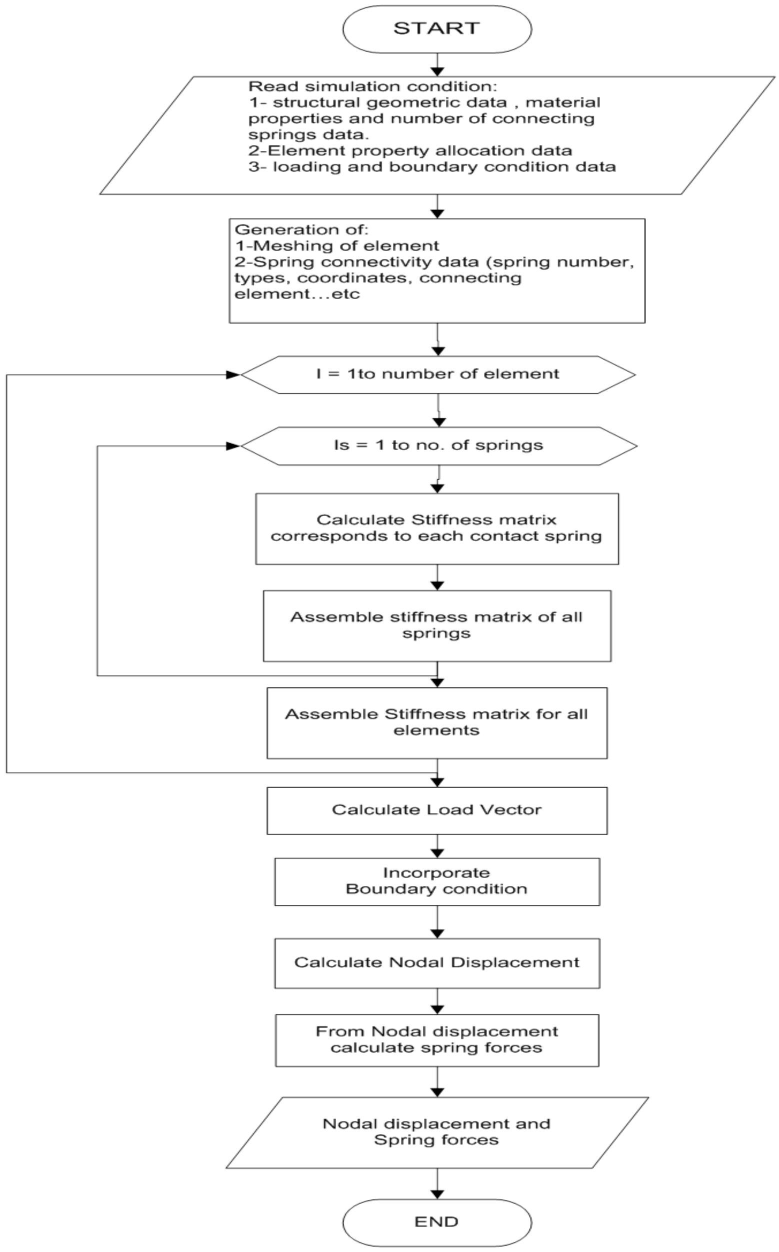


Figure 4.1: Flow chart of AEM program

- **Restraint condition**: The next step is to define the boundary conditions. Restraint conditions are defined at nodes corresponding to degree of freedom at that node. Input required is about total number of boundary conditions and for each boundary condition specified displacements. For preparation of input data file a separate program is developed. The Program carry out meshing of the geometry and prepare data related to number of nodes, elements, element connectivity, nodal coordinates, restraint conditions and joint loads.

4.2.2 Data processing

Data supplied is processed as follows to complete the analysis.

- **Construction of Stiffness Matrix**: The stiffness matrix is an inherent property of the structure and is based upon the structural data only. Element stiffness matrix is first initialized. Then element loop is entered to assemble element stiffness matrix. It starts with initializing global stiffness matrix and load matrix. The depending on number of springs stiffness matrix is obtained by summation of contribution of all connected springs. With the help of nodal connectivity details, the position of each value of element stiffness matrix in global matrix is identified and added to existing value. When element loop is completed global stiffness matrix is available.

- **Identification of Load Data** : All loads acting on the structure must be specified in a manner suitable for computer programming. Loads acting on member are converted into nodal load. Loads are directly applied to the nodes, no member loads are considered as members are assumed to be connected virtually by means of springs. Using load details nodal loads are to be assembled. The equivalent joint loads may then be added to construct total joint loads to produce a problem in which the structure is imagined to be loaded. All nodal values are summed up to get final load vector {F}. After, the stiffness equations are ready, Gauss Jordan elimination solution is used to solve the equation to get nodal variables. In the final phase of the analysis all of the joint displacements in elements are computed.

4.2.3 Postprocessing

Based on solution of equilibrium equations deformed shapes of members are drawn with obtained results. From the nodal displacements deformation of springs are obtained. Based on spring stiffness and spring deformation , forces in springs are obtained. Forces in springs represents stresses in element. The flow of analysis listed above constitute an orderly approach having certain essential features that are advantageous when dealing with large complicated frameworks. Listing of following programs are included in Appendix A.

a. Meshing of continuum.

b. Analysis of 2-D programs.

4.3 Summary

In this chapter development of computer program of AEM using C-language is discussed. Three major stages of program:Preprocessing, Processing and Post processing is discussed. The application of computer program is discussed in next chapter.

Chapter 5

Application of Applied Element Method

5.1 General

Theory and methodology for application of applied element method (AEM) is discussed in chapter 3. Computer implementation of AEM is discussed in Chapter 5. Static linear analysis using AEM is illustrated in this chapter. One dimensional and two dimensional structural engineering problems are solved using computer program developed in Chapter 4. The problems are solved considering varying number of elements and number of springs. The analysis results are compared with results obtained from finite element analysis.

5.2 One Dimensional Problem

A structural engineering problem is considered as one dimensional, if following conditions are satisfied.

a. When its dimension along major axis (length) is much higher as compared to cross sectional dimensions.

b. Direction of applied force and deformation of member is in plane along major axis of a member.

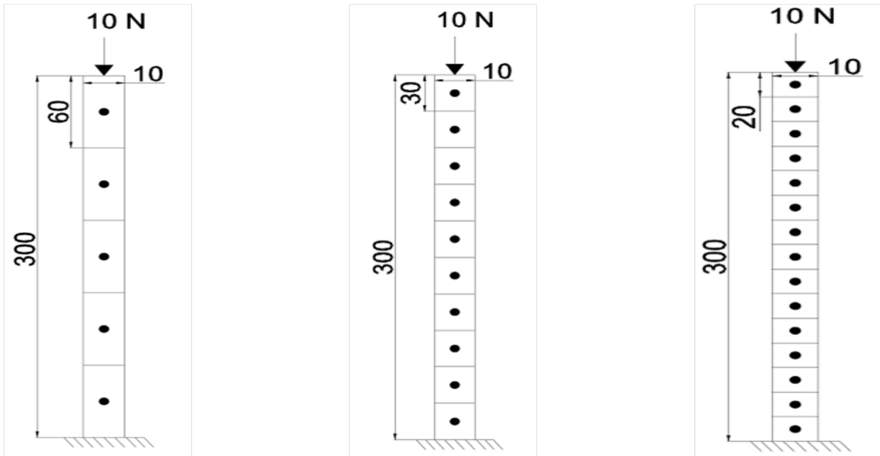
Examples of one dimensional problems are bar, axially loaded column.. etc. Axially loaded column is considered to illustrate application of AEM for one dimensional problem.

5.2.1 Axially Loaded Column

Geometrical data,material properties, restraint conditions and type of loading considered for analysis are given below.
Length of column = 300.00 mm
Width of column = 10.00 mm
Thickness of column = 10.00 mm
Modulus of Elasticity = 100.00 N/mm^2
Load case 1: Point load of 10 N at top.
Load case 2 : Uniformly distributed load of 10 N/mm.

The column is divided into various number of elements like 5, 10 and 15 as shown in Fig5.1, to understand the effect of element size on analysis results. All elements are assumed to be connected by 5 springs. As problem is one dimensional, only axial spring is considered for connecting elements.

Analysis results in terms of displacements at nodes are obtained for for each case of discretization. Two types of loads are applied for analysis as mentioned above.



(a) 5 Elements (b) 10 Elements (c) 15 Elements

Figure 5.1: Discretization of column into number of elements

Case (1): Axially loaded column subjected to point load at top with different number of nodes is as shown in Fig5.1. Table5.1, 5.2 and 5.3 shows the displacement at nodes considering 5,10 and 15 elements for column respectively. Fig5.2 shows graphical comparison of the displacement results. Results are also compared with that obtained by finite element method.

Table 5.1: Nodal displacement considering 5 Elements

Element no.	**Dist.**(mm)	**Disp.**(mm)
1	270	2.4
2	210	1.8
3	150	1.2
4	90	0.6
5	30	0.0

Table 5.2: Nodal displacement considering 10 Elements

Element no.	**Dist.** (mm)	**Disp.** (mm)	**Element no.**	**Dist.** (mm)	**Disp.** (mm)
1	285	2.7	6	135	1.2
2	255	2.4	7	105	0.9
3	225	2.1	8	75	0.6
4	195	1.8	9	45	0.3
5	165	1.5	10	15	0.0

Table 5.3: Nodal displacement considering 15 Elements

Element no.	**Dist.** (mm)	**Disp.** (mm)	**Element no.**	**Dist.** (mm)	**Disp.** (mm)
1	290	2.8	8	150	1.4
2	270	2.6	9	130	1.2
3	250	2.4	10	110	1.0
4	230	2.2	11	90	0.8
5	210	2.0	12	70	0.6
6	190	1.8	13	50	0.4
7	170	1.6	14	30	0.2
			15	10	0.0

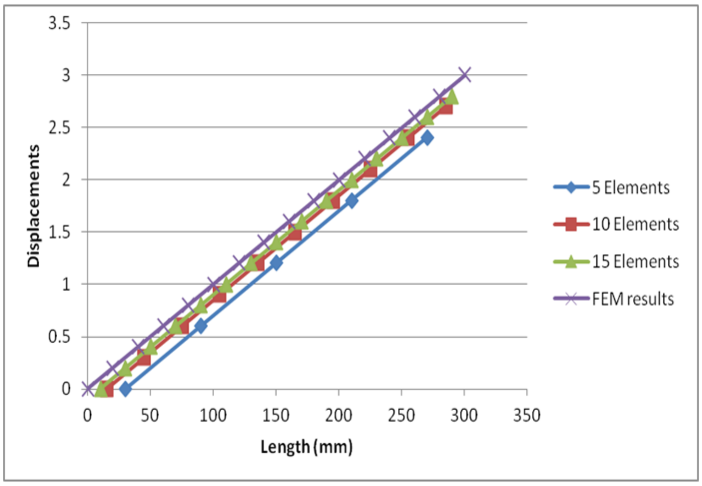


Figure 5.2: Displacement results for axially loaded column subjected to point load.

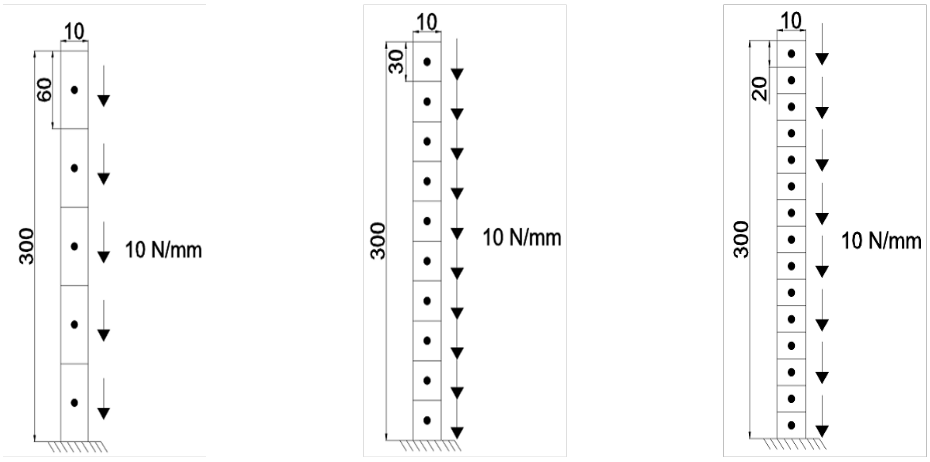
The maximum displacement obtained at top of column with different discretization is shown in Table5.4.

Table 5.4: Maximum displacement in column with varying number of elements

No. of Elements	Maximum displacement (mm)
5	2.4
10	2.7
15	2.8
Exact	3.0

It is observed that with increase in number of elements maximum displacement tends to exact displacement.

Case 2: Axially loaded column subjected to uniformly distributed load with different number of nodes is shown in Fig5.3. Uniformly distributed load is converted in to equivalent nodal load and analysis is carried out. Nodal displacement results are as shown in Table 5.5, 5.6, 5.7. The deflection along the height with different no. of elements and its comparison with that of finite element method are shown in Fig5.4.



(a) 5 Elements (b) 10 Elements (c) 15 Elements

Figure 5.3: Discretization of column into number of elements

Table 5.5: Nodal displacement considering 5 Elements

Element no.	Dist(mm)	Disp.(mm)
1	270	36.0
2	210	32.4
3	150	25.2
4	90	14.4
5	30	0.0

Table 5.6: Nodal displacement considering 10 Elements

Element no.	Dist. (mm)	Disp. (mm)	Element no.	Dist. (mm)	Disp. (mm)
1	285	40.5	6	135	27.0
2	255	39.6	7	105	21.6
3	225	37.8	8	75	15.3
4	195	35.1	9	45	8.1
5	165	31.5	10	15	0.0

Table 5.7: Nodal displacement considering 15 Elements

Element no.	**Dist.** (mm)	**Disp.** (mm)	**Element no.**	**Dist.** (mm)	**Disp.** (mm)
1	290	42.0	8	150	30.8
2	270	41.6	9	130	27.6
3	250	40.8	10	110	24.0
4	230	39.6	11	90	20.0
5	210	38.0	12	70	15.6
6	190	36.0	13	50	10.8
7	170	33.6	14	30	5.6
			15	10	0.0

Table 5.8: Maximum displacement in column with varying number of elements

No. of Elements	Maximum displacement (mm)
5	36
10	40.5
15	42
Exact	45

It is observed from Fig5.4 that with increase in number of elements maximum displacement tends to exact results, as obtained by FEM. The maximum displacement obtained at top of column with different discretization is shown in Table5.4.

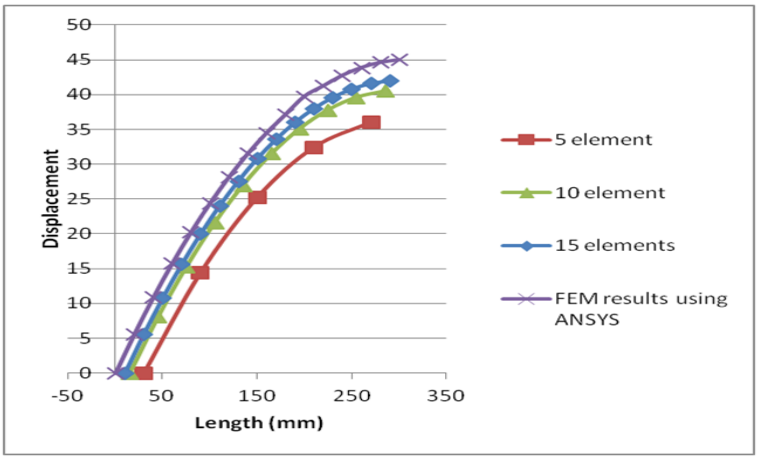


Figure 5.4: Displacement results for axially loaded column subjected to UDL.

5.3 Two Dimensional Problem

Two dimensional problems has three degrees of freedom at each node i.e. translation in horizontal and vertical direction and rotation. Development of stiffness matrix for analysis is discussed in Chapter 3. Application of AEM in two dimensional problems is illustrated by considering cantilever beam, deep beam and portal frames.

5.3.1 Cantilever Beam

Geometrical data, material properties, restraint condition and type of loading considered for analysis are given below:

Length of beam = 3000 mm

Depth of beam = 600 mm

Width of beam = 300 mm

Modulus of Elasticity of concrete E = 25000 N/mm^2 (for M25 grade concrete)

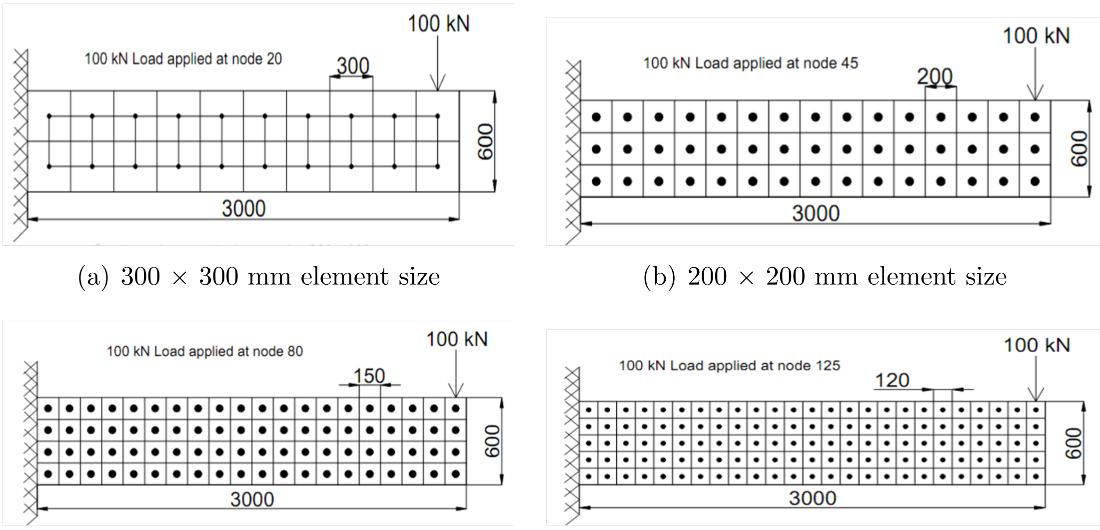
Poission's ratio = 0.15

Shear modulus of material G = 3750 N/mm^2

Moment of inertia (I) of section = 5.4×10^9 mm^4.

Loading : Point load of 100 kN at free end

The beam is divided into varying size of square elements to understand the effect of element size on analysis results. The sizes of elements considered for analysis are 300 mm, 200 mm, 150 mm and 120 mm as shown in Fig5.5. The elements are connected by both normal and shear springs. To understand the effect of number of connecting springs on analysis results the elements are assumed to be connected by 1, 3, 5, 7 and 10 springs (both normal and shear). The point load of 100 kN is applied to node closer to free end, while joints are restraint considered at node closer to fixed support. The discretization of cantilever beam is as shown in Fig5.5.



(a) 300 × 300 mm element size

(b) 200 × 200 mm element size

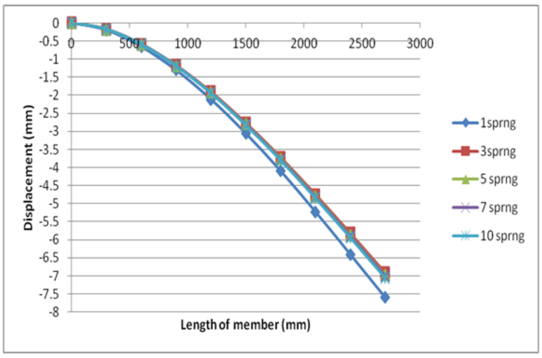
(c) 150 × 150 mm element size

(d) 120 × 120 mm element size

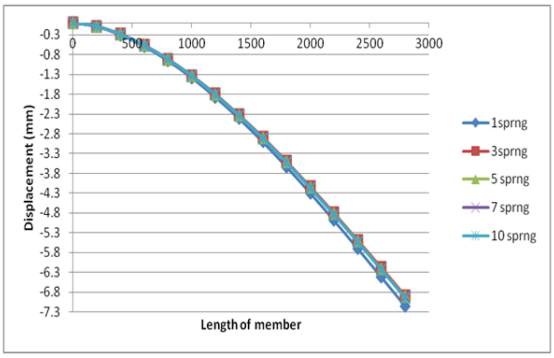
Figure 5.5: Geometry of beam with different discretization

Analysis results in terms of nodal displacement are obtained by using computer program as discussed in Chapter 4. The nodal displacement result of beam with 300 mm, 200 mm, 150 mm and 120 mm size elements and with varying number of connecting springs are as shown in Fig5.6.

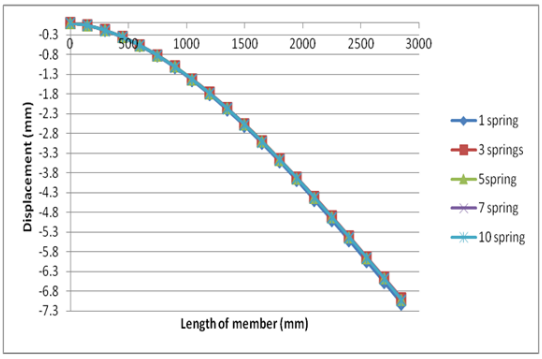
The variation in displacement with 1, 3, 5, 7 and 10 springs considering varying sizes of elements are as shown in Fig5.6.



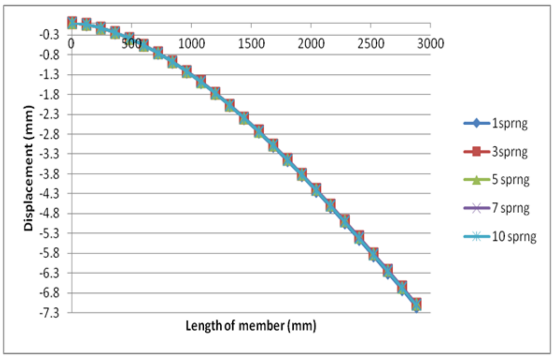
(a) 300 mm Element size



(b) 200 mm Element size



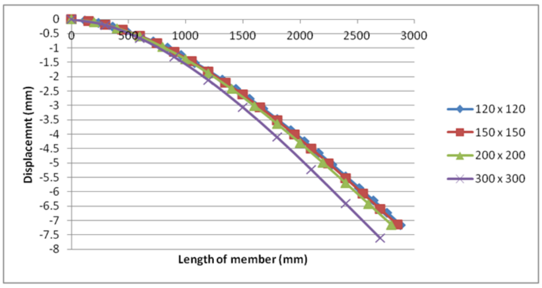
(c) 150 mm Element size



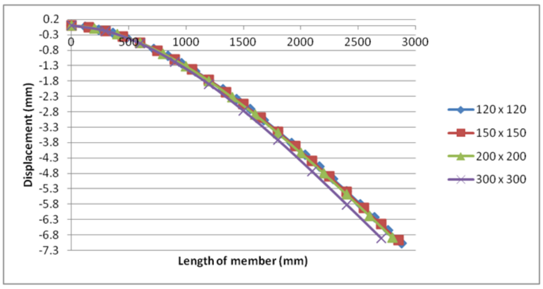
(d) 120 mm Element size

Figure 5.6: Displacement of cantilever beam with varying sizes of elements

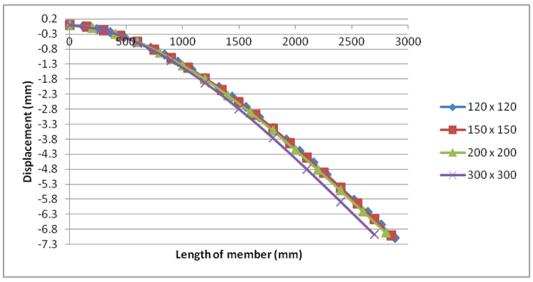
Variation in displacement along the length of beam for varying number of spring between elements are shown in Fig.5.7.



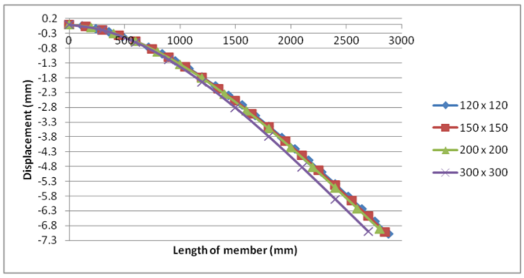
(a) 1 Spring



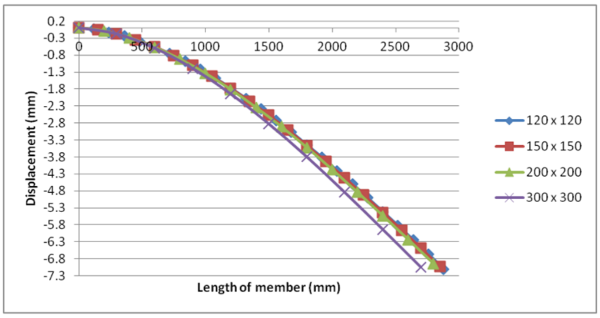
(b) 3 Springs



(c) 5 Springs



(d) 7 Springs



(e) 10 Springs

Figure 5.7: Displacement results for cantilever beam with constant number of springs.

The maximum displacement at free end with varying number of springs and varying size of elements and its comparison with finite element analysis results are as shown in Fig5.8 and Table5.9. Finite element analysis is carried out using ANSYS software.

Table 5.9: Maximum displacement in column with varying number of elements

Size of Element	No. of Springs	Max Displacement (mm)	
		AEM	FEM
	1	7.16	
	3	7.08	
120	5	7.09	7.217
	7	7.08	
	10	7.11	
	1	7.13	
	3	6.98	
150	5	7.01	7.186
	7	7.01	
	10	7.03	
	1	7.15	
	3	6.89	
200	5	6.91	7.142
	7	6.92	
	10	6.95	
	1	7.60	
	3	6.91	
300	5	6.98	7.071
	7	7.00	
	10	7.05	

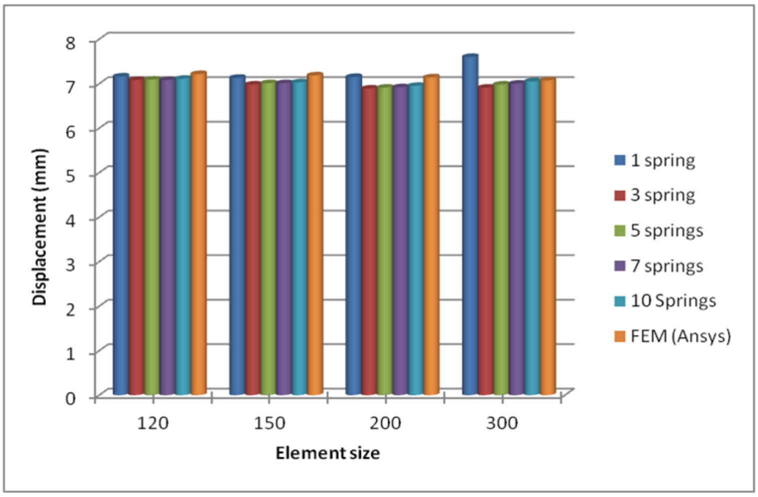


Figure 5.8: Maximum displacement in cantilever beam

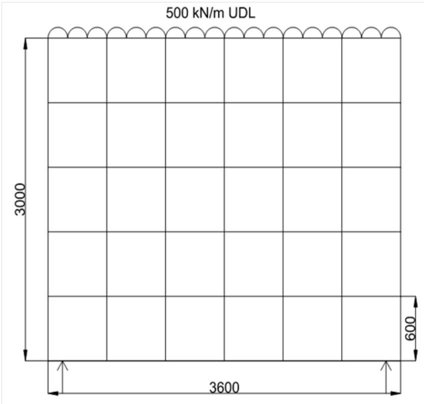
From Table5.9 and Fig5.8, it is observed that with increase in number of spring the displacement reduces. There is not much difference in displacement given by AEM and FEM with same size of element. With larger size of element more numbers of springs gives displacement close to FEM results. But with small sizes of elements less number of springs are adequate.

5.3.2 Solid Deep Beam

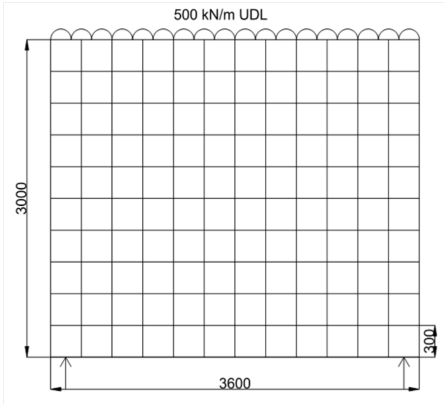
Deep beam is defined as a beam having a ratio of effective span to overall depth less than 2 for simply supported beam as per IS 456-2000. They are used in transfer girder, pile cap, foundation wall, raft beam. wall of rectangular tank and floor diaphragm.

Geometrical data, material properties, restraint condition and type of loading considered for analysis are given below.
Support condition : Simply supported
Span of deep beam (L) = 3600 mm
Depth of deep beam (D) = 3000 mm
L/D ratio = 1.2
Width of deep beam = 300 mm
Modulus of Elasticity of concrete (E) = 25000 N/mm^2 (For M25 grade concrete)
Poission's ratio = 0.15
Shear modulus of material G = 3750 N/mm^2
Loading : 500 kN/m uniformly distributed load.

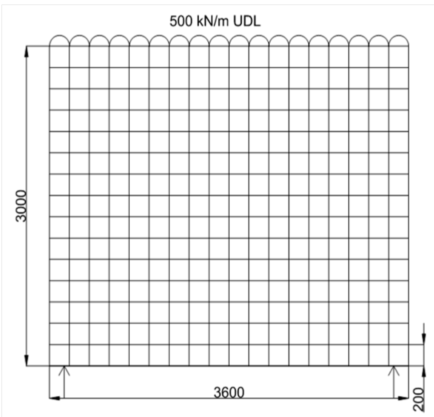
Deep beam is divided into various size of square elements as shown in Fig5.9. Uniformly distributed load is converted into equivalent nodal loads and applied at centroid of top row of square elements. The elements are assumed to be connected by 1, 3, 5, 7 and 10 springs.



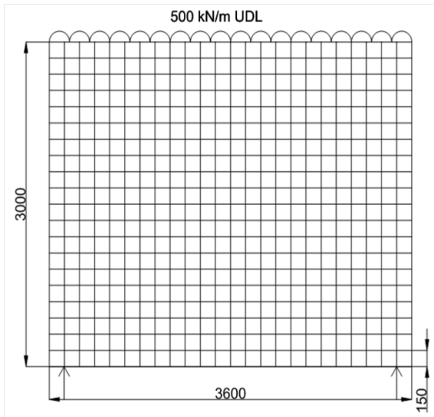
(a) 600 Element size



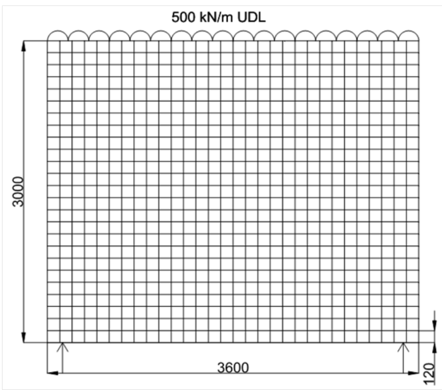
(b) 300 Element size



(c) 200 Element size

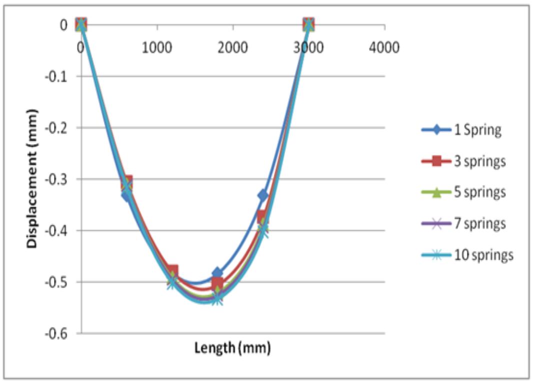


(d) 150 Element size

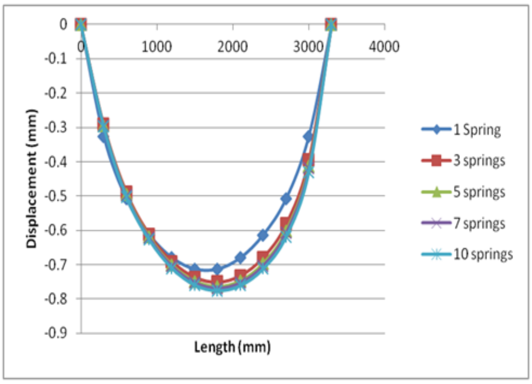


(e) 120 Element size

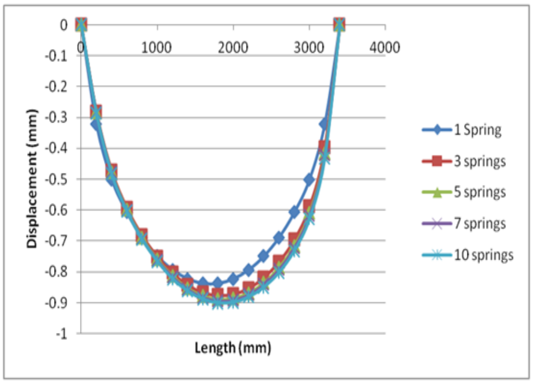
Figure 5.9: Discretization of solid deep beam with different no. of elements



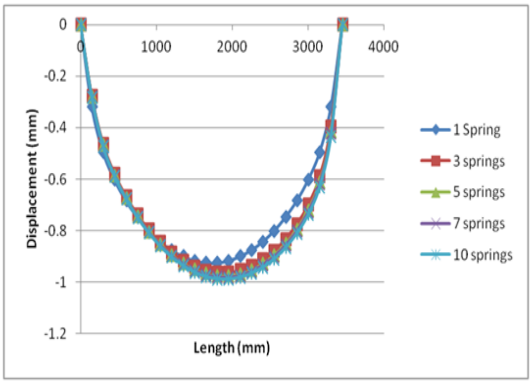
(a) Size of element 600 mm



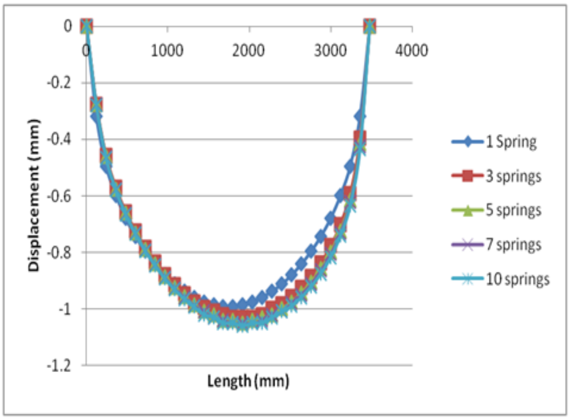
(b) Size of element 300 mm



(c) Size of element 200 mm



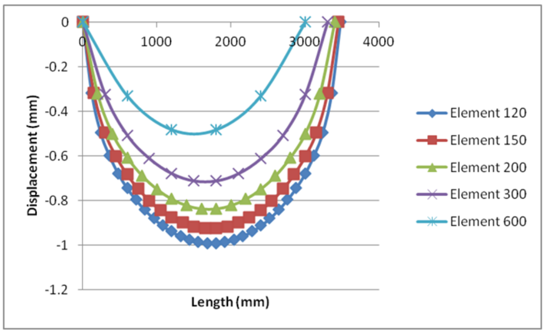
(d) Size of element 150 mm



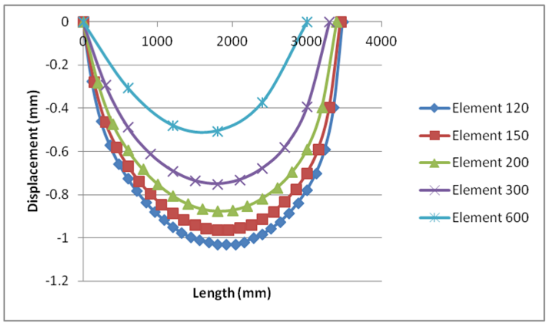
(e) Size of element 120 mm

Figure 5.10: Nodal displacement of simply supported deep beam with AEM.

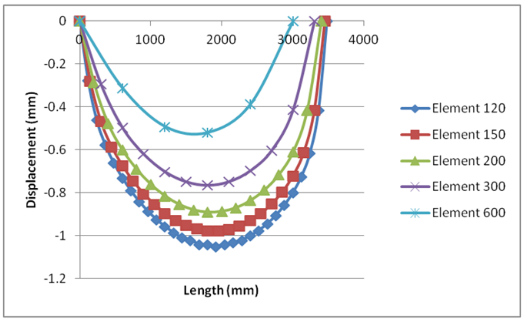
The displacement along the span of beam for varying size of element i.e. 600 mm, 300 mm, 200 mm, 150 mm and 120 mm connected by different number of springs are shown in Fig 5.10(a) to 5.10(e). Effect of increasing in number of spring for different size square element is shown in Fig5.11.



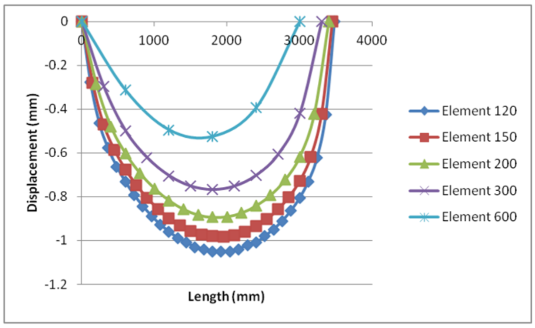
(a) 1 Spring



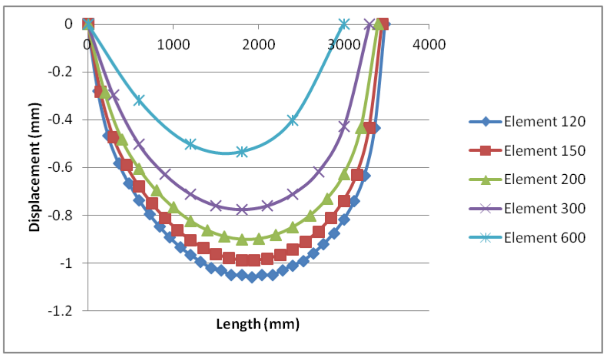
(b) 3 Springs



(c) 5 Springs



(d) 7 Springs



(e) 10 Springs

Figure 5.11: Displacement result for solid deep beam with constant number of springs.

The maximum displacement at joint nearer to the center of beam with varying size of element and varying number of springs and its comparison with FEA results are shown in Table5.10 and Fig5.11.

Table 5.10: Maximum displacement in column with varying number of elements

Size of Element	No. of Springs	Max Displacement (mm)	
		AEM	FEM
120	1	1.090	1.612
	3	1.120	
	5	1.130	
	7	1.140	
	10	1.150	
150	1	1.020	1.553
	3	1.060	
	5	1.070	
	7	1.080	
	10	1.090	
200	1	0.932	1.477
	3	0.963	
	5	0.976	
	7	0.980	
	10	0.996	
300	1	0.804	1.373
	3	0.830	
	5	0.843	
	7	0.847	
	10	0.854	
600	1	0.562	1.200
	3	0.571	
	5	0.584	
	7	0.588	
	10	0.595	

From Table5.9 and Fig5.12, it is observed that the finite element analysis gives higher displacement as compared to AEM. With reduction in size of elements the difference in FEM and AEM result is reducing. With increase in number of spring for particular size of element there is not much variation in displacement. Smaller size of element with more number of spring gives better accuracy of solution for solid deep beam considered here.

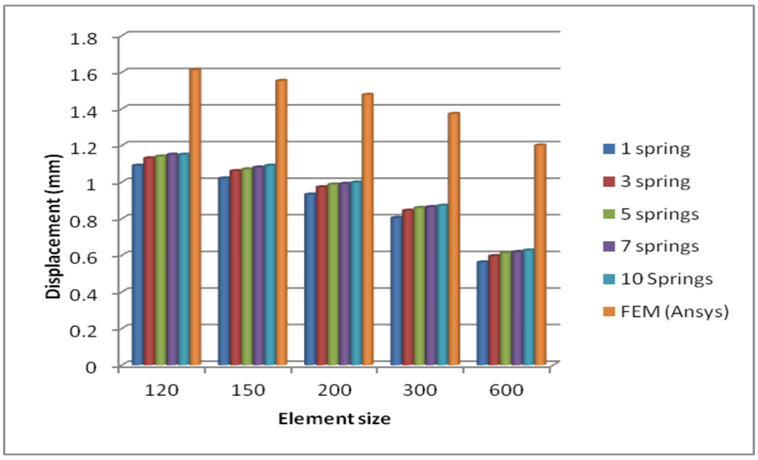
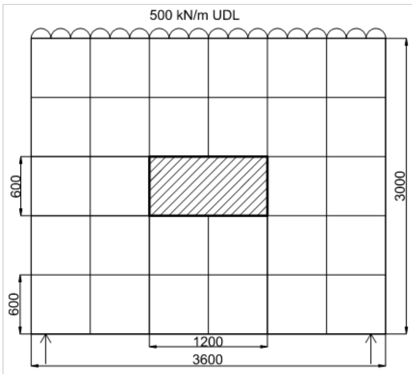


Figure 5.12: Comparison of maximum displacement of solid deep beam.

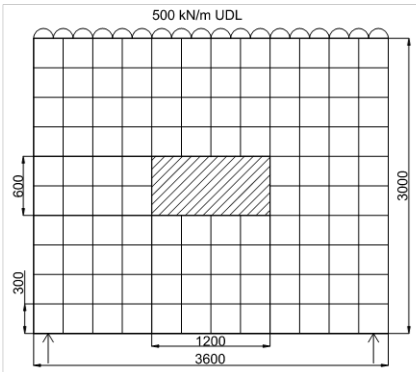
5.3.3 Deep Beam with Opening

Geometrical data, Loading, Material properties and Support condition are same as considered for solid deep beam in Section 5.3.2. In this study one opening of dimension 1200 $mm \times 600$ mm is provided at the center of deep beam as shown in Fig5.13. Different element sizes used for discretizing the deep beam with opening are shown in Fig5.13.

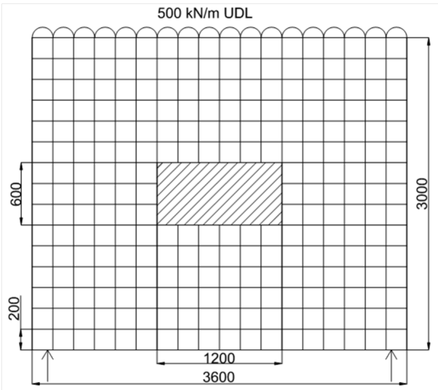
Deep beam with opening is divided into various size of element as shown in Fig.5.13. Deep beam is subjected to uniformly distributed load of 500 kN/m at top. This load is converted into equivalent point load and applied at centroid of element in top layer. Effect of opening on displacement of beam is studied using varying number of springs and size of elements.



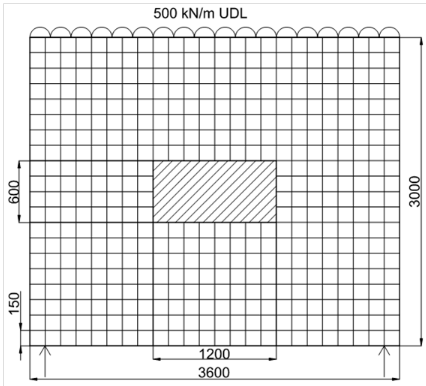
(a) Element size 600 mm



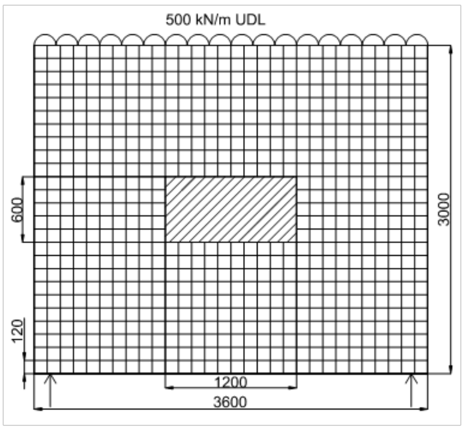
(b) Element size 300 mm



(c) Element size 200 mm

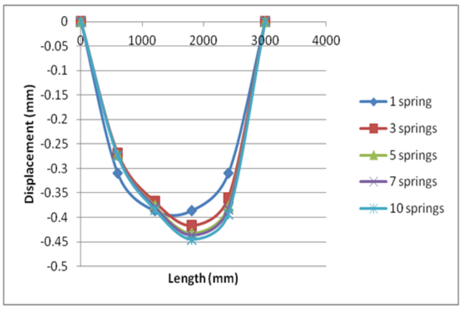


(d) Element size 150 mm

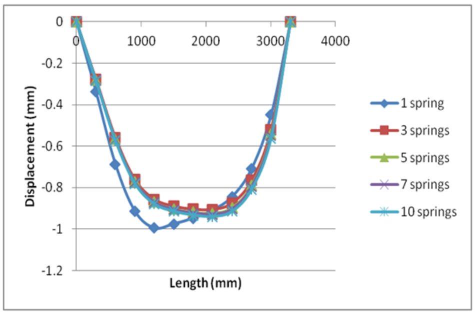


(e) Element size 120 mm

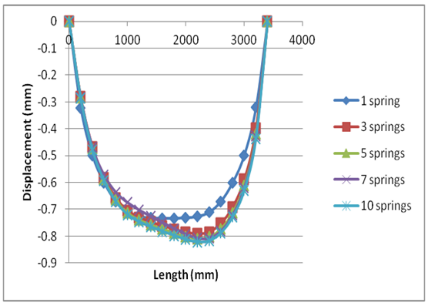
Figure 5.13: Various discretization of deep beam with opening



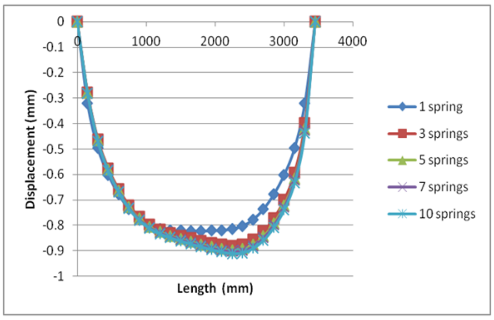
(a) Element size = 600 mm



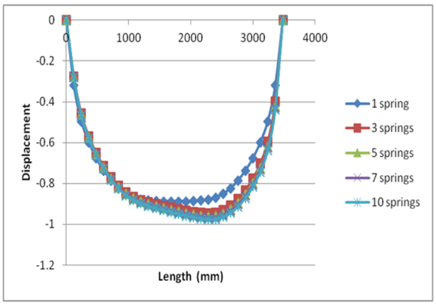
(b) Element size = 300 mm



(c) Element size = 200 mm



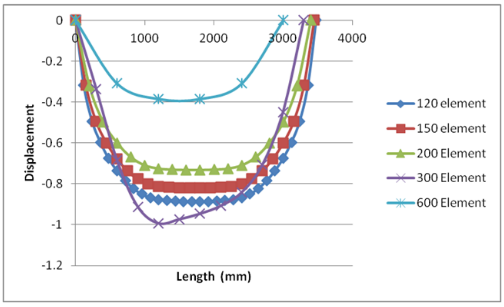
(d) Element size = 150 mm



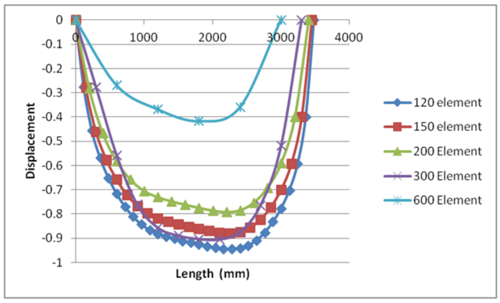
(e) Element size = 120 mm

Figure 5.14: Displacement of deep beam with opening by varying no. of springs

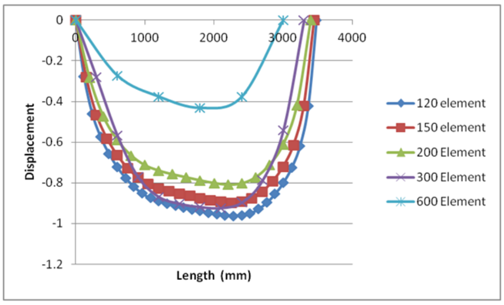
Displacements along the span of beam for varying size of element are obtained as shown in Fig5.14. Presence of opening in deep beam increases the central deflection of beam. Effect of increasing number of springs for different size square element is shown in Fig 5.15.



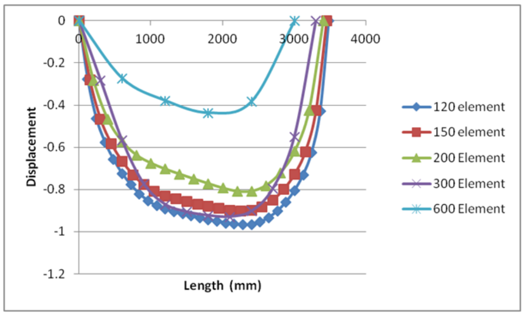
(a) 1 Spring



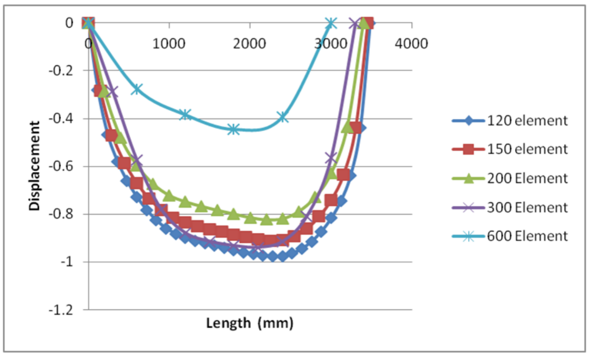
(b) 3 Springs



(c) 5 Springs



(d) 7 Springs



(e) 10 Springs

Figure 5.15: Displacement of deep beam with opening by varying size of elements

The maximum displacement near the centre of span for deep beam with opening for varying number of contact springs and size of elements and its comparison with FEA results are shown in Fig5.16 and Table5.11.

Table 5.11: Maximum displacement in column with varying number of elements

Size of Element	No. of Springs	Max Displacement (mm)	
		AEM	FEM
120	1	1.260	1.681
	3	1.290	
	5	1.300	
	7	1.310	
	10	1.310	
150	1	1.200	1.605
	3	1.230	
	5	1.250	
	7	1.250	
	10	1.260	
200	1	1.120	1.506
	3	1.140	
	5	1.160	
	7	1.200	
	10	1.170	
300	1	1.170	1.378
	3	1.580	
	5	1.610	
	7	1.620	
	10	1.630	
600	1	0.796	1.211
	3	0.765	
	5	0.786	
	7	0.793	
	10	0.805	

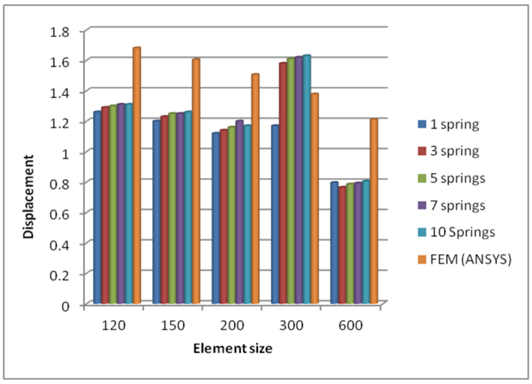


Figure 5.16: Maximum displacement in deep beam with opening

From Table5.11 and Fig5.16, it is observed that finite element analysis gives higher displacement compared to AEM. The difference in AEM and FEM result is almost same with different sizes of element. For particular size of element, variation in number of spring has no significant effect on displacement.

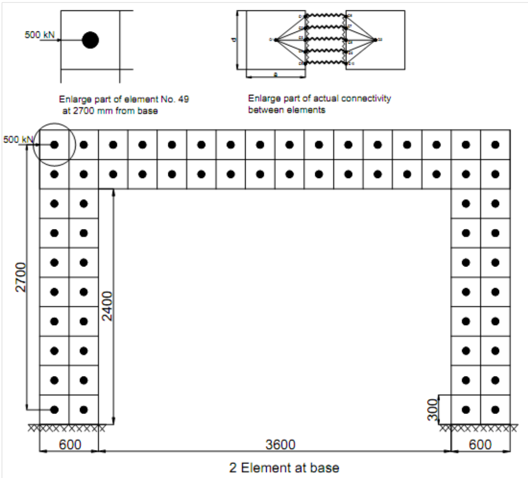
5.3.4 Portal Frame Subjected to Lateral Load

In this section, the application of Applied Element Method for analysis of portal frame subjected to lateral load and combined lateral and vertical load is discussed.

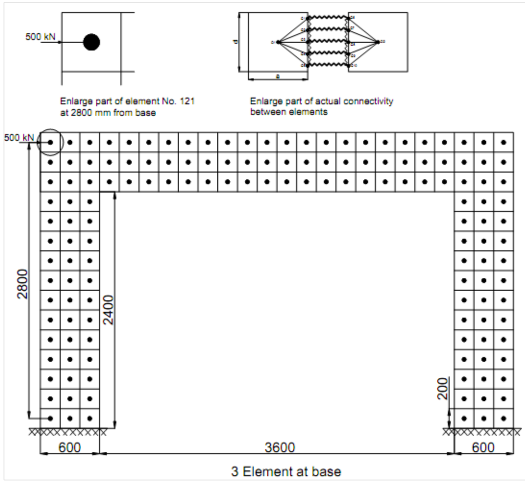
Portal frame subjected to lateral load have been analysed using Applied Element Analysis. Geometrical data Material properties, End conditions and type of loading conditions are as follows:

Column spacing = 4200 mm c/c
Storey height = 3000 mm
Width of beam and column = 300 mm
Depth of beam and column = 600 mm
Cross section area of beam and column = 180×10^3 mm^2.
Modulus of Elasticity of concrete E = 25000 N/mm^2 (for M25 grade of concrete)
Poission's ratio = 0.15
Shear modulus of material G = 3750 N/mm^2
Moment of inertia (I) of section = 5.4 x 10^9 mm^4.
Support condition = Fixed at column base.
Loading : Point load of 500 kN at top in lateral direction.

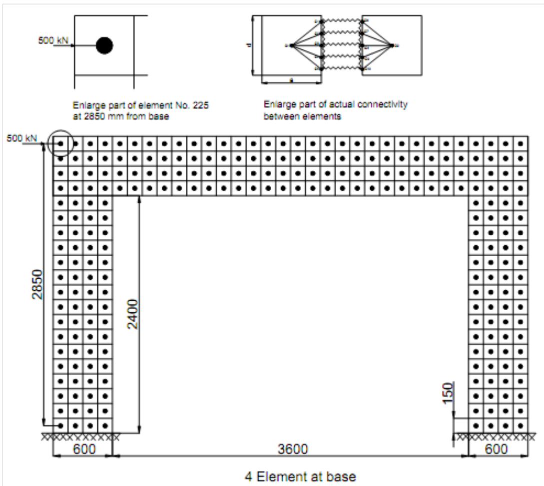
The Frame is divided into varying size of square elements, to understand the effect of element size on analysis results. The sizes of elements considered for analysis are 300 mm, 200 mm, 150 mm and 120 mm as shown in Fig5.17. To understand the effect of number of connecting springs on analysis results the elements are assumed to be connected by 1, 3, 5, 7 and 10 number of springs. The discretization of portal frame is as shown in Fig5.17.



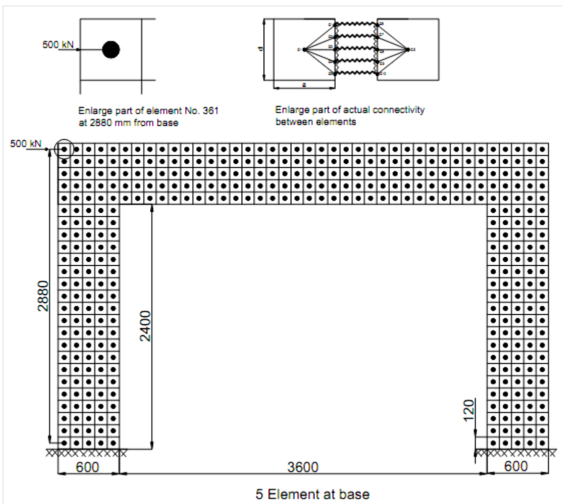
(a) 2 Element at base



(b) 3 Elements at base

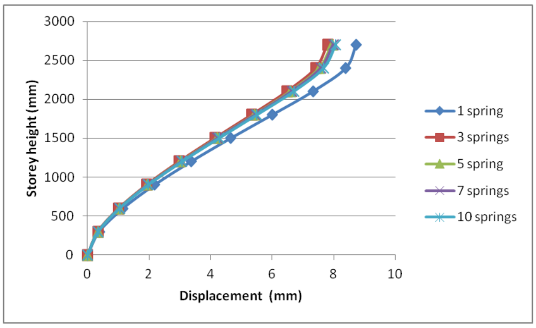


(c) 4 Elements at base

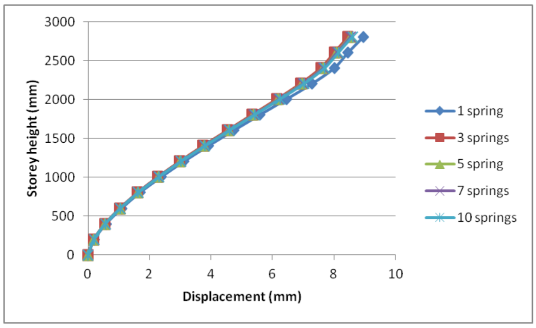


(d) 5 Elements at base

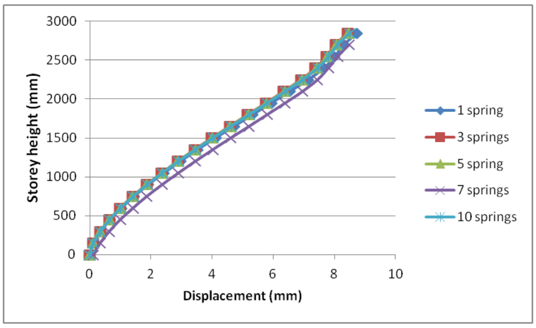
Figure 5.17: Various discretization of portal frame subjected to lateral load.



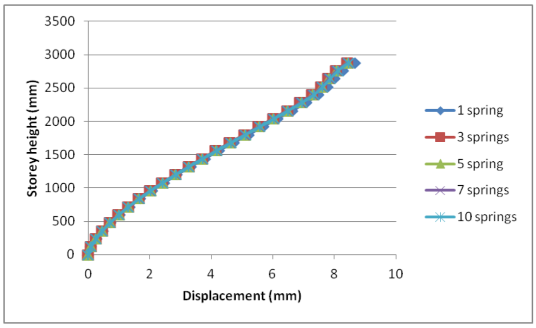
(a) 300 mm Element size



(b) 200 mm Element size



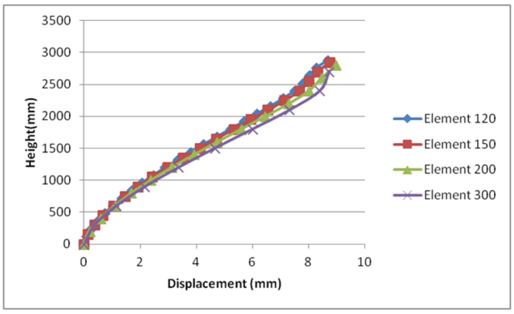
(c) 150 mm Element size



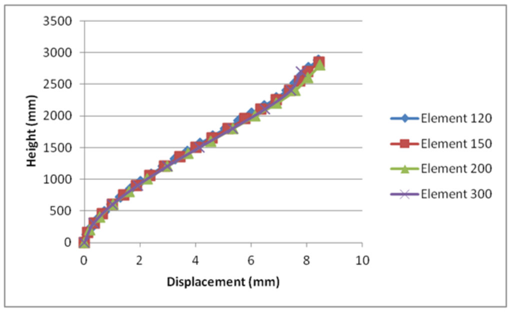
(d) 120 mm Element size

Figure 5.18: Comparison of displacement by varying no. of springs

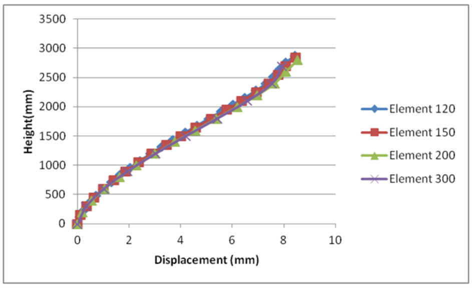
Lateral displacement along the height of portal frame from varying size of elements i.e. 300 mm, 200 mm, 150 mm and 120 mm connected by different no. of springs are as shown in Fig5.18. Effects of varying numbers of spring for different square size element are shown in Fig5.19.



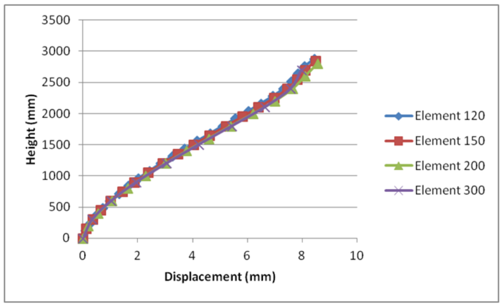
(a) 1 Spring



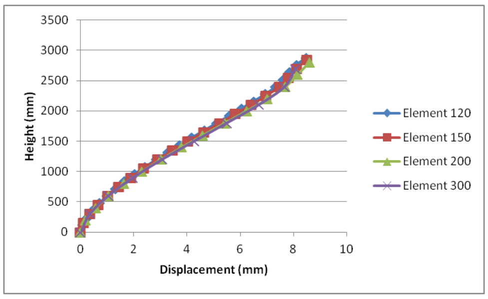
(b) 3 Springs



(c) 5 Springs



(d) 7 Springs



(e) 10 Springs

Figure 5.19: Comparison of displacement by varying size of element

The maximum displacement at node near to top of frame with varying size of element and with varying no. of spring and its comparison with FEM results are shown in Table5.12 and Fig5.20.

From Table5.12 and Fig5.20, it is observed that with larger size of square elements the difference in FEM and AEM result is more. With reduction in size of element AEM results tend to FEM result. The effect of increasing number of contact springs for a particular size of element is not significant. The small size of element with less number of springs represent the behaviour of portal frame accurately.

Table 5.12: Comparison of maximum displacement in frame with varying number of elements

Size of Element	No. of Springs	Max Displacement (mm)	
		AEM	FEM
120	1	8.69	8.61
	3	8.42	
	5	8.45	
	7	8.44	
	10	8.48	
150	1	8.75	8.455
	3	8.43	
	5	8.46	
	7	8.49	
	10	8.51	
200	1	8.96	8.246
	3	8.46	
	5	8.53	
	7	8.55	
	10	8.60	
300	1	8.73	7.922
	3	7.80	
	5	7.94	
	7	7.98	
	10	8.07	

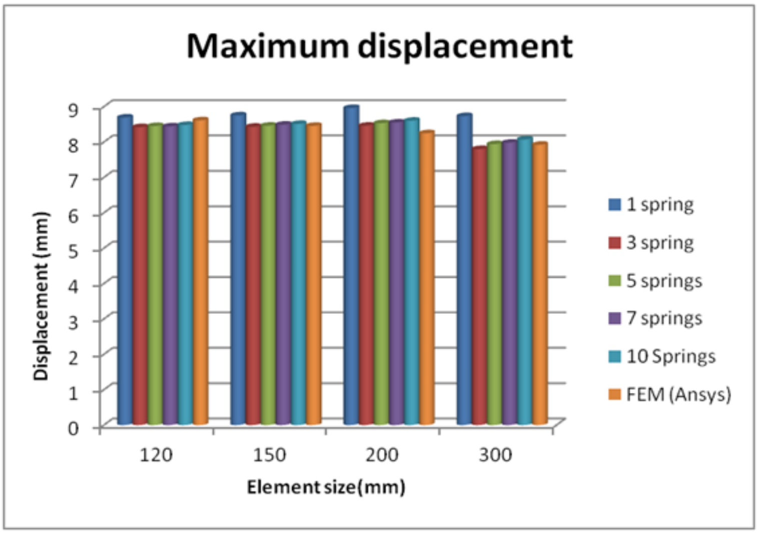


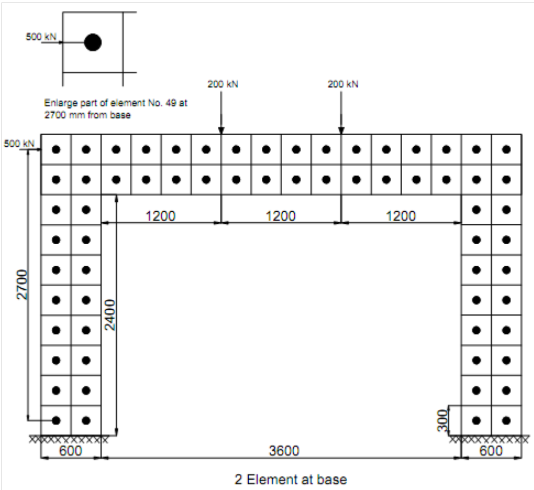
Figure 5.20: Comparison of maximum displacement of frame subjected to lateral load

5.3.5 Portal Frame Subjected to Combined Loading

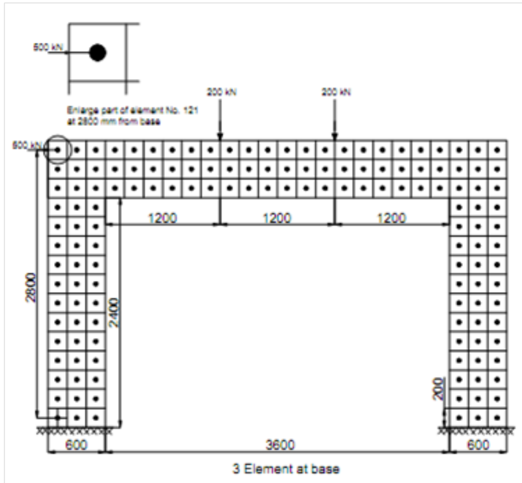
Geometrical data, Discretization, Material properties and end conditions are as shown in Fig5.21.

Column spacing = 4200 mm c/c
Storey height = 3000 mm
Width of beam and column = 300 mm
Depth of beam and column = 600 mm
Cross section area of beam and column = 180 $\times$ 10^3 mm^2.
Modulus of Elasticity of concrete E = 25000 N/mm^2 (for M25 grade of concrete)
Poission's ratio = 0.15
Shear modulus of material G = 3750 N/mm^2
Moment of inertia (I) of section = 5.4 x 10^9 mm^4.
Support condition = Fixed at column base.
Loading : 500 kN in lateral direction and two point loads of 200 kN in vertical direction.

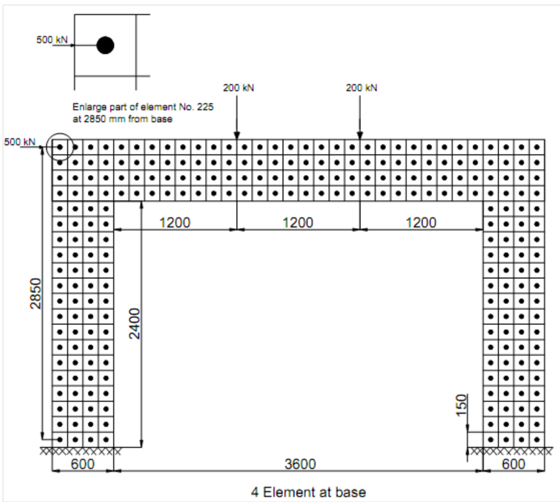
The Frame is divided into varying size of square elements, to understand the effect of element size on analysis results. The sizes of elements considered for analysis are 300 mm, 200 mm, 150 mm and 120 mm as shown in Fig5.21. To understand the effect of number of connecting springs on analysis results the elements are assumed to be connected by 1, 3, 5, 7 and 10 number of springs. The discretization of portal frame is as shown in Fig5.21.



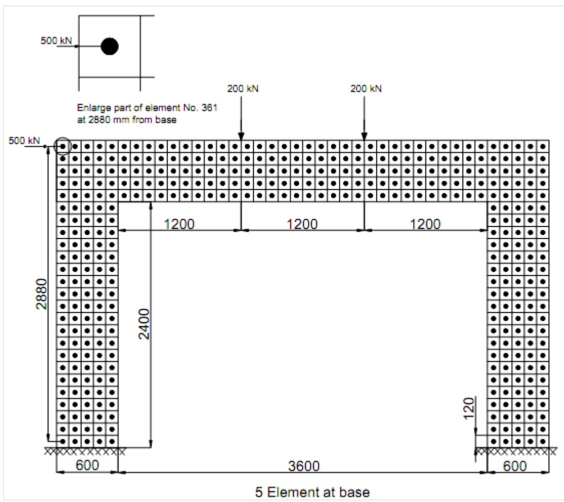
(a) 2 Element at base



(b) 3 Elements at base



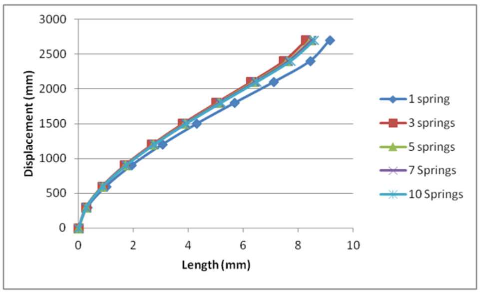
(c) 4 Elements at base



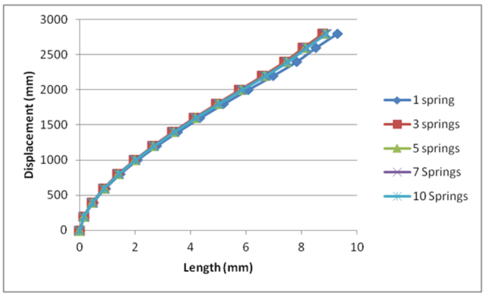
(d) 5 Elements at base

Figure 5.21: Various meshing patterns of member for frame subjected to combine loading

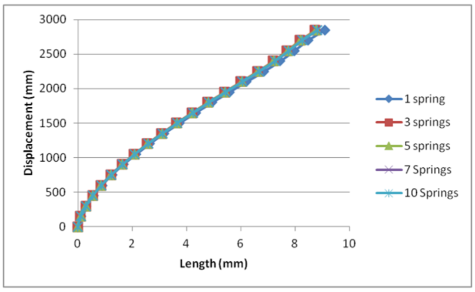
Lateral displacement along the height for varying size of elements i.e. 300 mm, 200 mm, 150 mm and 120 mm by varying number of springs are shown in Fig5.22.



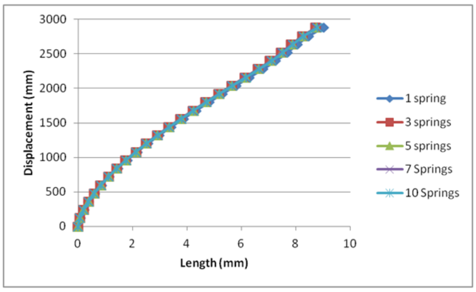
(a) 300 mm Element size



(b) 200 mm Element size



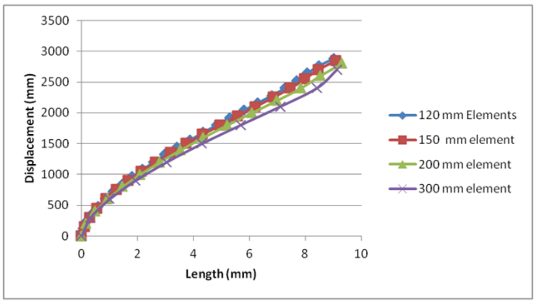
(c) 150 mm Element size



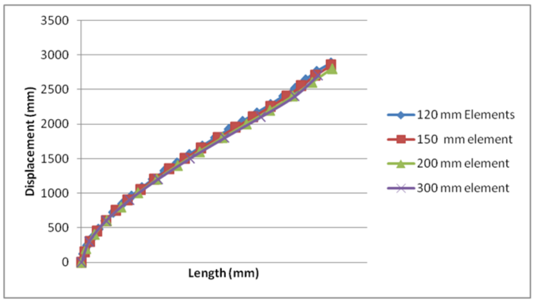
(d) 120 mm Element size

Figure 5.22: Displacement of portal frame subjected to combine loading with varying sizes of elements.

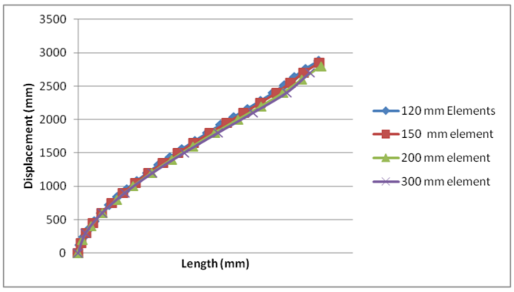
The effect of increase in number of spring for different size of elements are as shown in Fig.5.23.



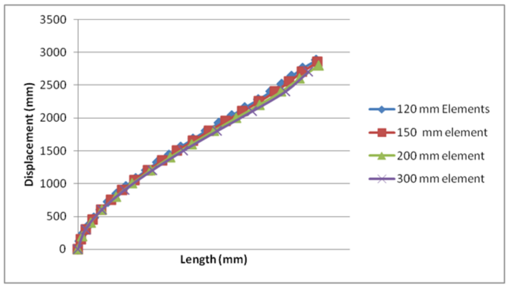
(a) 1 Spring



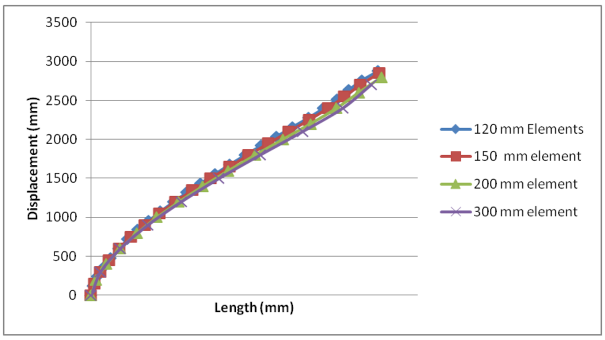
(b) 3 Springs



(c) 5 Springs



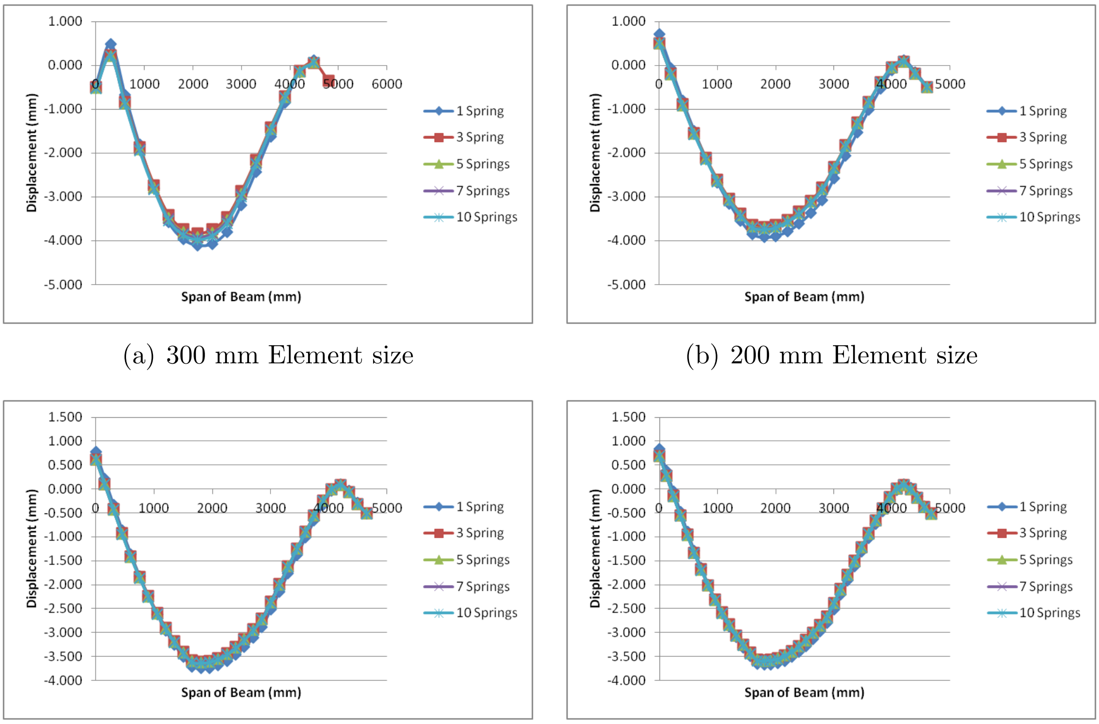
(d) 7 Springs



(e) 10 Springs

Figure 5.23: Displacement of portal frame subjected to combine loading with varying number of springs.

Vertical displacement along the span of beam in portal frame for varying size of elements i.e. 300 mm, 200 mm, 150 mm and 120 mm by varying number of springs are shown in Fig5.24.



(a) 300 mm Element size

(b) 200 mm Element size

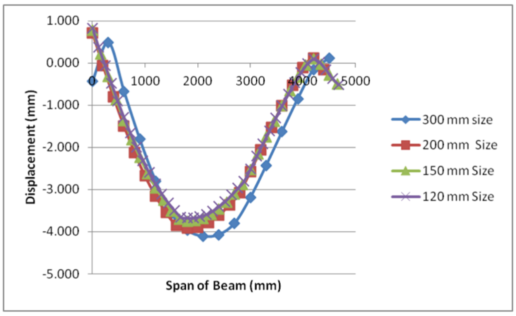
(c) 150 mm Element size

(d) 120 mm Element size

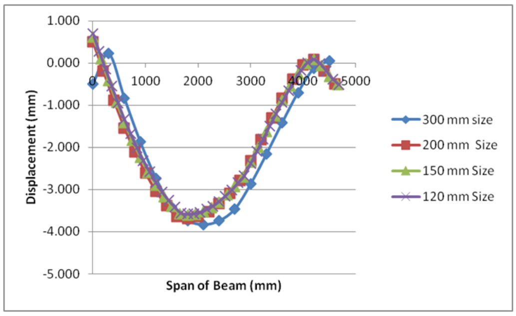
Figure 5.24: Displacement of portal frame subjected to combine loading with varying sizes of elements.

The effect of increase in number of spring for different size of elements are as shown in Fig.5.25.

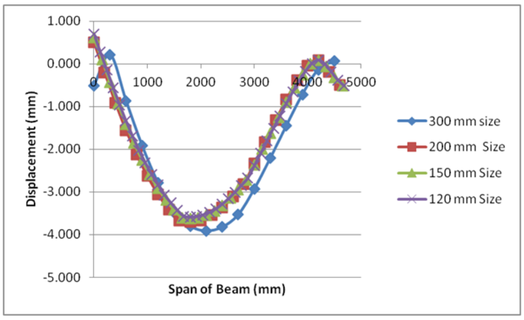
The maximum displacement along the height of portal frame under combine loading with varying size of element and varying number of springs and its comparison with FEM are as shown in Table5.13 and Fig.5.26.



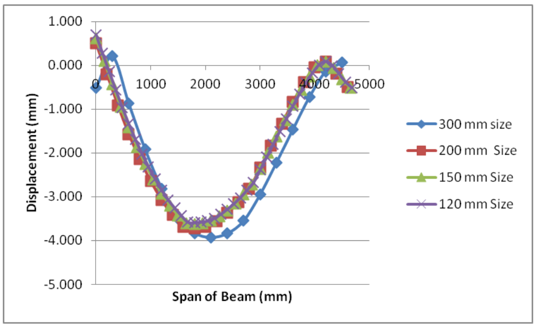
(a) 1 Spring



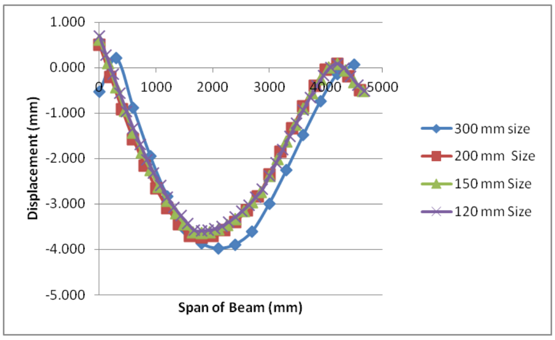
(b) 3 Springs



(c) 5 Springs



(d) 7 Springs



(e) 10 Springs

Figure 5.25: Displacement of portal frame subjected to combine loading with varying number of springs.

Table 5.13: Comparison of maximum displacement of portal frame subjected to combine loading

Size of Element	No. of Springs	Max Displacement (mm) AEM	FEM
120	1	3.65	3.96
	3	3.56	
	5	3.57	
	7	3.58	
	10	3.59	
150	1	3.71	3.92
	3	3.58	
	5	3.6	
	7	3.61	
	10	3.62	
200	1	3.85	3.85
	3	3.64	
	5	3.67	
	7	3.68	
	10	3.7	
300	1	3.8	3.73
	3	3.46	
	5	3.53	
	7	3.55	
	10	3.6	

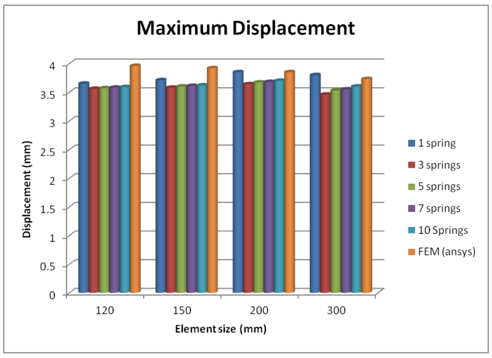


Figure 5.26: Comparison of maximum horizontal displacement for portal frame subjected to combine loading.

5.4 Summary

The application of Applied Element Method (AEM) in linear static analysis of one-dimensional and two-dimensional problems are discussed in this chapter. Axially loaded column is considered as one-dimensional problem and cantilever beam, deep beam and portal frames are considered for two-dimensional problems. The results of AEM are compared with that obtained by FEM.

Chapter 6

Summary and Conclusions

6.1 Summary

In present report introduction to Applied Element Method, its difference with finite element method, methodology used for applied element analysis (AEA) and applications of AEA for linear static analysis of structures are presented.

Introduction and overview of the Applied Element Method (AEM) and its advantages for numerical modelling of large deformation, crack initiation, crack propagation in structure are discussed.

Methodology of applied element analysis is illustrated. The stiffness matrix for one dimensional and two dimensional elements are derived and various factors affecting analysis results of applied element analysis are discussed.

Computer programs using C-language are developed for applied element analysis. Using computer programs various one dimensional and two dimensional problems are solved. One dimensional problem includes column with point load and uniformly distributed load. Two dimensional problems include cantilever beam with point load at free end, solid deep beam and deep beam with opening subjected to uniformly distributed load and portal frame subjected to vertical and lateral load. The variations in results with different size of elements and with different number of connecting springs are observed. The results of applied element method are compared with that obtained by finite element method to check the accuracy of applied element method.

6.2 Conclusions

From study carried out in this project following observations are made:

- The deformation and forces in element are represented by deformation and the forces in springs. Two types of springs i.e. axial and shear, if considered in various directions it can model one-dimensional, two dimensional and three dimensional problems. Elements connected by one axial spring represents one-dimensional problem, elements connected by one shear and one axial spring represents two-dimensional problem and elements connected by two shear and one axial spring represents three-dimensional problem.
- Stiffness matrix can be obtained by applying unit displacement in direction of degrees of freedom and considering forces in spings of each direction. Derivation of stiffness matrix is much simpler compared to that of FEM.
- The variables affecting accuracy of AEM results are : size of element and number of contact springs. Smaller size of element with less number of springs gives accurate results while larger size of element with more number of springs gives less errors in analysis results.
- From solution of various one dimensional and two dimensional problems it is observed that the accuracy of applied element method in analysis results is similar to that of finite element method. The applied element method can be used in place of finite element analysis for linear static analysis.
- The computer program developed for elastic analysis using applied element method can be extended to non linear analysis.

6.3 Future scope of work

The present work can be extended in future to include following aspects:

- In present study program is developed to calculate nodal displacement. In future, stress and strain in various elements can be obtained using nodal displacements, and spring forces.
- Linear static analysis of structural problems is carried out in this study. So, further computer program can be extended for nonlinear analysis also.
- Work can also be extended to calculate failure or cracking load of structure using nonlinear load-deformation relationship.

References

[1] A.S.Dessousky,,"Collapse analysis of stone-blocks structures under seismic excitation using Applied Element Method",Twelfth International Colloquium on Structural and Geotechnical Engineering,10-12 December 2007.

[2] Hamed Salem, "Computer-Aided Design of Framed Reinforced Concrete Structures Subjected to Flood Scouring", Journal of American Science,(ISSN: 1545-1003),2011,pp 191-200.

[3] Harinadha Babu Raparla, "2D Numerical Modeling of Progressive Collapse During Earthquakes:A Case Study on RC Bare Frame", Department of Civil Engineering, Sir CRR College of Engineering Eluru, West Godavari.

[4] Hatem Tagel-Din and Kimiro Meguro, "Applied element simulation for collapse analysis", bulletein of earthquake resistant structure,No.32, 1999, 1-4.

[5] Hatem Tagel-Din and Kimiro Meguro, " Analysis of a small scale RC building subjected to shaking table tests using applied element method", Technical University of Tokyo,Institute of Industrial Science, 1999.

[6] Joshua Wayne Griffin, "Experimental and Analytical Investigation of Progressive Collapse through Demolition Scenarios and Computer Modeling", Raleigh, North Carolina, 2008,pp. 2129.

[7] Kimiro Meguro, "Applied element method: a new efficient tool for design of structure considering its failure behavior", Institution of Industrial Science(IIS), The university of tokyo, Japan, September 2001,pp. 1-20.

[8] Kimiro Meguro and Hatem Tagel-Din, "Applied element simulation of RC structures under cyclic loading", Journal of structural engineering,Vol 127,No.11,November 2001,pg1295-1305.

[9] Kimiro Meguro and Hatem Tagel-Din, "Simulation of buckling and post-buckling behaviour of structure using applied element method", Bulletin,ERS,No.32,1999.

[10] Marin LUPOAE and Carmen BUCUR, " Use of Applied Element Method to simulate the collapse of a building", SISOM 2009 and Session of the Commission of Acoustics, Bucharest, May 2009, pp 28-29.

[11] Meguro K. and Tagel-Din. H. S. Applied Element Method for Structural Analysis: Theory and Application for Linear Materials, Structural Eng./Earthquake Eng., International Journal of the Japan Society of Civil Engineers,(2000), 17(1) 21-35.

[12] Meguro K. and Tagel-Din, H. S. (2002). Applied Element Method Used for Large Displacement Structural Analysis, Journal of Natural Disaster Science, 24 (1), pp. 25-34.

[13] R. Pradeep Kumar and Kimiro Meguro, "Applied element simulation of non-linear behaviour of dip-slip fault for studying ground surface deformation", seismic fault-induced failures, January 2001, pp. 109-114.

[14] Tarsicio Blendez,Cristian Neipp and Augusto Belendez, "Large and small deflection of cantilever beam", Europian journal of physics, vol.23,No.3,ISSN 0143-0807,May 2002,pp 371-379.

Appendix A

List of Useful Websites

- www.sciencedirect.com
- www.asce.com
- www.pdf-search-engine.com
- www.elsevier.com

Appendix B

Source code of Applied Element Method

```
#include<stdio.h>
#include<math.h>
#include<conio.h>
#include<malloc.h>
#define cal(zz,qq) ((qq *) calloc(zz,sizeof(qq)))
float **sff,*df,*aj;
void main()
{
int i,j,k,ns,is,ne,nn,ie,*je,*ke,im[7],nrj,*rj,u,nlj,lj,type;
int *jrl,*kb,c1,nra,n1,*id,i1,i2,ir,ic,item1,nb;
float kn1,ks1,kn,ks,alp,the,len,alp1,the1,len1,sm[7][7],**sj,*ac;
float *e,*g,*x,*y,*th,*dispx,*dispy,*rotz,a, a1, b, b1,ad,d1,gama;
float temp,temp1,load,dm[7],am[7];
float *djg,*aj1;
char ch1[30],ch2[30];
void banfac(int n,int nb);
void bansol(int n,int nb);
FILE *f1,*f2;
```

```
clrscr();
printf(“Enter the name of input file :”);
gets(ch1);
printf(“Enter the name of output file :”);
gets(ch2);
f1 = fopen(ch1,“r”);
f2 = fopen(ch2,“w”);
printf(“Welcome to first AFEM Program ”);
fscanf(f1,“%d %d %d”,&ne,&nn,&ns);
fprintf(f2,“No. of elements = %dNo. of nodes = %dNo.of springs=%d”,ne,nn,ns);
je = cal( (ne + 1) , int );
ke = cal( (ne + 1) , int );
th = cal( (ne + 1) , float );
e = cal( (ne + 1) , float );
g = cal( (ne + 1) , float );
x = cal( (nn + 1) , float );
y = cal( (nn + 1) , float );
jrl = cal( (3*nn + 1) , int );
id = cal( (3*nn + 1) , int );
ac = cal( (3*nn + 1) , float );
aj = cal( (3*nn + 1) , float );
aj1 = cal( (3*nn + 1) , float );
sj = (float **) calloc(3*nn+1,sizeof(float *));
for(i=1;i¡=3*nn;i++)
sj[i] = (float *) calloc(3*nn+1,sizeof(float));
for(i=1;i¡=ne;i++)
{
fscanf(f1,“%d %d %f %f %f”),&je[i],&ke[i],&th[i],&e[i],&g[i]);
fprintf(f2,“Member no.=%d”,i);
fprintf(f2,“j end= %d, k end= %d, thickness= %f ”,je[i],ke[i],th[i]);
```

```
fprintf(f2,”Elasticity= %f, Modulus of rigidity= %f ”,e[i],g[i]);
}
for(i=1;i¡=nn;i++)
{
fscanf(f1,“%f %f”,&x[i],&y[i]);
fprintf(f2,“joint no.=%d, X-co-ordinate=%f Y-co-ordinate=%f ”,i,x[i],y[i]);
}
fscanf(f1,“%d”,&nrj);
rj = cal( (nrj + 1) , int );
dispx = cal( (nrj + 1) , float );
dispy = cal( (nrj + 1) , float );
rotz = cal( (nrj + 1) , float );
fprintf(f2,“No. of restrained joints=%d ”,nrj);
for(i=1;i¡=nrj;i++)
{
fscanf(f1,“%d %f %f %f”,&rj[i],&dispx[i],&dispy[i],&rotz[i]);
fprintf(f2,“joint no=%d,Displacement in x=%f, Displacement in y=%f, Rotation @
z=%f”,rj[i],dispx[i],dispy[i],rotz[i]);
}
fscanf(f1,“%d”,&nrj);
kb = cal( (nrj + 1) , int );
for(i=1;i¡=3*nn;i++)
jrl[i]=0;
fprintf(f2,“NRJ= %d”,nrj);


for(i=1;i¡=nrj;i++)
{
fscanf(f1,“%d”,&kb[i]);
fprintf(f2,“rest joint = %d”,kb[i]);
for(j=1;j¡=3;j++)
```

```
{
fscanf(f1,“%d”,&c1);
fprintf(f2,“c1 = %d”,c1);
if(c1 ! = 1) jrl[3*kb[i]-(3-j)] = 1;
}
}
for(i=1;i¡=3*nn;i++)
fprintf(f2,“jrl[%d]=%d”,i,jrl[i]);
nra=0;
for(i=1;i¡=3*nn;i++)
{
if(jrl[i] ! = 0)continue;
nra = nra + 1;
}
fprintf(f2,“NRA=%d”,nra);
n1 = 0.;
for(i=1;i¡=3*nn;i++)
{
n1 = n1 + jrl[i];
if(jrl[i] > 0)
{
id[i] = nra + n1;
}
else
{
id[i] = i - n1;
}
}
for(i=1;i¡=3*nn;i++)
fprintf(f2,“id[%d]=%3d ”,i,id[i]);
```

```
fprintf(f2,“Restr.Jt.:Restrain X : Restrain Y :Rot z:”);
for(i=1;i¡=nn;i++)
{
fprintf(f2,“%4d — %4d — %4d — %4d —”,i,jrl[3*i-2],jrl[3*i-1],jrl[3*i]);
}
nb = 0;
for(i=1;i¡=ne;i++)
{
k=0;
for(j=1;j¡=3;j++)
{
if(jrl[3*je[i]+1-j]==0)continue;
else k=k+1;
}
for(j=1;j¡=3;j++)
{
if(jrl[3*ke[i]+1-j]==0)continue;
else k=k+1;
}
if(nb¡= (3*(abs(je[i]-ke[i])+1)-k)) nb=3*(abs(je[i]-ke[i])+1)-k;
}

fprintf(f2,“Band Width = %d”,nb);

sff = (float **) calloc(nra+1,sizeof(float *));
for(i=1;i¡=nra;i++)
sff[i] = (float *) calloc(nb+1,sizeof(float));

for(i=1;i¡=nra;i++)
for(j=1;j¡=nb;j++)
```

```
sff[i][j]=0.0;
for(ie=1;ie¡=ne;ie++)
{
for(i=1;i¡=6;i++)
for(j=1;j¡=6;j++)
sm[i][j] = 0.0;
ad=sqrt((x[je[ie]]-x[ke[ie]])*(x[je[ie]]-x[ke[ie]])+(y[je[ie]]-y[ke[ie]])*(y[je[ie]]-y[ke[ie]]));
gama=asin((y[ke[ie]]-y[je[ie]])/(ad))*180.0/3.14159253;
printf("Gamma = %f",gama);
gama=atan((y[je[ie]]-y[ke[ie]])/(x[je[ie]]-x[ke[ie]]))*180.0/3.14159253;
gama = 0.0;
kn1=e[ie]*th[ie]/ad;
ks1=g[ie]*th[ie]/ad;
fprintf(f2,"Element No. %d Length of ele = %f,KN1 = %f,KS1= %f",ie,ad,kn1,ks1);
fprintf(f2,"Inclination of element = %f",gama);
for(is=1;is¡=ns;is++)
{
d1=ad/ns;
kn=kn1*d1;ks=ks1*d1;
len=sqrt((ad/2)*(ad/2)+((ad/2)-((is-0.5)*d1))*(ad/2-((is-0.5)*d1)));
fprintf(f2,"length(%d) = %f",is,len);
alp=asin((ad/2)/len)*180.0/3.14159253;
the=90.0-alp+gama;
fprintf(f2,"alpha(%d) = %f , thetha(%d) = %f",is,alp,is,the);
a = (alp + the) * 3.141592536/180.0;
a1 = alp * 3.141592536/180.0;
alp1=alp;len1=len;the1=90-alp1-gama;
// fprintf(f2,"length1(%d) = %f",is,len1);
// fprintf(f2,"alpha1(%d) = %f , thetha1(%d) = %f",is,alp,is,the);
b = (alp1 + the1) * 3.141592536/180.0;
```

```
b1 = alp1 * 3.141592536/180.0;
sm[1][1] += kn*sin(a)*sin(a) + ks*cos(a)*cos(a);
sm[1][2] += -kn*cos(a)*sin(a) + ks*sin(a)*cos(a);
sm[1][3] += -kn*sin(a)*len*cos(a1) + ks*cos(a)*len*sin(a1);
sm[1][4] += -kn*sin(b)*sin(a) + ks*cos(b)*cos(a);
sm[1][5] += -kn*cos(b)*sin(a) - ks*sin(b)*cos(a);
sm[1][6] += kn*len1*cos(b1)*sin(a) + ks*len1*sin(b1)*cos(a);


sm[2][2] += kn*cos(a)*cos(a) + ks*sin(a)*sin(a);
sm[2][3] += kn*cos(a)*len*cos(a1) + ks*sin(a)*len*sin(a1);
sm[2][4] += kn*sin(b)*cos(a) + ks*cos(b)*sin(a);
sm[2][5] += kn*cos(b)*cos(a) - ks*sin(b)*sin(a);
sm[2][6] += -kn*len1*cos(b1)*cos(a) + ks*len1*sin(b1)*sin(a);
sm[3][3] += kn*len*len*cos(a1)*cos(a1) + ks*len*len*sin(a1)*sin(a1);
sm[3][4] += kn*sin(b)*len*cos(a1) + ks*cos(b)*len*sin(a1);
sm[3][5] += kn*cos(b)*len*cos(a1) - ks*sin(b)*len*sin(a1);
sm[3][6] += -kn*len1*cos(b1)*len*cos(a1) + ks*len1*sin(b1)*len*sin(a1);
sm[4][4] += kn*sin(b)*sin(b) + ks*cos(b)*cos(b);
sm[4][5] += kn*cos(b)*sin(b) - ks*sin(b)*cos(b);
sm[4][6] += -kn*len1*cos(b1)*sin(b) + ks*len1*sin(b1)*cos(b);
sm[5][5] += kn*cos(b)*cos(b) + ks*sin(b)*sin(b);
sm[5][6] += -kn*len1*cos(b1)*cos(b) - ks*len1*sin(b1)*sin(b);
sm[6][6] += kn*len1*len1*cos(b1)*cos(b1) + ks*len1*len1*sin(b1)*sin(b1);
}
for(i=2;i¡=6;i++)
for(j=1;j¡=(i-1);j++)
sm[i][j] = sm[j][i];
fprintf(f2,“Stiffness Matrix of element %d:”,ie);
for(i=1;i¡=6;i++)
{
```

```
for(j=1;j¡=6;j++)
fprintf(f2,"%f ",sm[i][j]);
fprintf(f2," ");
}
im[1]=3*je[ie]-2;
im[2]=3*je[ie]-1;
im[3]=3*je[ie];
im[4]=3*ke[ie]-2;
im[5]=3*ke[ie]-1;
im[6]=3*ke[ie];
for(j=1;j¡=6;j++)
{
i1 = im[j];
if(jrl[i1] > 0)continue;
for(k=j;k¡=6;k++)
{
i2 = im[k];
if(jrl[i2] ¿ 0)continue;
ir = id[i1];
ic = id[i2];
if(ir¡ic) ic=ic-ir+1;
else{
item1=ir;
ir=ic;
ic=item1;
ic=ic-ir+1;
}
sff[ir][ic] = sff[ir][ic] + sm[j][k];
}
}
```

```
}
fprintf(f2,“Overall Stiffness Matrix :”);
for(i=1;i¡=nra;i++)
{
for(j=1;j¡=nb;j++)
fprintf(f2,“%f ”,sff[i][j]);
fprintf(f2,“”);
}
fscanf(f1,“%d”,&nlj);
for(i=1;i¡=nlj;i++)
{
fscanf(f1,“%d %d %f”,&lj,&type,&load);
ac[3*lj+(type-3)] += load;
}
fprintf(f2,“Combined load vector : ”);
for(i=1;i¡=3*nn;i++)
{
fprintf(f2,“ac[%d]=%f”,i,ac[i]);
fprintf(f2,“”);
}
for(i=1;i¡=nra;i++)
aj[i]=0.;
for(i=1;i¡=3*nn;i++)
aj1[id[i]]=ac[i];
for(i=1;i¡=nra;i++)
aj[i]=aj1[i];


df=cal((nra+1),float);
djg = cal((3*nn+1),float);
```

```
banfac(nra,nb);
bansol(nra,nb);
for(i=1;i¡=3*nn;i++)
djg[i]=0.0;
k=1;
for(i=1;i¡=3*nn;i++)
{
if(jrl[i] ! = 0) djg[i] = 0.;
else
{
djg[i] = df[k];
k = k+1;
}
}
fprintf(f2,"GLOBAL DISPLACEMENT VECTOR.");
fprintf(f2,"Joint Disp.- X Disp.- Y Rotn.- Z ");
for(i=1;i¡=nn;i++)
fprintf(f2,"%.5d %9.2e %9.2e %9.2e ",i,djg[3*i-2],djg[3*i-1],djg[3*i]);
```

Appendix C

C program for meshing of beam

The developed c program is used for generating meshing for beam and solid deep beam

```
#include<stdio.h>
#include<math.h>
void main()
{
int i,j,k,nb,ns,nspr;
float bl,sh,elast,g,th ;
float mibeam,abeam,massb;
char ch1[30];
FILE *f1;
clrscr();
printf("Enter name of Input file to be prepared :");
gets(ch1);
f1=fopen(ch1,"w");
printf("Enter no. of bays :");
scanf("%d",&nb);
printf("Enter width of bays :");
scanf("%f",&bl);
printf("Enter no. of storeys :");
```

```
scanf(”%d”,&ns);
printf(”Enter height of storeys :”);
scanf(”%f”,&sh);
printf(”Enter number of springs :”);
scanf(”%d”,&nspr);
printf(”Enter thickness of element:”);
scanf(”%f”,&th);
printf(”Enter Modulus of Elasticity of material:”);
scanf(”%f”,&elast);
printf(”Enter Modulus of Rigidity of Material:”);
scanf(”%f”,&g);
fprintf(f1,”%d %d %d”,(ns*(2*nb+1))+nb,((nb+1)*(ns+1)),nspr);
for(i=1;i¡=ns;i++)
{
if(i == 1)
{
for(j=1;j¡=nb;j++)
{
fprintf(f1,”%d %d %f %f %f”,j,j+1,th,elast,g);
}
}
for(j=1;j¡=nb+1;j++)
{
if(j == 1)
{
fprintf(f1,”%d %d %f %f %f”,(i-1)*(nb+1)+j,i*(nb+1)+j,th,elast,g);
}
if(j == (nb+1))
{
fprintf(f1,”%d %d %f %f %f”,(i-1)*(nb+1)+j,i*(nb+1)+j,th,elast,g);
```

```
}
if(j ! = 1 && j ! = (nb+1))
{
fprintf(f1,”%d %d %f %f %f”,(i-1)*(nb+1)+j,i*(nb+1)+j,th,elast,g);
}
}
for(j=1;j¡=nb;j++)
{
fprintf(f1,”%d %d %f %f %f”,i*(nb+1)+j,i*(nb+1)+j+1,th,elast,g);
}
}
for(i=1;i¡=ns+1;i++)
{
for(j=1;j¡=nb+1;j++)
{
fprintf(f1,”%f %f”,(j-1)*bl,(i-1)*sh);
}
}
}
```

Appendix D

C program for meshing of portal frame

Using this program an element connectivity and nodal coordinates are generated for portal frame.

```
#include<stdio.h>
#include<math.h>
void main()
{
int i,j,k,nb,ns,nspr,ncol,nbb,nsb;
float bl,sh,elast,g,th,colspac;
float mibeam,abeam,massb;
char ch1[30];
FILE *f1;
clrscr();
printf("Enter name of Input file to be prepared :");
gets(ch1);
f1=fopen(ch1,"w");
printf("Enter no. of columns : ");
scanf("%d",&ncol);
printf("Enter column spacing :");
```

```
scanf("%f",&colspac);
printf("Enter no. of bays :");
scanf("%d",&nb);
printf("Enter width of bays :");
scanf("%f",&bl);
printf("Enter no. of storeys :");
scanf("%d",&ns);
printf("Enter height of storeys :");
scanf("%f",&sh);
printf("Enter no. of bays in beam : ");
scanf("%d",&nbb);
printf("Enter no. of storeys in beam : ");
scanf("%d",&nsb);
printf("Enter number of springs :");
scanf("%d",&nspr);
printf("Enter thickness of element:");
scanf("%f",&th);
printf("Enter Modulus of Elasticity of material:");
scanf("%f",&elast);
printf("Enter Modulus of Rigidity of Material:");
scanf("%f",&g);
fprintf(f1,"%d %d %d",((nsb*(2*nbb+1))+ncol*(nb+1)+nbb)+ncol*((ns*(2*nb+1))+nb),
(nsb+1)*(nbb+1)+ncol*((nb+1)*(ns+1)),nspr);
for(k=1;k<ncol;k++)
{
if(k==1) {
for(i=1;i<=ns;i++)
{
if(i == 1)
{
```

```
for(j=1;j<=nb;j++)
{
fprintf(f1,“%d %d %f %f %f”,j,j+1,th,elast,g);
}
}
for(j=1;j<=nb+1;j++)
{
if(j == 1)
{
fprintf(f1,“%d %d %f %f %f”,(i-1)*(nb+1)+j,i*(nb+1)+j,th,elast,g);
}
if(j == (nb+1))
{
fprintf(f1,“%d %d %f %f %f”,(i-1)*(nb+1)+j,i*(nb+1)+j,th,elast,g);
}
if(j ! = 1 && j ! = (nb+1))
{
fprintf(f1,“%d %d %f %f %f”,(i-1)*(nb+1)+j,i*(nb+1)+j,th,elast,g);
}
}
for(j=1;j¡=nb;j++)
{
fprintf(f1,“%d %d %f %f %f”,i*(nb+1)+j,i*(nb+1)+j+1,th,elast,g);
}
}
}
```

```
for(i=1;i<=ns;i++)
{
if(i == 1)
{
for(j=1;j<=nb;j++)
{
fprintf(f1,“%d %d %f %f %f”,2*(ns*nb)+2*nb+j,
2*(ns*nb)+2*nb+j+1,th,elast,g);
}
}
for(j=1;j<=nb+1;j++)
{
if(j == 1)
{
fprintf(f1,“%d %d %f %f %f”,2*(ns*nb)+2*nb+(i-1)*(nb+1)+j,
2*(ns*nb)+2*nb+i*(nb+1)+j,th,elast,g);
}
if(j == (nb+1))
{
fprintf(f1,“%d %d %f %f %f”,2*(ns*nb)+2*nb+(i-1)*(nb+1)+j,
2*(ns*nb)+2*nb+i*(nb+1)+j,th,elast,g);
}
if(j ! = 1 && j ! = (nb+1))
{
fprintf(f1,“%d %d %f %f %f”,2*(ns*nb)+2*nb+(i-1)*(nb+1)+j,
2*(ns*nb)+2*nb+i*(nb+1)+j,th,elast,g);
}
}
for(j=1;j<=nb;j++)
{
```

```
fprintf(f1,“%d %d %f %f %f”,2*(ns*nb)+2*nb+i*(nb+1)+j,
2*(ns*nb)+2*nb+i*(nb+1)+j+1,th,elast,g);
}
}
}
for(i=1;i<=nsb;i++)
{
if(i == 1)
{
for(j=1;j<=nbb;j++)
{
fprintf(f1,“%d %d %f %f %f”,2*(2*nb*(ns+1))+j,
2*(2*nb*(ns+1))+j+1,th,elast,g);
}

    for(j=1;j<=nbb;j++)
{
if(j<=nb+1)
{
fprintf(f1,“%d %d %f %f %f”,2*(nb*ns)+j,
2*(2*nb*(ns+1))+j,th,elast,g);
}

    if(j>=(colspac/bl)+nb)
{
fprintf(f1,“%d %d %f %f %f”,2*(ncol*(nb*ns)+ncol*nb)-(nb+1)+j,
2*(2*nb*(ns+1))+(colspac/bl)+nb+j,th,elast,g);
}
}
for(j=1;j<=nbb+1;j++)
```

```
{
if(j == 1)
{
fprintf(f1,"%d %d %f %f %f",2*(2*nb*(ns+1))+(i-1)*(nbb+1)+j,
2*(2*nb*(ns+1))+i*(nbb+1)+j,th,elast,g);
}
if(j == (nbb+1))
{
fprintf(f1,"%d %d %f %f %f",2*(2*nb*(ns+1))+(i-1)*(nbb+1)+j,
2*(2*nb*(ns+1))+i*(nbb+1)+j,th,elast,g);
}
if(j ! = 1 && j ! = (nbb+1))
{
fprintf(f1,"%d %d %f %f %f",2*(2*nb*(ns+1))+(i-1)*(nbb+1)+j,
2*(2*nb*(ns+1))+i*(nbb+1)+j,th,elast,g);
}
}
for(j=1;j<=nbb;j++)

fprintf(f1,"%d %d %f %f %f",2*(2*nb*(ns+1))+i*(nbb+1)+j,
2*(2*nb*(ns+1))+i*(nbb+1)+j+1,th,elast,g);
}
}
}
for(k=1;k<ncol;k++)
{
if(k==1)
{
for(i=1;i<=ns+1;i++)
{
```

```
for(j=1;j<=nb+1;j++)
{
fprintf(f1,"%f %f",(j-1)*bl,(i-1)*sh);
}
}
}
for(i=1;i<=ns+1;i++)
{
for(j=1;j<=nb+1;j++)
{
fprintf(f1,"%f %f",colspac+bl+(j-1)*bl,(i-1)*sh);
}
}
}
for(i=1;i<=nsb+1;i++)
{
for(j=1;j<=nbb+1;j++)
{
fprintf(f1,"%f %f",(j-1)*bl,(i-1)*sh+((ns+1)*sh));
}
}
}
```

Appendix E

Sample Input and Output File

Input file for axially loaded column subjected to UDL

\∗ "Height of column" "Width of column" "Thickness of column" "No. of elements" "No. of springs" "E"

300 10 10 5 10 100

\∗"Element connectivity"

1 2

2 3

3 4

4 5

\∗ No. of restraint joint (nrj)

5

\∗ Restraint condition

1 1

2 1

3 1

4 1

5 0

\∗"Nodal load condition (column subjected to UDL of 10 N/mm. UDL is converted into equivalent nodal load of 200 N/mm.

5

1 -600

2 -600

3 -600

4 -600

5 -600

\∗ End of input for column subjected to UDL.

Output file for axially loaded column subjected to UDL

length of an element = 300.00

Width of an element = 10.00

thickness of element= 10.00

elasticity=100.00

No. of springs = 05

No.of element =05

length of spring element= 60.00

Width of a spring element = 2.00

Number of member= 04

1 (30.00)

2 (90.00)

3 (150.00)

4 (210.00)

5 (270.00)

member data for 1 member: je= 1 ke= 2

member data for 2 member: je= 2 ke= 3

member data for 3 member: je= 3 ke= 4

member data for 4 member: je= 4 ke= 5

No. of restrained joint= 05

restrained joint no.=1, dispx=1

restrained joint no.=2, dispx=1

restrained joint no.=3, dispx=1

restrained joint no.=4, dispx=1

restrained joint no.=5, dispx=0

stiffness matrix for member 1, 2, 3, 4 & 5

spring stiffness matrix 1

33.33 -33.33

-33.33 33.33

spring stiffness matrix 2

33.33 -33.33

-33.33 33.33

Likewise, stiffness matrix for each spring can be obtained.

Equivalent stiffness matrix for element 1, 2, 3 & 4

166.67 -166.67

-166.67 166.67

Assembled stiffness matrix:

166.67 -166.67 0.00 0.00 0.00

-166.67 333.33 -166.67 0.00 0.00

0.00 -166.67 333.33 -166.67 0.00

0.00 0.00 -166.67 333.33 -166.67

0.00 0.00 0.00 -166.67 166.67

Combined load Vector:

ac[1]=-600.00

ac[2]=-600.00

ac[3]=-600.00

ac[4]=-600.00

ac[5]=-600.00

Displacements:

Disp[1]=-36.00

Disp[2]=-32.40

Disp[3]=-25.20

Disp[4]=-14.40

Disp[5]=-0.00

* End of output for column subjected to UDL.

Input file for cantilever beam subjected to point load at free end

\∗ “No. of member” “No. of Elements” “No. of springs”

28 20 5

\∗“J-end” “K-end” “Thickness of member” “E” “G”

1 2 300.00 25000.00 3750.00

2 3 300.00 25000.00 3750.00

3 4 300.00 25000.00 3750.00

4 5 300.00 25000.00 3750.00

5 6 300.00 25000.00 3750.00

6 7 300.00 25000.00 3750.00

7 8 300.00 25000.00 3750.00

8 9 300.00 25000.00 3750.00

9 10 300.00 25000.00 3750.00

1 11 300.00 25000.00 3750.00

2 12 300.00 25000.00 3750.00

3 13 300.00 25000.00 3750.00

4 14 300.00 25000.00 3750.00

5 15 300.00 25000.00 3750.00

6 16 300.00 25000.00 3750.00

7 17 300.00 25000.00 3750.00

8 18 300.00 25000.00 3750.00

9 19 300.00 25000.00 3750.00

10 20 300.00 25000.00 3750.00

11 12 300.00 25000.00 3750.00

12 13 300.00 25000.00 3750.00

13 14 300.00 25000.00 3750.00

14 15 300.00 25000.00 3750.00

15 16 300.00 25000.00 3750.00

16 17 300.00 25000.00 3750.00

17 18 300.00 25000.00 3750.00

18 19 300.00 25000.00 3750.00

19 20 300.00 25000.00 3750.00

*"X-coordinate" "Y-coordinate"

0.00 0.00

300.00 0.00

600.00 0.00

900.00 0.00

1200.00 0.00

1500.00 0.00

1800.00 0.00

2100.00 0.00

2400.00 0.00

2700.00 0.00

0.00 300.00

300.00 300.00

600.00 300.00

900.00 300.00

1200.00 300.00

1500.00 300.00

1800.00 300.00

2100.00 300.00

2400.00 300.00

2700.00 300.00

*"No. of restraint joint"

2

*"Joint No." "Horizontal DOF" "Vertical DOF" "Rotational DOF"

1 0 0 0

11 0 0 0

\∗"No. of joint loads"

1

\∗"Joint No." "Load type" "Intensity of load"

20 2 -100000

\∗ End of input file for cantilever beam subjected to $100kN$ point at free end

Output file for cantilever beam subjected to point load (300 mm Element size & 5 No. of Springs

No. of elements = 28

No. of nodes = 20

No.of springs=5

"Member connectivity and Material properties"

Member no.=1

j end= 1, k end= 2, thickness= 300.00 Elasticity= 25000.00, Modulus of rigidity= 3750.00

Member no.=2

j end= 2, k end= 3, thickness= 300.00 Elasticity= 25000.00, Modulus of rigidity= 3750.00

Member no.=3

j end= 3, k end= 4, thickness= 300.00 Elasticity= 25000.00, Modulus of rigidity= 3750.00

Member no.=4

j end= 4, k end= 5, thickness= 300.00 Elasticity= 25000.00, Modulus of rigidity= 3750.00

Similarly, for all members 1-28.

joint no.=1, X-co-ordinate=0.00 Y-co-ordinate=0.00

joint no.=2, X-co-ordinate=300.00 Y-co-ordinate=0.00

joint no.=3, X-co-ordinate=600.00 Y-co-ordinate=0.00

joint no.=4, X-co-ordinate=900.00 Y-co-ordinate=0.00

joint no.=5, X-co-ordinate=1200.00 Y-co-ordinate=0.00

joint no.=6, X-co-ordinate=1500.00 Y-co-ordinate=0.00

joint no.=7, X-co-ordinate=1800.00 Y-co-ordinate=0.00

joint no.=8, X-co-ordinate=2100.00 Y-co-ordinate=0.00

joint no.=9, X-co-ordinate=2400.00 Y-co-ordinate=0.00

joint no.=10, X-co-ordinate=2700.00 Y-co-ordinate=0.00

joint no.=11, X-co-ordinate=0.00 Y-co-ordinate=300.00

joint no.=12, X-co-ordinate=300.00 Y-co-ordinate=300.00

joint no.=13, X-co-ordinate=600.00 Y-co-ordinate=300.00

joint no.=14, X-co-ordinate=900.00 Y-co-ordinate=300.00

joint no.=15, X-co-ordinate=1200.00 Y-co-ordinate=300.00

joint no.=16, X-co-ordinate=1500.00 Y-co-ordinate=300.00

joint no.=17, X-co-ordinate=1800.00 Y-co-ordinate=300.00

joint no.=18, X-co-ordinate=2100.00 Y-co-ordinate=300.00

joint no.=19, X-co-ordinate=2400.00 Y-co-ordinate=300.00

joint no.=20, X-co-ordinate=2700.00 Y-co-ordinate=300.00

\∗ Stiffness matrix for element.

Stiffness Matrix of element 1:

7500000.00 -0.481299 -540000000.00 -7500000.00 -0.651169 540000000.00

-0.481299 1125000.00 168750032.00 0.651169 -1125000.00 168749952.00

-540000000.00 168750032.00 79312502784.00 540000000.00 -168749952.00 -28687503360.00

-7500000.00 0.651169 540000000.00 7500000.00 0.481299 -540000000.00

-0.651169 -1125000.00 -168749952.00 0.481299 1125000.00 -168750032.00

540000000.00 168749952.00 -28687503360.00 -540000000.00 -168750032.00 79312502784.00

* Applied Load Vector{F}

ac[1]=0.00

ac[2]=0.00

ac[3]=0.00

.

.

.

ac[58]=0.00

ac[59]=-100000.00

ac[60]=0.00

* GLOBAL DISPLACEMENT VECTOR.

Joint Disp.- X Disp.- Y Rotn.- Z

1 0.00e+000 0.00e+000 0.00e+000

2 -1.70e-001 -1.84e-001 -9.55e-004

3 -3.09e-001 -6.01e-001 -1.62e-003

4 -4.27e-001 -1.20e+000 -2.16e-003

5 -5.27e-001 -1.94e+000 -2.61e-003

6 -6.08e-001 -2.81e+000 -2.97e-003

7 -6.71e-001 -3.77e+000 -3.26e-003

8 -7.16e-001 -4.81e+000 -3.46e-003

9 -7.42e-001 -5.89e+000 -3.55e-003

10 -7.42e-001 -6.97e+000 -3.45e-003

11 0.00e+000 0.00e+000 0.00e+000

12 3.84e-002 -1.79e-001 -8.71e-004

13 8.30e-002 -5.94e-001 -1.52e-003

14 1.24e-001 -1.19e+000 -2.06e-003

15 1.59e-001 -1.93e+000 -2.51e-003

16 1.87e-001 -2.80e+000 -2.87e-003

17 2.10e-001 -3.76e+000 -3.16e-003

18 2.26e-001 -4.80e+000 -3.36e-003

19 2.36e-001 -5.88e+000 -3.47e-003

20 2.45e-001 -6.98e+000 -3.46e-003

\∗ End of output cantilever beam subjected to point load (300 mm Element size & 5 No. of Springs).

Appendix F

List of Papers Published/ Communicated

- Vikas Gohel and Dr. Paresh Patel, “Static Analysis of Beam Subjected to Point Load at Free End using Applied Element Method (AEM)”, 8^{th} Biennial Conference (SEC-2012), SVNIT, Surat, India, 19-21 December. (Abstract Communicated)

- Vikas Gohel and Dr. Paresh Patel, “Static Analysis of Portal Frame Subjected to Lateral Load using Applied Element Method (AEM)”, 3^{rd} International Conference,NUiCON-2012, Nirma University, Ahmedabad, Gujarat, India, 6-8 December. (Abstract Communicated)