ADDIS ABABA UNIVERSITY

ADDIS ABABA INSTITUTE OF TECHNOLOGY

SCHOOL OF CIVIL AND ENVIRONMENTAL ENGINEERING

COMPARATIVE STATE-OF-THE-ART REVIEW ON FINITE ELEMENT METHOD AND APPLIED ELEMENT METHOD

A Thesis in Structural Engineering

By: Sirak Dejene

July, 2021

Addis Ababa

A Thesis

Submitted in Partial Fulfillment of the Requirements for the Degree of Master of Science

The undersigned have examined the thesis entitled "**Comparative State-of-the-Art Review on Finite Element Method and Applied Element Method**" presented by Sirak Dejene, a candidate for the degree of Master of Science and hereby certify that it is worthy of acceptance.

Dr. Shifferaw Taye ____________________ Advisor	__________________ Signature	________________ Date
____________________ Internal Examiner	__________________ Signature	________________ Date
____________________ External Examiner	__________________ Signature	________________ Date
____________________ Chair Person	__________________ Signature	________________ Date

UNDERTAKING

I certify that research work titled “Comparative State-of-the-Art Review on Finite Element Method and Applied Element Method is my own work. This thesis has not been presented for any other university and is not concurrently submitted in candidature of any other degree, and that all sources of material used for the thesis have been duly acknowledged.

Sirak Dejene

:

ABSTRACT

The intent of this study is presenting Applied Element Method (AEM), AEM is as of late new expansion of discrete element method of structural analysis which is applicable for static linear and non-linear analysis of continuum and framed structures. In AEM structure is partitioned in to discrete elements. Consecutive elements connected to each other with a pair of springs. Simple 2D linear static structural analysis done utilizing AEM, the influence of key parameters such as number of springs, number of elements, one-dimensional discretization and two-dimensional discretization were investigated in this study. The aim of the analysis is to highlight which parameter affects the result and which level of accuracy is expected. Theoretical and Numerical comparison of Finite Element Method (FEM) and Applied Element Method (AEM) performed, by comparing AEM and FEM with their respective features and convergence test is also performed, comparing the numerical results of AEM and FEM with the analytical results respectively.

In order to study the influence of parameters a classic cantilever beam with a concentrated load at the tip is selected. To investigate the parameters in AEM different codes developed utilizing numerical software “Scilab”, Numerical results of AEM plotted against number of elements for the pre-listed parameters. FEM analysis is performed utilizing a finite element software Ansys 2020 R2. To show the accuracy and verification of both methods, numerical results obtained by the AEM and FEM are compared with analytical results. For convergence test, Numerical results of both AEM and FEM are plotted against number of elements and compared with the analytical result.

Keywords: Applied Element Method, Ansys 2020 R2, Finite Element Method, Discrete Element Method

ACKNOWLEDGMENTS

First and foremost, I would like to praise and thank God, the Almighty, who has granted countless blessings, knowledge and opportunity, so that I have been able to accomplish my thesis.

I am extremely grateful for my advisor Dr. Shifferaw Taye for giving me the opportunity to do the research on this topic and providing invaluable advice, continuous support, guidance and patience during this study. His vision, sincerity and motivation have deeply inspired me. He taught me the methodology to carry out the research and to present the research as clearly as possible. His insightful feedback pushed me to sharpen my thinking. I would also like to thank him for his empathy, and great sense of humor. It was a great privilege and honor to work and study under his guidance.

In addition, to my parents, whose love and guidance are with me in whatever I pursue and for their prayer, caring and sacrifice for educating and preparing me for my future. Finally, I could not have completed this thesis without the support of Dr. Lincy Christy and Mr. Daniel Tekie who provided me treasured support.

TABLE OF CONTENTS:

LIST OF TABLES

LIST OF FIGURES

NOTATIONS

AEM: Applied Element Method

FEM: Finite Element Method

FEA: Finite Element Analysis

RBSM: Rigid Body and Spring Model

EDEM: Extended Distinct Method

ANSYS: Analysis System

CPU: Central Processing Unit

PDEs: Partial differential equations

ASI: Applied Science International

ELS: Extreme Loading for Structures

DOF: Degrees of freedom

2D: Two-Dimensional

3D: Three-Dimensional

K: Stiffness

K_G: Global stiffness

K_n: Normal spring stiffness

Ks, Ksy, Ksz: Shear spring stiffness

E: Young’s Modulus

G**:** Shear modulus

d: Distance between springs

t: Thickness

a: Length of the representing area

v: Poisson's ratio

F: Applied load

$K_{G:}$ Global stiffness

Δ: Displacement

LPG: Liquefied petroleum gas

RC: Reinforced concrete

T: Transformation matrix

$K_{h:}$ Stiffness along longitudinal direction

K_v : Stiffness along transverse direction

CHAPTER ONE

1.0 Introduction

1.1 Background of the study

A building is an arrangement of connected components, which are either structural or nonstructural. Structural components are skeleton that supports the structure, they are load-conveying portions of all-regular and man-made forms, it is the component of building which enables them to stand under their own weight and under the most noticeably terrible states conditions of externally applied forces. Non-structural components don't contribute to the structure, they are functional requirements. Their weights are considered and the supports for these non-structural components are designed to account for them. Every part of a building is subject to the effects of outside forces gravity, wind, earthquakes, and temperature changes, to name a few. Buildings should withstand these forces over a long period of time.

A structure is a system of connected components used to resist applied loads. Structures are divided in various groups according to their function and type of elements used to make up the structure. Structures have supports at various points, which are useful when all loads applied to the structure these supports oppose the movement of the structures caused by the applied load. In order to design a structure, one must analyze them, structural engineer needs to address the adequacy of structure while designing or assessing, most important issue is failure.

There are three types of failure, Instability failure if either the number or nature of the reactions is insufficient to satisfy the equilibrium conditions, the structure is said to be Initially Unstable. Even structure is supported adequately it still may be initially unstable if the members are not properly connected together. Generally, initial instability occurs due to lack of appropriate support or an inadequate arrangement of members. Second type of failure which relies on stress-strain relationship of a material is called loss of stability due to material failure. This failure occurs when the level of stress in components reaches the ultimate stress for the material. Last type of failure which is associated with slender structures is called buckling failure it occurs when slender structures are subjected to a compressive load, when this applied load exceeds the axial compressive strength of a material. Structural engineer should avoid these types of failures specially avoid sudden or catastrophic failures.

In designing a structure, a structural engineer must consider all loads that can be reasonably expected on the structure during its design life. Loads that act on common civil engineering structures generally classified in direction of application Vertical loads including dead load, live load and impact loads, and Lateral loads such as wind and earthquake. Source of loads on structures are caused by various actions which include, (a) interaction of structure with the natural environment, these loads depend on the nature of the structure and geographic location of the site includes gravity, snow, wind, earthquake, water, ice, earth pressure, etc. (b) carrying out function they are expected to serve, (c) construction of the structure and (d)terrorist activity. The fundamental aim of a structural engineer is to design a structure that will be able to withstand all the loads to which it is subjected while serving its intended purpose through its intended life span.

In addition to estimating the magnitudes of the design loads, an engineer must also consider the possibility that some of these loads might act simultaneously on the structure. The structure is finally designed so that it will be able to withstand the most unfavorable combination of loads that is likely to occur in its lifetime. The minimum design loads and the load combinations for which the structures must be designed are usually specified in building codes (Kassimali, 2011).

Structural engineering is the science and craft of preparation planning, designing, and constructing safe and economical structures that will serve their intended purposes. Structural analysis is a vital piece of any structural engineering project, its function being the prediction of the performance of the proposed structure. Structural analysis is the determination and assurance of the impacts of loads on physical structures and their components, to fulfill fundamental prerequisites of function, safety, and sometimes aesthetics. Structural analysis is thus a key part of the engineering design of structures. Structures must serve a predefined function for public use, the engineer must account for its safety, aesthetics, and serviceability while taking into consideration economic and environmental constraints.

Structural analysis is concerned with quantifying the reaction of structures subjected to external loading. The scope includes determining the magnitude of reactions, internal forces, and displacements (Faraji, 2013). There are various methods of structural analysis, applied in a range of simple to complex, determinate to indeterminate structures. This study only considers on two numerical methods of structural analysis, Finite Element Method (FEM) is broadly utilized method for numerically addressing differential equations arising in mathematical and engineering modeling, and Applied Element Method (AEM) is a numerical analysis only applied for structural analysis which is used in predicting the continuum and discrete behavior of structures.

1.2 Objective of the study

The main purpose of the present study is to lead to an improved understanding of AEM, show the effects of parameters including number of springs, number of elements, one-dimensional discretization and two-dimensional discretization in utilizing AEM for structural analysis and the aim is focused on the analysis of structure to study the influence of key parameters in AEM. Effects of parameters are showed by analyzing a simple classic cantilever beam with a concentrated load at the tip. To show the accuracy and applicability of the AEM it is compared with the well-known FEM both numerically and theoretically.

1.2.1 General objectives

The salient objectives of the present study:

a) Introduction of Applied Element Method (AEM).
b) Comparison of AEM and FEM on their respective features.

1.2.2 Specific objectives

a) Analysis of simple structure using AEM.
b) Parametric study of structural analysis using AEM.
c) Comparison of numerical results by AEM and FEM with the analytical result.

1.3 Statement of the problem

Structural analysis is the determination of the impacts of loads on physical structures and their components, to fulfill fundamental prerequisites of function, safety, and sometimes aesthetics. There are different structural analysis techniques, one of the exceptionally productive techniques is the numerical analysis method. The numerical analysis methods ought to fulfill the necessities of "Precision", "Effortlessness" and "Appropriateness". The expression "Precision" means that realistic results should be obtained, "Effortlessness" indicates that the method should not be complicated and performed at ease and "Appropriateness" signifies the method can be applied for a wide scope of utilizations. Existing Numerical methods for structural analysis can be categorized into two classifications dependent on the understanding of target material. The first-class model is based on continuum material equations. The subsequent class utilizes discrete element techniques. As of now accessible numerical methods, Finite Element Method (FEM) is a continuum strategy while Rigid Body and Spring Model (RBSM) and Extended Distinct Method (EDEM) are Discrete Element techniques.

From existing numerical methods, the three necessities can’t be satisfied by one determined method. Lately, there is a numerical method that is added to the group of discrete element techniques that is applicable for static linear and non-linear framed structures. AEM is similarly productive, it isn't however well known as FEM it seems to be not mainstream because of limited literature discussing. This study centers around conferring better comprehension about this new numerical structural analysis method called Applied Element Method (AEM).

1.4 Significance of the study

The significance of the study presents a recently new efficient method used for structural analysis. The fundamental benefit of this strategy effortlessness in modeling and programming. It combines the advantage of both continuum and discrete element methods and confers high exactness of the outcomes. It gives a better understanding of the method and clearly shows the effect of studied parameters. Generally, the result of this study provides essential base line to the new applied element method and it leads to utilization of this method to be used for various structural analysis.

1.5 Scope and limitation of the research

The scope of this study is to perform a two-dimensional linear static structural analysis on a simple cantilever beam using two numerical method AEM and FEM. The ANSYS 2020 R2 package program was used as a tool for finite element analysis, numerical software Scilab 6.1.0 used as a tool for AEM.

a) This study will focus on structural analysis of two-dimensional classic cantilever beam using AEM and FEM.
b) Cantilever beam modeled in Ansys 2020 R2 for FEM and for AEM only a numerical software Scilab v 6.1.0 used.

1.6 Thesis organization

This thesis is organized into five chapters as follows: Chapter 1 presents background of the study, statement of problem, objective of the study, and scope and limitation of the study. The content of this work is summarized here. Chapter 2 presents a literature review related to the research. This chapter begins with brief history of AEM and FEM, Overview of AEM, and stiffness matrix of AEM as a method of structural analysis and AEM application to blast and earthquake analysis. Chapter 3 describes the methodology employed in this study. Chapter 4 presents a numerical validation by a two-dimensional structural analysis using AEM, also explains the effect of selected parameter on Numerical results by AEM and comparing with analytical result and with numerical results of FEM found from two-dimensional structural analysis using a finite element software ANSYS 2020 R2. It also summarizes comparison of AEM and FEM with their respective futures. Chapter 5 presents significant conclusions based on the findings and the recommendations for the future researches in the area. Finally, reference materials used during the study are listed and in appendices a sample code developed in numerical software Scilab is given for the analysis of a cantilever beam in one- and two-dimensional discretization.

CHAPTER TWO

2.0 Literature Review

2.1 Introduction

The Finite Element Method (FEM) and other numerical methods are very effectively implemented in linear and nonlinear analysis of structures. Recently, a new displacement-based method called Applied Element Method developed. Applied Element Method (AEM) is a newly developed numerical analysis which is only used for structural analysis to prediction the behavior of structures. The modeling of objects in AEM has the ability to stimulate structural behavior through all stages of loading as well as evaluating seismic behavior of structures (Tagel-Din, 2000).

AEM, is an innovative method for direct progressive collapse simulation, in which strong geometric nonlinearity, element separation and collision can automatically be considered. This method combines the advantages of both, the Finite Element Method (FEM) and the Discrete Element Method (DEM). By using the AEM, the response of the structure can be followed all the way to collapse involving the elastic stage, crack initiation and propagation, reinforcement yielding, large deformations, element failure and separation, rigid body motion of falling elements, collision between elements, and impact forces resulting from falling debris, with acceptable accuracy and within a reasonable CPU time. This method can be easily applied to a wide range of applications; both for small and large deformation ranges under static or dynamic loading conditions, and for linear or nonlinear materials (AlHafian, 2013).

This literature review is divided into four parts. The first part explains a brief history of FEM and AEM. A brief summary of the current numerical methods for structural analysis is provided in the second part. The third part provides Introduction to AEM. The FEM is notable and generally utilized, henceforth for brevity, it isn't presented here. The fourth part of the literature review can be summarized as Collapse simulation of slab applying blast loadings using AEM and FEM, Collapse simulation of a building during an earthquake using AEM and FEM.

2.2 PART I: Brief History

This part presents a concise history of key advancements in the Finite Element Method (FEM) and Applied Element Method (AEM).

2.2.1 History of Finite Element Method

Although the label **Finite Element Method** first appeared in 1960, when it was used by Clough on the paper, "The Finite element Method in Plane Stress Analysis" (Clough, 1952) the thoughts of finite element investigation date back much further. Truth be told, even today, there is some discussion on who could be known as the "developer" of the method, who started the finite element method? furthermore, when did it start? have three distinct answers relying upon whether one asks an applied mathematician, a physicist, or an architect. These experts have some support for guaranteeing the finite element method as their own, on the grounds that each built up the fundamental thoughts independently at various occasions and for various reasons. The applied mathematicians were concerned about boundary value problems of continuum mechanics; in particular, they wanted to find approximate upper and lower bounds for eigenvalues. The physicists were additionally keen on tackling continuum problems, yet they looked for intends to get piecewise surmised capacities to address their persistent capacities.

The FEM method was developed more by engineers using physical insight than by mathematicians using abstract methods. The scientific pillars of the finite element method are a direct result of the need to solving complex elasticity and structural analysis problems in civil and aeronautical engineering. Its development can be traced back to the work by Alexander Hrennikoff (1896–1984), a Russian-Canadian engineer, was considered by many as the primary founder of the finite element method (Hrennikoff, 1940) and Richard Courant (1888–1972), a German-American mathematician/engineer, was also considered as the father of this method (R., 1942). Priority aside, both are credited at the beginning for currently developed finite element methods for structural analysis. While the methodologies utilized by these pioneers are significantly extraordinary, they share one fundamental characteristic: mesh discretization of continuous elements into a small set of elements, called finite elements.

Hrennikoff's work discretizes the domain by utilizing a lattice (grid) analogy while Courant's approach divides the domain into finite triangular sub-regions for the solution of second-order elliptic partial differential equations (PDEs). Courant's contribution was evolutionary, drawing on a large body of earlier results for PDEs developed by Rayleigh, Ritz, and Galerkin. Developments of the finite element method began in earnest in the middle to late 1950s for airframe and structural analysis and gathered momentum through the work of Argyris (1913–2004) at Stuttgart and Clough at Berkeley in the 1960s. Clough refers John H. Argyris, Ray W. Clough, M. J. Turner, and O. C. Zienkeiwicz “each man has made the best contribution to FEM history”. Clough also refers the finite element methods as “the Argyris method” and considers his work Argyris (1954) to be the most important series of papers ever published in the field of Structural Mechanics (Clough, 2004).

The finite element method is a numerical analysis to solve differential and integral equations since behavior of almost all physical system can be represented by these equations finite element can be applied for wide variety of engineering problems for obtaining approximate solutions. Although originally developed to study stresses in complex airframe structures, it has since been extended and applied to the broad field of continuum mechanics. FEM is the most widely used numerical method for solving certain problems of engineering and mathematical physics. Typical problem areas of interest in engineering and mathematical physics that are solvable by use of the finite element method include structural analysis, heat transfer, fluid flow, mass transport, and electromagnetic potential. For problems involving complicated geometries, loadings, and material properties, it is generally not possible to obtain analytical mathematical solutions. Analytical solutions are those given by a mathematical expression that yields the values of the desired unknown quantities at any location in a body (here total structure or physical system of interest) and are thus valid for an infinite number of locations in the body. These analytical solutions generally require the solution of ordinary or partial differential equations, which, because of the complicated geometries, loadings, and material properties, are not usually obtainable. Hence, we need to rely on numerical methods, such as the finite element method, for acceptable solutions. (D.Logan, 2012)

The method is especially important in structural stress analysis. Although it enjoys general acceptance, it still has a large potential for further growth. There are substantial incentives for making the method more efficient and more responsive to the technological requirements of current design practice (Szab, 1979). The solution for structural problems typically refers to determining the displacements at each node and the stresses within each element making up the structure that is subjected to applied loads (D.Logan, 2012).

Since the FEM is a numerical/computational analysis method, it should come as no surprise that its popularity has motivated the development of multiple finite element programs, both commercial and research-oriented (Wiley, 2018). A lot of finite element software packages for structural analysis are presently accessible, Although the first commercial finite element software developed in the late 1960s. Today there are many commercial finite element software packages to select from.

Lately, there is a numerical method that is added to the group of discrete element techniques that is applicable for static linear and non-linear framed structures.

2.2.2 History of Applied Element Method

Applied Element Method (AEM) has been created, which is relevant for static linear and nonlinear analysis of framed and continuum structures (Vikas Gohel, 2012). Starting 1995, professors Hatem TAGEL-DIN (Institute of Industrial Science, The University of Tokyo, Tokyo, Japan) and Kimiro MEGURO (International Center for Disaster-Mitigation Engineering - INCEDE), have developed a new method for structure modeling and analysis. The actual term "Applied Element Method," however was first coined in 2000 in a paper called "Applied Element Method for structural analysis: Theory and application for linear Materials" (Tagel-Din, 2000). This is the “Applied Element Method” and it combines features from finite element and discrete element methods. The Applied Element Method (AEM) is a displacement-based method of structural analysis. Applied Science International (ASI), has developed the only software for AEM called ‘Extreme Loading for Structures’ (ELS), using which the behavior of structures under extreme loads such as earthquakes, hurricanes, bomb blasts and other disasters can be studied (Lincy Christy D., 2018).

2.3 PART II: Numerical methods for structural analysis

Many methods have been developed to be utilized for structural analysis, one of them is numerical methods. Numerical methods are playing a leading role in structural analysis. In spite of the fact that there are numerous numerical methods, some are time taking and some are less exact when contrasted with different methods. Numerical methods for structural analysis can be grouped into two classes named continuum and discrete element models. Where on the first class the model is based on continuum material equations and the subsequent class can follow the structural behavior from zero loading and up to collapse of the structure, which utilizes discrete element methods by considering a structure as a collection of blocks. Currently available numerical methods. Finite Element Method (FEM) is typical example of continuum methods while Rigid Body and Spring Model (RBSM) and Extended Distinct Method (EDEM) are Discrete Element techniques.

The Finite Element Method (FEM) is in the first category of numerical methods for structural analysis i.e., model based on continuum material equation. Finite Element Method is a numerical method which can be used for the accurate solution of engineering problems that are either simple or complex. The power of FEM is its ability to discretize complex problems and analyze it part by part. Irrespective of the geometry of the problem, with proper mesh refinement, FEM provides very accurate solution (Erhunmwun, 2017)

Rigid Body and Spring Model (RBSM) and Extended Distinct Element Method (EDEM) are classified in second category which uses the discrete element technique. The main advantage of these methods is that they can simulate the cracking process with relatively simple technique compared to FEM, while the main disadvantage is that crack propagation depends mainly on the element shape, size and arrangement. EDEM can follow the structural behavior from zero loading up to complete collapse of the structure. However, the accuracy of EDEM in small deformation is less than that of FEM. EDEM can only answer the second question which is “How does the structure collapse?” (Tagel-Din, 2000).

Partial or complete collapse of Structural elements is an important topic under research because it causes extensive causalities inside and outside of the structure. In addition, the collapse of the structure may lead to failure of or collapse of near structures (Bargi, 2015). Albeit this subject is vital for the safety of people. Analysis of continuum media like steel structures, using the FEM showed very high accuracy. Analysis of cracked media, like reinforced concrete is very complicated. Concrete is complex and heterogeneous material and FEM assumes that the structure medium is continuum or uncracked. This means that the special techniques should be used to consider the effects of cracks (Meguro, 2000). The FEM can answer only the following questions, “Will the structure fail or not?”, Using FEM it’s very difficult to answer the second most important question which is “How does the structure collapse?” (Bargi, 2015).

Various fracture analysis of concrete structure has been made by the FEM in which concrete has been considered a homogenous, continuous medium. The FEM, however, is suitable only for studying the process that take place up to fracture. Concrete is a complex, extremely heterogeneous material, whose fracture strength and mode depend on the strength, size, quantity and distribution of gravel as well as the quality of the mortar used. Therefore, the fracture properties of concrete cannot be analyzed by FEM, and establishing method of analysis by which its fracture properties can be analyzed is a major goal of concrete fracture mechanics (Hakuno, 1989).

There is no proper numerical method among the current available technique by which total behavior of structures from zero loading to collapse can be followed with reliable accuracy and reasonable CPU time. These is where AEM has edge over other numerical methods its 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 analysis domain of AEM is proved by developers it can range from initial loading to progressive collapse where FEM is not suitable for such analysis (Tagel-Din, 2002).

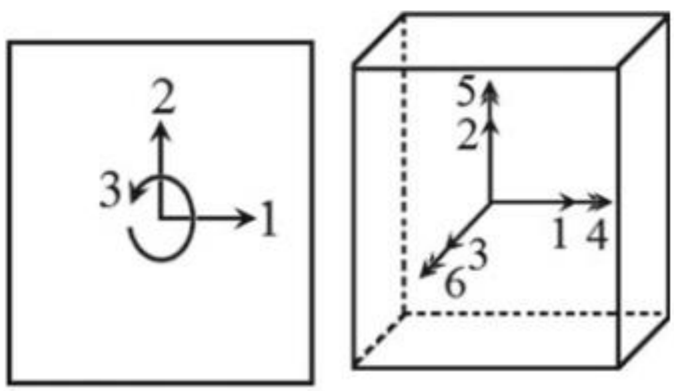
The new displacement based Applied Element Method (AEM) has been developed, which is applicable for static linear and non-linear analysis of framed continuum structures (Vikas Gohel, 2012). The AEM is a relatively recent addition to the discrete element methods family (Daniele Malomo, 2018). Applied Element Method" combines features of Finite Element Method and Discrete Element Method, having as main advantage the possibility to describe the behavior of structure beginning with loadings application, initiation and propagation of cracks, elements separation until total collapse of the structure (Marin Lupoae, 2009). This study focuses on the two methods of numerical structural analysis AEM and FEM.

The Applied Element Method (AEM) is a numerical method of structural analysis, which can be a new tool for discussing a new philosophy of design of structure. Although formulation and material model adopted in the method are simple, highly nonlinear behavior of structures in both static and dynamic conditions can be simulated accurately in reasonable CPU time (Tagel-Din, n.d.).

2.4 PART III: Introduction to Applied Element Method

2.4.1 Overview of Applied Element Method

This section summarizes the theoretical background to the Applied Element Method. **AEM** numerical analysis, relatively recent addition to the discrete elements methods family used in predicting the continuum and discrete behavior of structures. The modeling in AEM method 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. With AEM, a structure is modeled by virtually dividing it into an assembly of small elements. There are two types of elements in AEM, two-dimensional and three-dimensional elements.

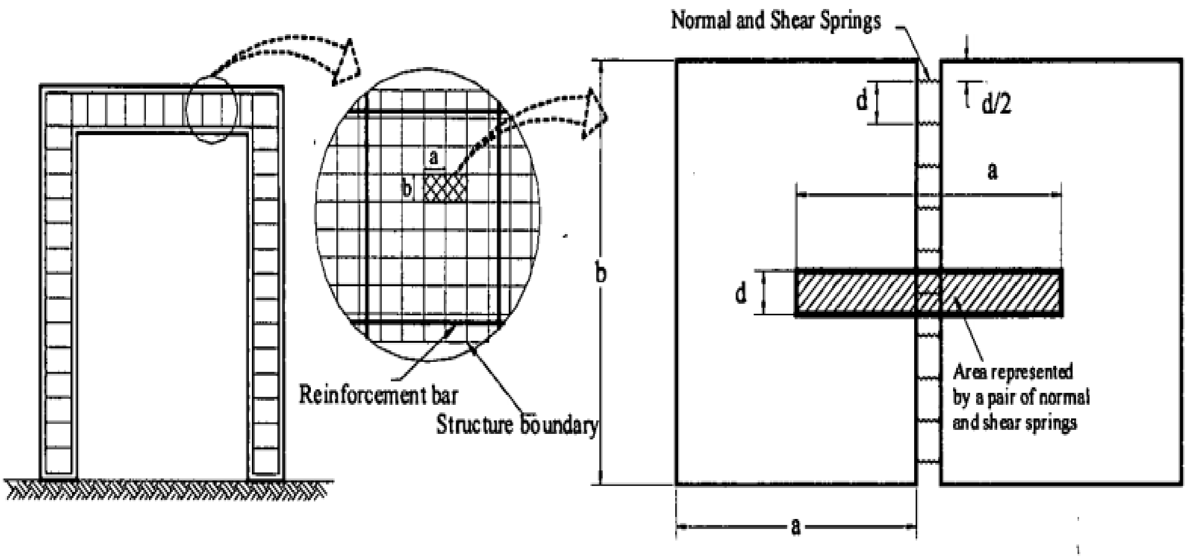


(a) **(b)**

Figure 2.1(a)- 2D elements, (b)- 3D elements (D. Lincy Christy, 2020)

Every two neighboring elements are connected through series of normal and shear springs located at contact points that are distributed over the surface of each element. Normal springs carry normal stress whereas shear spring transfer shear stress from one element to the other. At each contact point, there is one normal spring and one shear springs for two dimensional problems. While one normal spring and two shear springs in orthogonal directions are considered for three dimensional problems.

Each pair of springs totally represent stresses deformations of certain area (hatched area in Fig. 2.2 (b)) of the element. The area of influence on each element for a set of springs is also highlighted in Fig. 2.2 (b).



(a) – Element Generation for AEM

(b) – Spring distribution and area of influence of each pair of springs

Figure 2.2 Modelling of Structure to AEM (Tagel-Din, 2000)

2.4.2 Assumptions in AEM

Assumptions (Vikas Gohel, 2012)

1. Elements are assumed to be rigid (i.e. shape and size doesn't change under applied loading).
2. Elements are assumed to be connected with number of springs.
3. Assembly of Rigid mass and springs behaves as Rigid Body spring mass model.
4. Deformation of an element are assumed to be equal to deformation of springs.
5. Direction of loading are assumed to be constant for analysis of problems.

2.4.3 Degrees of freedoms

In 2D, each element has three degrees of freedom (DOF) at its centroid representing the rigid body motion of the element. These degrees of freedom indicate two translations and one rotation of the element shown in Fig. 2.1 (a). Element moves as a rigid body i.e., element shape does not change.

Although the element motion is a rigid body motion, its internal stresses and deformation can be calculated by the spring deformation around each element. This means that although the element shape does not change during analysis, the behavior of assembly of elements is deformable. (Sushma Pulikanti, 2012). The stiffness matrix size (6×6). Stiffness matrix depends on the contact spring stiffness and the spring location.

In 3D, each element has six degrees of freedom (DOF) at its centroid representing the rigid body motion of the element shown in Fig. 2.1 (b). The element stiffness matrix size (12×12).

2.4.4 Properties of Applied Element Method

Element Shape:

In AEM, each element has 3-D physical coordinates and shape. Hence, elements are not just lines or shells but a group of 3D elements which can be separated and or collided together. In 3-D AEM analysis, cuboids are used to model the structure to be analyzed. Each element center of gravity is calculated at which degrees of freedom (unknown displacements) are calculated.

Element Connectivity:

In FEM elements are connected at nodes, A node is simply a point in space, defined by its coordinates, at which DOF are defined. In contrast with FEM in AEM, elements are connected using the elements entire surface, through a series of connecting springs. These connecting springs represent stresses, strains and connectivity between elements.

2.4.5 Stiffness Matrix 2-D Element

Stiffness is determined by:

Stiffness, $$K = \frac{\text{Force}}{\text{Displacement}} = \frac{\frac{\text{Stress}}{\text{Strain}} \times \text{Area}}{\text{Initial Dimension}} \qquad (1)$$

Theoretical normal and shear stiffness of springs are calculated respectively:

$$k_n = \frac{E \times d \times t}{a} \quad ; \quad k_s = \frac{G \times d \times t}{a} \qquad (2)$$

Where, E and G are Young's Modulus and Shear modulus, respectively, 'd' is the distance between springs, 't' is the thickness, and 'a' is the length of the representing area. Equation 2 simply represents the axial stiffness of each spring. Each element shown in Figure 2.3 has three degrees of freedom at its centroid representing the rigid body motion of the element.

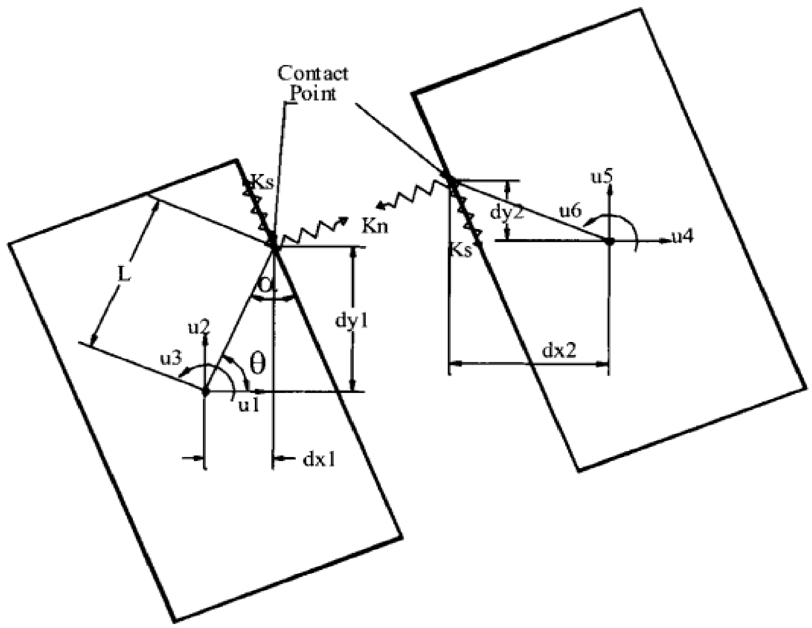


Figure 2.3 Element shape, contact point and DOF (Tagel-Din, 2000)

Fig. 2.3, shows the general position of an element. The two elements are assumed to be connected with pair of normal and shear springs at each contact points distributed along the faces of each element. Coordinates (d_x,d_y) of each contact point are obtained with respect to centroid of an element.

Each pair of springs totally represent stress, strain, deformations and failure of a certain portion of the structure (Tagel-Din, 2002). Spring stiffness is calculated for each spring according to the stress situation and material type of each spring (Meguro, 2001).

In AEM for a 2D element, it has 6 DOF similarly stiffness matrix is size of 6x6, Components of each element stiffness matrix could be determined by applying a unitary displacement to a DOF while keeping the remaining DOF’s fixed.

The forces needed to generate this configuration are the stiffness matrix components, which are equal to the summation of the contributions of the springs surrounding the element. Eq. (3) gives the stiffness matrix of a pair of springs connecting 2 identical elements (Christy D & Nagarajan, 2019).

Allowing a unit displacement in a DOF while keeping other DOF's and using equilibrium equations components of stiffness matrix are found as shown below.

$$k = \begin{bmatrix} k_n & 0 & -k_n y & -k_n & 0 & k_n y \\ 0 & k_s & k_s x & 0 & -k_s & k_s x \\ -k_n y & k_s x & k_n y^2 + k_s x^2 & k_n y & -k_s x & -k_n y^2 + k_s x^2 \\ -k_n & 0 & k_n y & k_n & 0 & -k_n y \\ 0 & -k_s & -k_s x & 0 & k_s & -k_s x \\ k_n y & k_s x & -k_n y^2 + k_s x^2 & -k_n y & -k_s x & k_n y^2 + k_s x^2 \end{bmatrix} \quad (3)$$

The stiffness matrix thus obtained is for a single set of spring. The sum of the stiffness matrix of all the springs in each face will give the stiffness matrix for the corresponding degrees of freedom. The stiffness matrices are then assembled to form the global stiffness matrix K_G.

2.4.6 Stiffness Matrix 3-D Element

The stiffness of normal spring (K_n) and shear springs ($K_{sy} = K_{sz}$) are determined by:

$$K_n = \frac{E \times d \times b}{L}; \qquad K_{sy} = K_{sz} = \frac{E \times d \times b}{L} \quad (4)$$

$$b = \frac{B}{n_z}\,;\ \mathrm{d} = \frac{\mathrm{D}}{\mathrm{n_y}}$$

Where: n_z and n_y are the number of pairs of springs along depth and width of the element

Similarly, also in 3D components of each element stiffness matrix could be determined by applying a unitary displacement to a DOF while keeping the remaining DOF's fixed (Christy D & Nagarajan, 2019).

The stiffness matrix of a pair of springs (*K*) for 3D analysis is given by Eq. (5). In the stiffness matrix, *ksy* and *ksz* are denoted by *ks*.

$$K=\begin{bmatrix}
k_n & 0 & 0 & 0 & k_n z & -k_n y & -k_n & 0 & 0 & 0 & -k_n z & k_n y \\
0 & k_s & 0 & -k_s z & 0 & k_s x & 0 & -k_s & 0 & k_s z & 0 & k_s x \\
0 & 0 & k_s & k_s y & -k_s x & 0 & 0 & 0 & -k_s & -k_s y & -k_s x & 0 \\
0 & -k_s z & k_s y & k_s(y^2+z^2) & -k_s xy & -k_s xz & 0 & k_s z & -k_s y & k_n(y^2+z^2) & -k_s xy & -k_s xz \\
k_n z & 0 & -k_s x & -k_s yx & k_n z^2+k_s x^2 & -k_n yz & -k_n z & 0 & k_s x & k_s yx & -k_n z^2+k_s x^2 & k_n yz \\
-k_n y & k_s x & 0 & -k_s zx & -k_n zy & k_n y^2+k_s x^2 & k_n y & -k_s x & 0 & k_s zx & k_n zy & -k_n y^2+k_s x^2 \\
-k_n & 0 & 0 & 0 & -k_n z & k_n y & k_n & 0 & 0 & 0 & k_n z & -k_n y \\
0 & -k_s & 0 & -k_s z & 0 & -k_s x & 0 & k_s & 0 & -k_s z & 0 & -k_s x \\
0 & 0 & -k_s & -k_s y & k_n x & 0 & 0 & 0 & k_s & k_s y & k_s x & 0 \\
0 & -k_s z & -k_s y & -k_s(y^2+z^2) & k_s xy & k_s xz & 0 & -k_s z & k_s y & k_s(y^2+z^2) & k_s xy & k_s xz \\
-k_n z & 0 & -k_s x & -k_s yx & -k_n z^2+k_s x^2 & k_n yz & k_n z & 0 & k_s x & k_s yx & k_n z^2+k_s x^2 & -k_n yz \\
k_n y & k_s x & 0 & -k_s zx & k_n zy & -k_n y^2+k_s x^2 & -k_n y & -k_s x & 0 & k_s zx & -k_n yz & k_n y^2+k_s x^2
\end{bmatrix} \quad (5)$$

The total stiffness matrix is determined by summing up the stiffness matrices of individual spring around each element. Failure of spring is modeled by assuming zero stiffness for the spring being considered. Consequently, the developed stiffness matrix is an average stiffness matrix for the element according to the stress situation around the element (Tagel-Din, n.d.).

It shows that there is no requirement to define shape functions and no integration processes to determine stiffness matrix in this method which makes AEM simpler and faster than FEM. After determining global stiffness matrix, displacements are calculated through governing system equation: The model can be analyzed by utilizing the following equation:

$$[F] = [K_G] \quad [\Delta] \qquad (6)$$

Where, $[F]$ is the applied load,

$[K_G]$ is the global stiffness

$[\Delta]$ is the displacement vector

2.4.7 Applications

Applications of the Applied Element Method (Anon., n.d.)

Structural Vulnerability Assessment

Due to the nature of extreme loads resulting from blasts, impacts, seismic events and progressive collapse, large amounts of structural damage can usually be expected. This damage cannot be studied solely as the initial loading type; the user must also be able to additionally consider the secondary effects of the failing elements on other parts of the structure. Using the Applied Element Method (AEM), structural engineers perform more complete structural vulnerability assessments through all stages of loading.

Forensic Engineering

Forensic engineering analysis is the study of materials, products, structures and/or components that fail or do not operate/function as intended. These failures or malfunctions can cause costly personal injury and damage to property. The Applied Element Method (AEM) can accurately analyze the same materials, products, structures and/or components through their different loading stages beginning from their linear elastic stage until element separation and collision with another element.

Performance Based Design

Performance-based design is one of engineering's most rapidly emerging fields. Subsequent to this growth in performance-based design, the need for the accurate modeling of structural response becomes paramount, both in the linear and non-linear range. The Applied Element Method (AEM) empowers engineers to analyze their designs using a fully nonlinear analysis method to analyze a structure's response to loads with an increased degree of precision.

Demolition Analysis

Demolition Analysis prior to the demolition and/or deconstruction buildings is becoming increasingly important. The Applied Element Method (AEM) is the optimally suited for this type of analysis as it is the only method that can automatically simulate and analyze the debris resulting from a demolition and predict the effect on neighboring structures.

Glass Performance Analysis

When subjected to blasts, impacts, seismic events or other loading scenarios, glass performance becomes extremely important relevant to the safety and protection of people and property. Whether the use of glass is structural or non-structural, accurate assessment of possible loading scenarios is necessary to provide the highest level of security and peace of mind for occupants.

Visual Effects

When it comes to creating amazing destruction effects for film and television you can bet time, money and realism are traditionally the biggest concerns for the visual effects industry. Current methods of creating destruction effects can often be time consuming, costly and sometimes just don't add up.

2.4.8 Limitations of AEM (AlHafian, 2013)

The material models for concrete and reinforcement in AEM do not consider the effects of the following:

1. Strain rate effects.
2. Tension softening.
3. Spalling of concrete cover, as well as pull out and buckling of reinforcement bar. This is because steel rebar and concrete have the same DOF, as they are modelled by springs attached to rigid elements.

2.5 PART IV: Collapse simulation study using AEM and FEM

2.5.1 Collapse simulation of a slab by blast loading

In general engineers design the building for the dynamic loads like earthquake and wind loadings. Over the last few decades' considerable attention has been raised on the engineering behavior of structures under the non-linear dynamic loadings like blast and impact loading. The explosion inside or near by a building leads to catastrophic damage on the building system and life. Not only the terrorist activity but also the accidental explosions of fuel tanks and LPG cylinders blast may also cause severe damage to the building structural system and leads to loss of life. (Jainu Karthik, 2019)

A brief review of experimental methods for testing blast effect on structure is presented. In today's modern era of computers which are becoming powerful tool implemented in all aspects of life and also scientific research, comparison of experimental and numerical techniques is also given. (H. Daganic, 2018)

The blast resistance of different types of civilian and military structures against accidental explosions and terrorist attacks is an important security issue. Attacks towards vulnerable structures can cause a large number of causalities especially if total collapse occurs. Reducing Vulnerability of existing and future buildings and transportation terminals is a topic of major concern for researchers in both civil and material engineering. (H. Daganic, 2018)

Structural damages caused by blast loading are the combination of both immediate effects and consecutive hazards, among which is progressive collapse. It is great importance to investigate and improve the response of structures to blast loading. Compared to other construction materials, concrete is generally known to have a relatively high extreme blast resistance capacity. To enhance the resistance of blast loads, existing concrete structures require additional retrofitting. (H. Daganic, 2018)

(Christoph Grunwalda, 2018) These paper compares the results of Finite Element Method (FEM) and Applied Element Method (AEM) simulations to experimental results when performing blast or earthquake analysis on those three scales. FEM is the only method able to explicitly consider shock waves caused by blast by performing a Multiphysics analysis. For the other techniques, blast load has to be modeled using a pressure time history or immaculate element removal. Although in many cases blast events can be replaced by simple column removal scenarios, neglecting initial conditions due to blast on other parts of the structure may in some cases lead to estimations that are too conservative. The blast event was approximated by removing the support of the center column, aimed to investigate the behavior of thin column-supported slabs under sudden column removal caused e.g., by a blast event or a vehicle impact.

The scaled experiments contained a RC slab, which was supported by nine columns and loaded with dead weights to represent the expected loading on the structure. Column B1 (compare Figure 2.4, top left) was placed on a device to rapidly remove the support condition and simulate the loss of an exterior column.

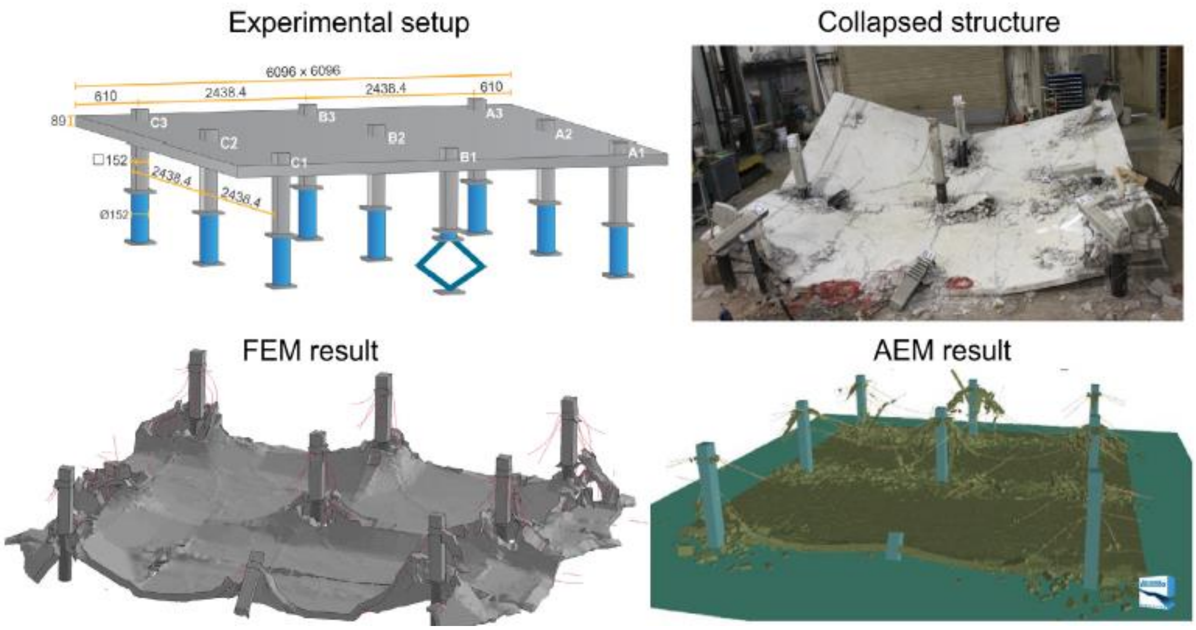


Figure 2.4 Simulated and real collapse shape for the exterior column removal scenario (Christoph Grunwalda, 2018)

Upon the first removal of the support, no collapse occurred in the test ("Drop 1"). A deflection of point B1 of around 29mm was measured. The slab was damaged but remained stable. The column was pushed back in position and the experiment was repeated with increased dead weights ("Drop 2"). Measured peak deflection was now 95 mm, but again no collapse occurred. Afterwards, the structure was pushed back in place a second time, the load was again increased and collapse finally occurred ("Drop 3"). Besides the vertical deflection, the horizontal movement of the column stubs was measured in order to detect compressive arch and catenary action.

FE simulation is done on LS-DYNA which is a general-purpose finite element program capable of simulating complex real world problems. It is used by the automobile, aerospace, construction, military, manufacturing, and bioengineering industries. AE simulation is done on Extreme Loading for Structures (ELS) which is commercial structural-analysis software based on the applied element method for the automatic tracking and propagation of cracks, separation of elements, element collision, and collapse of structures under extreme loads. Both the FE and AE model are able to reproduce the first loading stage and the final collapse reasonably well. The vertical displacements are represented very well, while the in-plane movements are still estimated fairly.

2.5.2 Collapse simulation of a building by earthquake

During earthquakes, buildings suffer from partial and complete collapse to structural or non-structural elements. Partial and complete collapse of structures is important topic under research because it causes extensive causalities inside and outside of the structures. In addition, collapse of a structure may lead to failure or collapse of near structures.

(Christoph Grunwalda, 2018) In 2011, a 6.3 magnitude earthquake occurred in Christchurch, New Zealand. With ground accelerations between 1.8 and 2.2 g near the epicenter. Among the partially collapsed buildings was the Pyne Gould Corporation (PGC) Building at 233 Cambridge Terrace, in which 18 fatalities occurred. The building was designed and built from 1963 to 1966 as a RC frame structure with an internal shear core and it has five floors.

The building was modeled using LS-DYNA and Extreme Loading for Structures in accordance with the technical drawings, including the retrofit measures. Less important structural details, such as stairs and parapet walls on the individual floors, were neglected in order to reduce the model size. Girders were discretized with four elements, slabs with one and walls with two elements over the thickness. Reinforcement was taken into account by explicitly modeling the different layers.

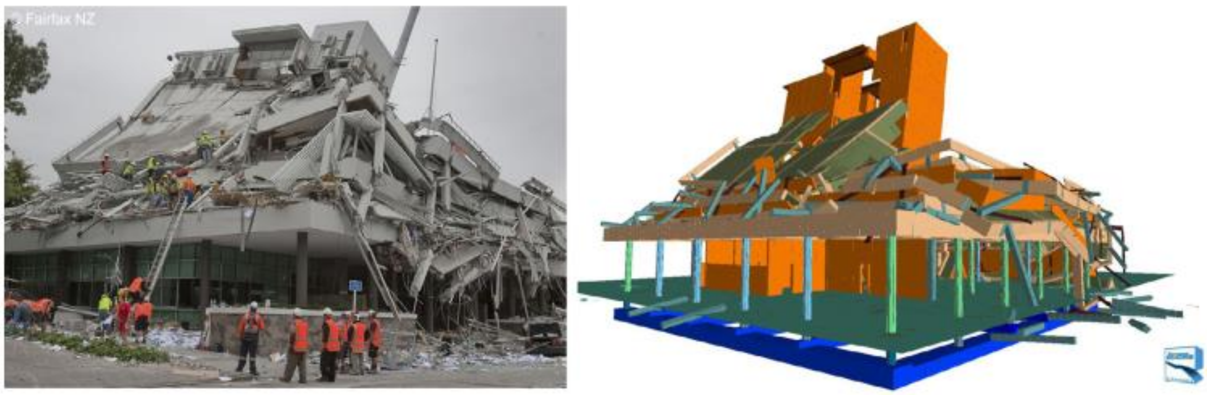


Figure 2.5 South-west elevation of the collapsed PGC building. comparison between real (left) and simulated (right) collapse shape (Christoph Grunwalda, 2018)

The above figure compares the final collapse shape after 25 s with the real structure and show a very close agreement. the predicted obliquity of the core (78°) is close to the real rotation (68°). Keeping in mind the rough approximation for the interaction with the adjacent building due to the lack of information and that furniture and other non-structural elements reduce the sliding of the structural parts, a deviation of 10° inclination is a very good result using AE simulation. Therefore, in the AE simulation the loss of stability in the first floor leads to the obliquity of the core and subsequent detachment of slabs. The final collapse shape is reached earlier and in better agreement with the real case.

CHAPTER THREE

3.0 Methodology

3.1 General

This section mainly focuses on the methods and materials used to conduct this study. This study mainly includes theoretical discussions of AEM and computer programming and structural analysis work is performed in order to study the effects of selected parameters. Two-dimensional modeling and analysis are done using a finite element software ANSYS 2020 R2, and for AEM by the help of a numerical software Scilab. The numerical results obtained by these methods were discussed with respect to the analytical result, parameters like number of elements, number of springs and discretization type are included in AEM method of structural analysis.

3.2 Materials used

After setting the research objectives, scope and limitation of the study. To meet the requirements the following materials used, Reference materials, Personal computer, Software package for analysis include ANSYS 2020 R2, Scilab for analysis, Microsoft Office to compile document and Wondershare EdrawMax for some drawings.

3.3 Procedure

The steps undertaken in the present study to achieve the above-mentioned objectives are as follows:

a) Carry out extensive literature review

b) Collection of required resources: This study needs the following materials including specialized software packages.

- Reference Materials
- Ansys 2020 R2
- Wondershare EdrawMax
- Microsoft Office
- Scilab v 6.1.0

c) Modelling of cantilever beam

- Two-dimensional structural analysis for the beam is done using AEM and FEM.
- Model and analyze the structure using finite element software Ansys 2020 R2 and a numerical software used for AEM.

2D shape of the finite element can be both triangular and quadrilateral for different kinds of approximations of element order Linear, Quadratic and Higher, and so forth. Degrees of freedoms in FEM at nodes are much greater than AEM degrees of freedoms at the center of elements, in 2D AEM elements can be square or rectangle to compare the result of the two numerical methods a 2D linear quadrilateral element is selected.

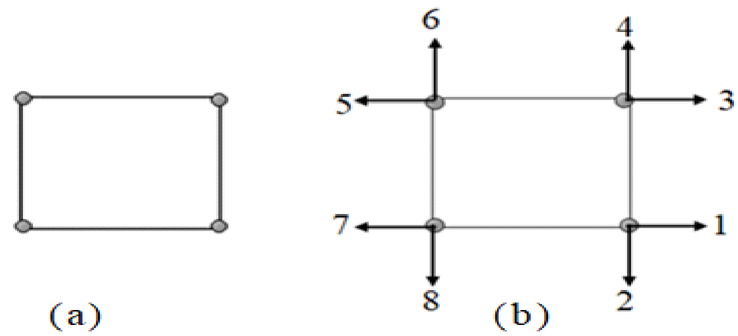


Figure 3.1: 2D quadrilateral element in FEM (D.Logan, 2012)

To investigate the convergence of AEM and FEM, a classic cantilever beam problem is selected and analyzed. A 2D quadrilateral element with eight degrees of freedom for FEM shown in figure 3.2 and numerical results are compared with 2D AEM elements shown in figure 2.1 (a). The analysis for AEM and FEM is done using Scilab codes and Ansys 2020 R2 respectively. To investigate the effects of selected parameters in AEM, to study the effect of number of springs a cantilever beam shown in figure 4.1 is selected and analyzed by varying the springs and element numbers and compare the numerical results for each case for the analytical result. The same beam is selected to study the 1D and 2D discretization using AEM and six random beams shown in figure 4.21 selected to compare the convergence and validity of the two numerical methods AEM and FEM.

d) Review previous works on blast loading analysis for the two methods i.e. AEM and FEM.
e) Conclusion and recommendation: Conclusion will be drawn based on the findings and recommendation will be given for further works.

After the analysis was completed, comparison of the numerical results obtained with respect to analytical results, for different cases graphs are plotted. Finally, conclusion was drawn and recommendation was suggested for future studies.

CHAPTER FOUR

4.0 Numerical Validation

This section presents procedures for analyzing structures using AEM, steps are summarized below:

1. Divide the structure into discrete elements.
2. Using the material and geometric properties determine normal and shear stiffness coefficients.
3. Determine stiffness matrix for each pair of springs.
4. Replace spring stiffness by an equivalent spring by adding stiffness of springs which connect same DOF's.
5. Determine global stiffness matrix by assembling the equivalent spring stiffnesses.
6. Determine displacement and reactions using the governing system equations.
7. Determine normal and shear stress along depth.
8. Draw the shapes of normal and shear stress using the stresses determined in step 7.

4.1 Analysis of simple structure using AEM

A cantilever beam with tip load is selected to be analyzed into two cases, first is discretize the beam into five elements along the longitudinal direction and second discretization of three elements along transversal direction in addition to case one. The beam is 4 m long and its cross section is 350 mm ×350 mm as shown in Fig. 4.1. The Modulus of elasticity and Poisson's ratio of the material are 25000 N/mm^2 and 0.2 respectively. 10 number of springs are provided along all faces in the case of AEM. The analysis by AEM is done using a numerical software Scilab.

In this study the effect of reinforcement is not considered and beams and loads are selected randomly there was no consideration and limitation of fracture/cracking load of a plain concrete cantilever beam.

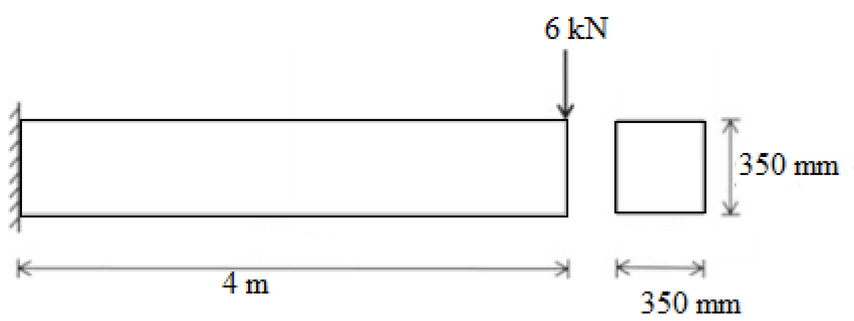


Figure 4.1- Cantilever beam considered for the analysis

Using AEM, the cantilever beams is analyzed in two different ways: -

1. Discretization of beam along Longitudinal direction only
2. Discretization of beam along both sides, along longitudinal and transverse

Case 1: Discretization of beam along Longitudinal direction only

The Scilab codes used to analyze the cantilever beam in 1D is shown in Appendix A.

Define Units Used:

- ✓ Force: N
- ✓ Length: mm

Given Geometric Properties:

- ✓ b = 350 mm
- ✓ D = 350 mm
- ✓ L = 4000 mm
- ✓ P = 6000 N
- ✓ E = 25000 N/mm^2
- ✓ v = 0.2
- ✓ $G = \frac{E}{2(1+v)} = 10416.667\ \mathrm{N}/mm^2$

For first case 5 elements along longitudinal direction and n_y =10 is adopted.

Where: n_y is number of springs in each face of element

d is the distance between springs

$$a = \frac{L}{5} = \frac{4000}{5} = 800\ mm$$

$$d = \frac{D}{n_y} = \frac{350}{10} = 35\ mm$$

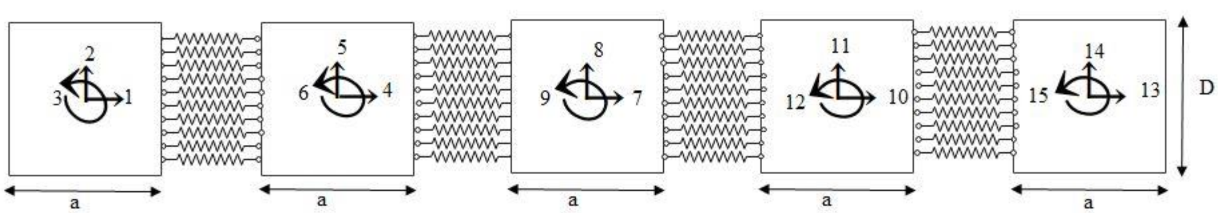


Figure 4.2-Five elements with 15 DOF's and 10 springs connecting adjacent elements

Using Eq. 2 Normal and Shear stiffness are determined:

$$k_n = \frac{E\times d\times t}{a} = \frac{25000\times 35\times 350}{800} = 382812.5\,\frac{N}{mm}$$

$$k_s = \frac{G\times d\times t}{a} = \frac{10416.667\times 35\times 350}{800} = 159505.21\,\frac{N}{mm}$$

Each pair of springs has shear and normal springs as shown in Fig. (a). Using Eq. (4) stiffness matrix is determined for each pair of springs by determining (x,y) values for each spring with respect to the centroid as shown in Figure 4.1 (b).

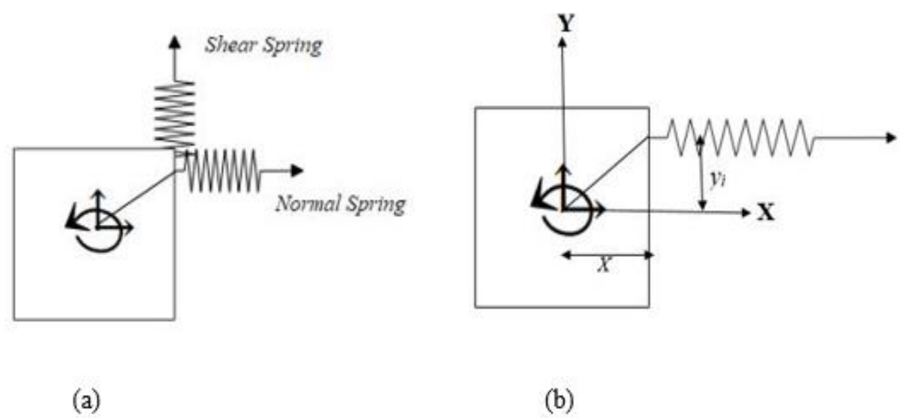


Figure 4.3- (a)- Shear and Normal springs

(b)-Values of x and y point of contact of spring at element edge with respect to centroid of selected element

Determining stiffness matrix for the ten pair of springs.

Spring 1:

$$x_1 = \frac{a}{2} = \frac{800}{2} = 400\;mm$$

$$y_1 = \frac{D}{2} - \frac{d}{2} = 157.5\,mm$$

Using Eq. 4 the stiffness matrix for spring 1 size (6 × 6).

$$k1 = \left\{\begin{array}{cccccc} 382812.5 & 0 & -60292969 & -382812.5 & 0 & 60292969 \\ 0 & 159505.21 & 63802083 & 0 & -159505.21 & 63802083 \\ -60292969 & 63802083 & 3.502D+10 & 60292969 & -63802083 & 1.602D+10 \\ -382812.5 & 0 & 60292969 & 382812.5 & 0 & -60292969 \\ 0 & -159505.21 & -63802083 & 0 & 159505.21 & -63802083 \\ 60292969 & 63802083 & 1.602D+10 & -60292969 & -63802083 & 3.502D+10 \end{array}\right\}$$

Similarly, for all other pair of springs (10 springs) the stiffness matrix will be determined using the same procedures.

After spring stiffness for all pair of springs are determined, for springs which connect similar DOF's are summed and replaced by an equivalent spring(k) as shown in Figure 4.4.

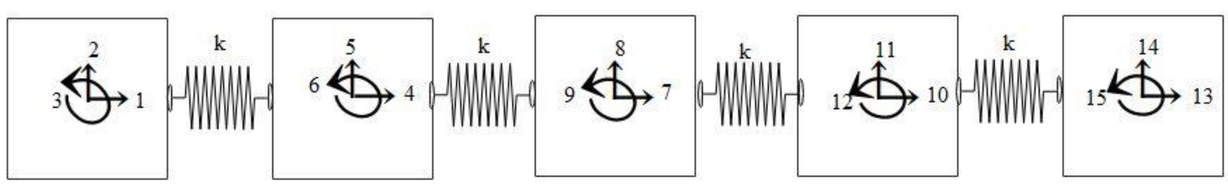


Figure 4.4-10 springs connecting elements replaced by an equivalent spring k

$$k = \left\{\begin{array}{cccccc} 3828125 & 0 & 0 & -3828125 & 0 & 0 \\ 0 & 1595052.1 & 6.380D+08 & 0 & -1595052.1 & 6.380D+08 \\ 0 & 6.380D+08 & 2.939D+11 & 0 & -6.380D+08 & 2.165D+11 \\ -3828125 & 0 & 0 & 3828125 & 0 & 0 \\ 0 & -1595052.1 & -6.380D+08 & 0 & 1595052.1 & -6.380D+08 \\ 0 & 6.380D+08 & 2.165D+11 & 0 & -6.380D+08 & 2.939D+11 \end{array}\right\}$$

Assemble the stiffness matrix by considering the DOF's the springs connect, size of global stiffness matrix equal to DOF's.

$$K_G = \begin{bmatrix}
3828125 & 0 & 0 & -3828125 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 1595052.1 & 6.380D+08 & 0 & -1595052.1 & 6.380D+08 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 6.380D+08 & 2.939D+11 & 0 & -6.380D+08 & 2.165D+11 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
-3828125 & 0 & 0 & 7656250 & 0 & 0 & -3828125 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & -1595052.1 & 2.939D+11 & 0 & 3190104.2 & 0 & 0 & -1595052.1 & 6.380D+08 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 6.380D+08 & 2.165D+11 & 0 & 0 & 5.878D+11 & 0 & -6.380D+08 & 2.165D+11 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & -3828125 & 0 & 0 & 7656250 & 0 & 0 & -3828125 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & -1595052.1 & -6.380D+08 & 0 & 3190104.2 & 0 & 0 & -1595052.1 & 6.380D+08 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 6.380D+08 & 2.165D+11 & 0 & 0 & 5.878D+11 & 0 & -6.380D+08 & 2.165D+11 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & -3828125 & 0 & 0 & 7656250 & 0 & 0 & -3828125 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & -1595052.1 & -6.380D+08 & 0 & 3190104.2 & 0 & 0 & -1595052.1 & 6.380D+08 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 6.380D+08 & 2.165D+11 & 0 & 0 & 5.878D+11 & 0 & -6.380D+08 & 2.165D+11 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & -3828125 & 0 & 0 & 3828125 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & -1595052.1 & -6.380D+08 & 0 & 1595052.1 & -6.380D+08 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 6.380D+08 & 2.165D+11 & 0 & -6.380D+08 & 2.939D+11
\end{bmatrix}$$

Applying Boundary conditions and determining force vector.

$$[F] = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ -6000 \\ -2400000 \end{bmatrix}$$

Using Eq.6, the displacement can be determined:

$$[\Delta] = [K_G]^{-1} \times [F]$$

$$[\Delta]_{V,tip} = 2.4964369\ mm$$

Procedure for determining reactions using AEM is presented here, after determining the displacement vector reaction forces at the support will be determined by using the equation:

$$[R]=[K][U] \qquad (7)$$

Where: $[R]$ is the reaction forces vector

$[K]$ is the global stiffness matrix (without deducting stiffness by applying B.C)

$[U]$ is the displacement vector.

Using eq. (7) the corresponding reaction forces at the support position for five elements along the longitudinal direction is shown below.

$$[R]=\begin{Bmatrix} 0 \\ 6000 \\ 21600000 \end{Bmatrix}$$

*Forces are in N and Moment is in Nmm.

When number of elements increased to 20 elements along longitudinal direction and calculating reaction forces:

$$[R]=\begin{Bmatrix} 0 \\ 6000 \\ 23400000 \end{Bmatrix}$$

These indicates to get the vertical displacement, reaction forces to be of greater accuracy we must use a greater number of elements until the numerical results converge to numerical results.

Case 2: Discretization of beam along both sides, along longitudinal and transverse

Five elements along the longitudinal and three elements along transverse direction are used as shown in Figure 4.5. Each element has three DOF's at the centroid of the element, total DOF's is forty-five. The Scilab codes used to analyze the cantilever beam in 2D is shown in Appendix B.

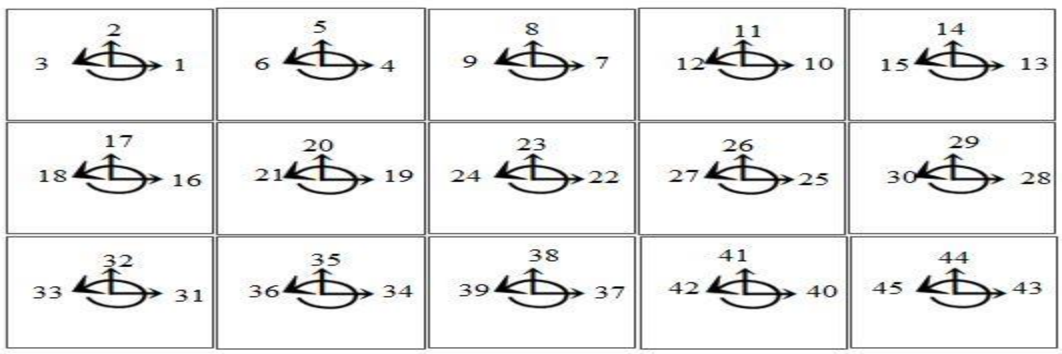


Figure 4.5- Two-dimensional discretization along longitudinal and transverse direction

Elements are connected at adjacent location by springs assuming ten springs in each face, adjacent elements are connected as shown in the Fig. 4.6.

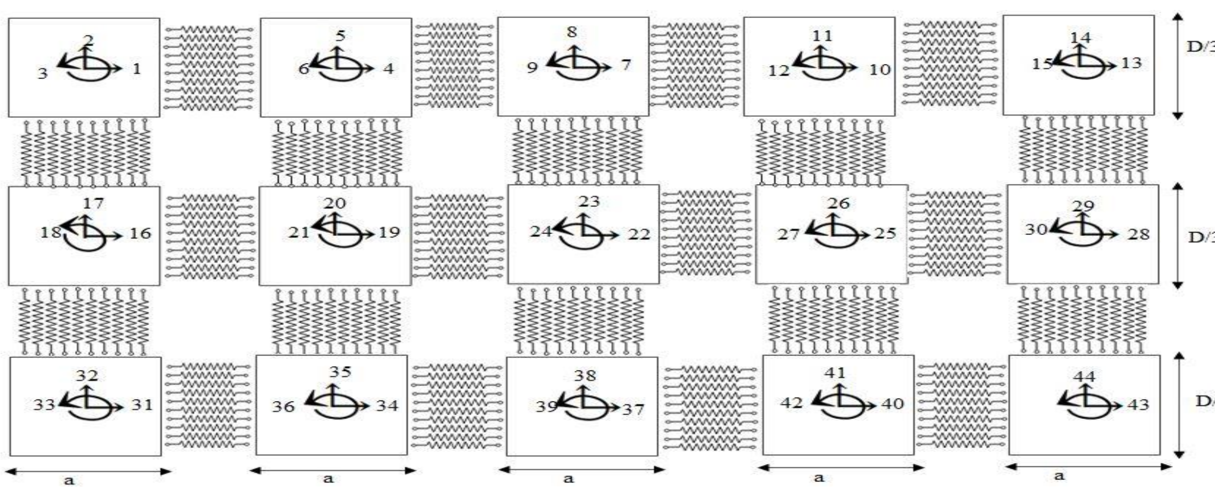


Figure 4.6-Ten pair of springs connecting adjacent elements

When assembling spring stiffness, stiffness along each direction i.e., along transverse and longitudinal should be considered which connect same DOF's.

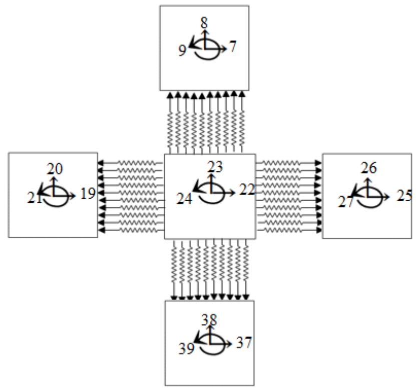


Figure 4.7- Pair of springs along transverse and longitudinal directions

- ✓ Stiffness along longitudinal direction:

$$k_{n}= \frac{E\times d\times t}{a} = \frac{25000\times 11.666667\times 350}{800} = 127604.17\ \frac{N}{mm}$$

$$k_{s}= \frac{G\times d\times t}{a} = \frac{10416.667\times 11.666667\times 350}{800} = 53168.403\ \frac{N}{mm}$$

$$k_h = \begin{Bmatrix} 1276041.7 & 0 & 8.382D\text{-}09 & 8.364D\text{+}10 & 0 & -8.382D\text{-}09 \\ 0 & 531684.03 & 8.382D\text{-}09 & 0 & -531684.03 & 2.127D\text{+}08 \\ 8.382D\text{-}09 & 2.127D\text{+}08 & 8.650D\text{+}10 & -8.382D\text{-}09 & -2.127D\text{+}08 & 8.364D\text{+}10 \\ -1276041.7 & 0 & -8.382D\text{-}09 & 1276041.7 & 0 & 8.382D\text{-}09 \\ 0 & -531684.03 & -2.127D\text{+}08 & 0 & 531684.03 & -2.127D\text{+}08 \\ -8.382D\text{-}09 & 2.127D\text{+}08 & 8.364D\text{+}10 & 8.382D\text{-}09 & -2.127D\text{+}08 & 8.650D\text{+}10 \end{Bmatrix}$$

- ✓ Stiffness along transverse direction:

$$k_{n}= \frac{E\times d\times t}{a} = \frac{25000\times 80\times 350}{116.66667} = 6000000\ \frac{N}{mm}$$

$$k_{s=\frac{G\times d\times t}{a}} = \frac{10416.667\times 80\times 350}{116.66667} = 2500000\ \frac{N}{mm}$$

$$k_d = \begin{Bmatrix} 60000000 & 0 & 0 & -60000000 & 0 & 0 \\ 0 & 25000000 & 1.458D{+}09 & 0 & -25000000 & 1.458D{+}09 \\ 0 & 1.458D{+}09 & 3.253D{+}12 & 0 & -1.458D{+}09 & -3.083D{+}12 \\ -60000000 & 0 & 0 & 60000000 & 0 & 0 \\ 0 & -25000000 & -1.458D{+}09 & 0 & 25000000 & -1.458D{+}09 \\ 0 & 1.458D{+}09 & -3.083D{+}12 & 0 & -1.458D{+}09 & 3.253D{+}12 \end{Bmatrix}$$

Stiffness matrix is derived for springs in the longitudinal direction, for springs along transverse direction the DOF's are aligned along different direction and must be transformed using the transformation matrix T.

$$T = \begin{Bmatrix} l & m & 0 & 0 & 0 & 0 \\ -m & l & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & l & m & 0 \\ 0 & 0 & 0 & -m & l & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{Bmatrix}$$

$l = \cos\varphi$

$m = \sin\varphi$

$\varphi = 90^o$

$$k_v = T^T \times \kappa_d \times T$$

$$k_v = \begin{Bmatrix} 25000000 & 0 & -1.458D{+}09 & -25000000 & 0 & -1.458D{+}09 \\ 0 & 60000000 & 0 & 0 & -60000000 & 0 \\ -1.458D{+}09 & 0 & 3.253D{+}12 & 1.458D{+}09 & 0 & -3.083D{+}12 \\ -25000000 & 0 & 1.458D{+}09 & 25000000 & 0 & 1.458D{+}09 \\ 0 & -60000000 & 0 & 0 & 60000000 & 0 \\ -1.458D{+}09 & 0 & -3.083D{+}12 & 1.458D{+}09 & 0 & 3.253D{+}12 \end{Bmatrix}$$

Similar to discretization along direction stiffness matrix for each pair of springs will be calculated and summing this stiffness of this pair of springs will replaced by an equivalent stiffness, K_h stiffness along longitudinal and K_v stiffness along transverse direction as shown in Fig. 4.8.

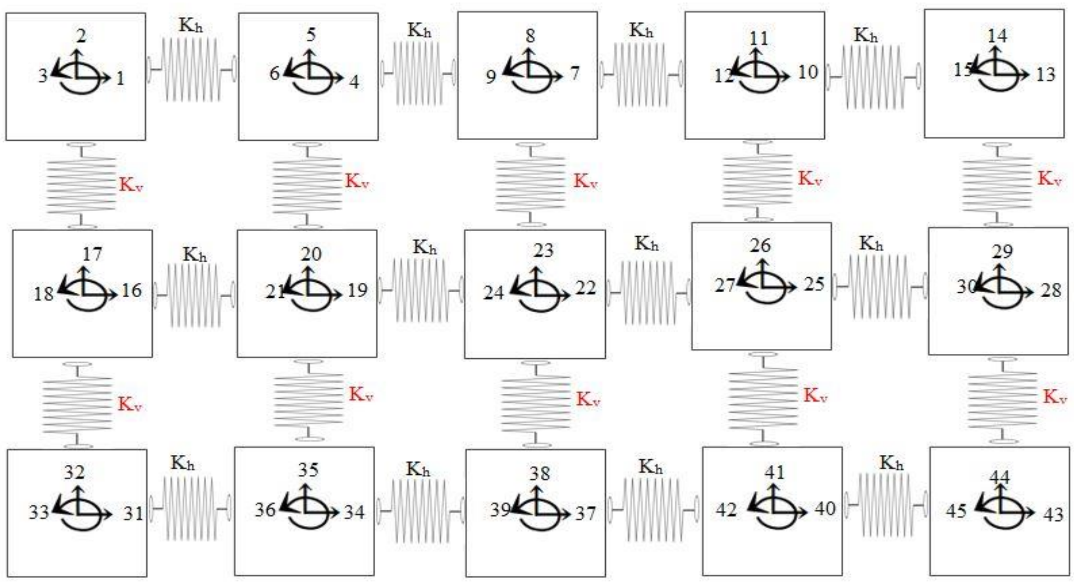


Figure 4.8- Ten pair of springs replaced by equivalent springs K_h for lateral stiffness and K_v for transversal stiffness of springs

Depending on the DOF's these springs connect stiffness matrix will be assembled with size (45 × 45).

Applying boundary conditions and the force vector is a transpose of $[F]$:

$[F]=[$0. 0. 0. 0. 0. 0. 0. 0. 0. 0.–6000.–350000. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0. 0.0. 0. 0.$]$

Using Eq.6, the displacement can be determined:

$$[\Delta] = [K_G]^{-1} \times [F]$$

$$[\Delta]_{V,tip} = 2.0667873\ mm$$

Using Eq. 7 reactions can be determined:

$$[R]=[K][U]$$

$$[R]=\begin{Bmatrix} 0 \\ 6000 \\ 4312542.8 \end{Bmatrix}$$

Using 5 elements along longitudinal and 3 elements along transversal while calculating the reactions it is exaggerated from the analytical result. When number of elements increased 10 elements along longitudinal keeping the same number of transversal elements the reactions will be:

$$[R]=\begin{Bmatrix} 0 \\ 6000 \\ 2360315.5 \end{Bmatrix}$$

Similarly, to get the vertical displacement and reaction forces converge to the analytical value we must use a greater number of elements along both directions.

4.1.1 Analytical displacement

The theoretical vertical displacement of the cantilever beam at the free end due to the concentrated load is calculated as shown below,' The deflection of the cantilever beam at the end is determined analytically by considering the effect of shear also, for the derivation of see (Goodier, 1951).

Analytical Result:

$$\Delta = \frac{PL^3}{3EI} + \frac{Pc^2L}{2GI} = 4.1178035\ mm$$

In this study a classic cantilever beam is selected to be analyzed and is modeled and analyzed for linear static behavior using finite element software ANSYS R2 2020. The results and discussions presented here are focused on tip deformation of a cantilever beam that support the objectives of the present study. Parametric study on investigation of effects of number of springs, discretization types, convergence test is done by varying the number of elements conducted. Once the numerical results found from the two methods will be compared with the analytical result.

4.1.2 Analytical rotation

The theoretical rotation of the cantilever beam at the free end due to the concentrated load is calculated below

$$\theta = \frac{d_y}{d_x} = \frac{FL^2}{2EI} = \frac{(6kN)x(4m)^2}{2(25x10^6 kN/m^2)\frac{(0.35m)^4}{12}} = 0.015rad$$

Analytical displacement using AEM, for 100 elements selecting the 3rd DOF from 100 th element and using trigonometric relation the rotation at the end is 0.016, AEM gives accurate rotation to check whether rotation depends on number of elements. A comparison of vertical displacement and rotation result of AEM compared in the table 4.1 below.

Comparison of vertical displacement and rotation at the end using AEM (10 pair of springs used) and analytical result:

Table 4-1 Comparison of vertical displacement and rotation

Number of elements in AEM	*Analytical results*		*AEM results*	
	Vertical Displacement (m)	Rotation (rad)	Vertical Displacement (m)	Rotation (rad)
20	0.0041	0.015	0.0042	0.016
50	0.0041	0.015	0.0042	0.016
100	0.0041	0.015	0.0042	0.016
150	0.0041	0.015	0.0042	0.016
200	0.0041	0.015	0.0042	0.016

From Table 4.1 it can be clearly seen than AEM accurately predict vertical displacement and rotation at the free end of a cantilever beam.

4.2 Parametric study

In order to guarantee the numerical results of AEM a parametric study is conducted by performing a similar static structural analysis’s on the selected cantilever beam shown on Figure 4.1 by varying number of elements, to study the influence of number of springs connecting elements and effect of discretization (which include two cases, case one: beam discretized along the longitudinal direction only and case two: it is discretized in both longitudinal and transversal direction.) on normal stress distribution, shear stress distribution and vertical tip displacement of a cantilever beam. Numerical results of AEM for each case are plotted against number of elements and compared with the analytical result to check the convergence of the numerical solution.

4.2.1 Effect of number of elements on normal and shear stress distribution

Comparison of normal and shear stress results of AEM with analytical values by varying number of elements and type of discretization is conducted in this section. A series of analysis has been conducted by varying number of elements and discretization types. It was found out the normal stress distribution doesn't depend on either number of elements nor type of discretization, it's identical with the analytical result of normal stress due to bending. No need to perform parametric study on normal stress, distribution of normal stress along depth for different number of elements in AEM compared with analytical result is shown below.

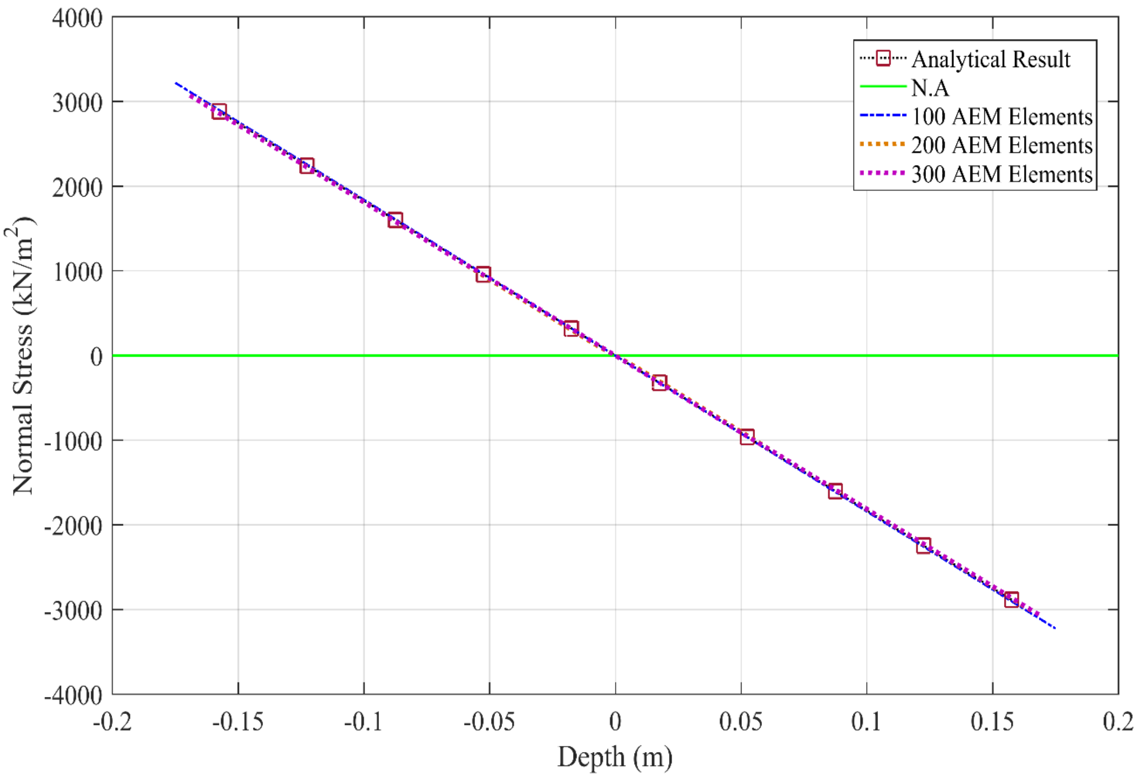


Figure 4.9 Normal stress distribution at 6 elements from support

As can be seen from Figure 4.9 for normal stress distribution along the depth is similar with AEM result no need to perform parametric study by varying number of elements and type of discretization. But for shear stress distribution a parametric study has been conducted.

For shear stress along the depth a series of analysis has been conducted utilizing AEM by varying number of elements and type of discretization and result for shear stress distribution has been plotted against the depth of element.

The shear distribution of a cantilever beam is determined analytically and also by utilizing AEM, for the derivation of determining shear stress in AEM see (A.S. Chepurnenko, 2018). During the analysis it was found out shear stress distribution is constant throughout depth of an element having a value of average shear stress, but the analytical shear stress distribution is a curve. Therefore, to get this curve in AEM greater number of elements along transversal direction should be used.

The parametric study conducted by keeping the number of elements along the longitudinal 50 elements and varying number of elements along transversal direction by 1,2,3,5,10 and 15number of elements along the transversal direction while the total number of elements are 50, 100, 150, 250, 500 and 750 respectively.

Determining the shear stress at the location of springs (10 pair of spring used) along the depth using AEM and results are compared with the analytical result. The results of AEM are compared with analytical result and summarized by plotting against depth of an element shown on Figure 4.11 (a)-(f) below.

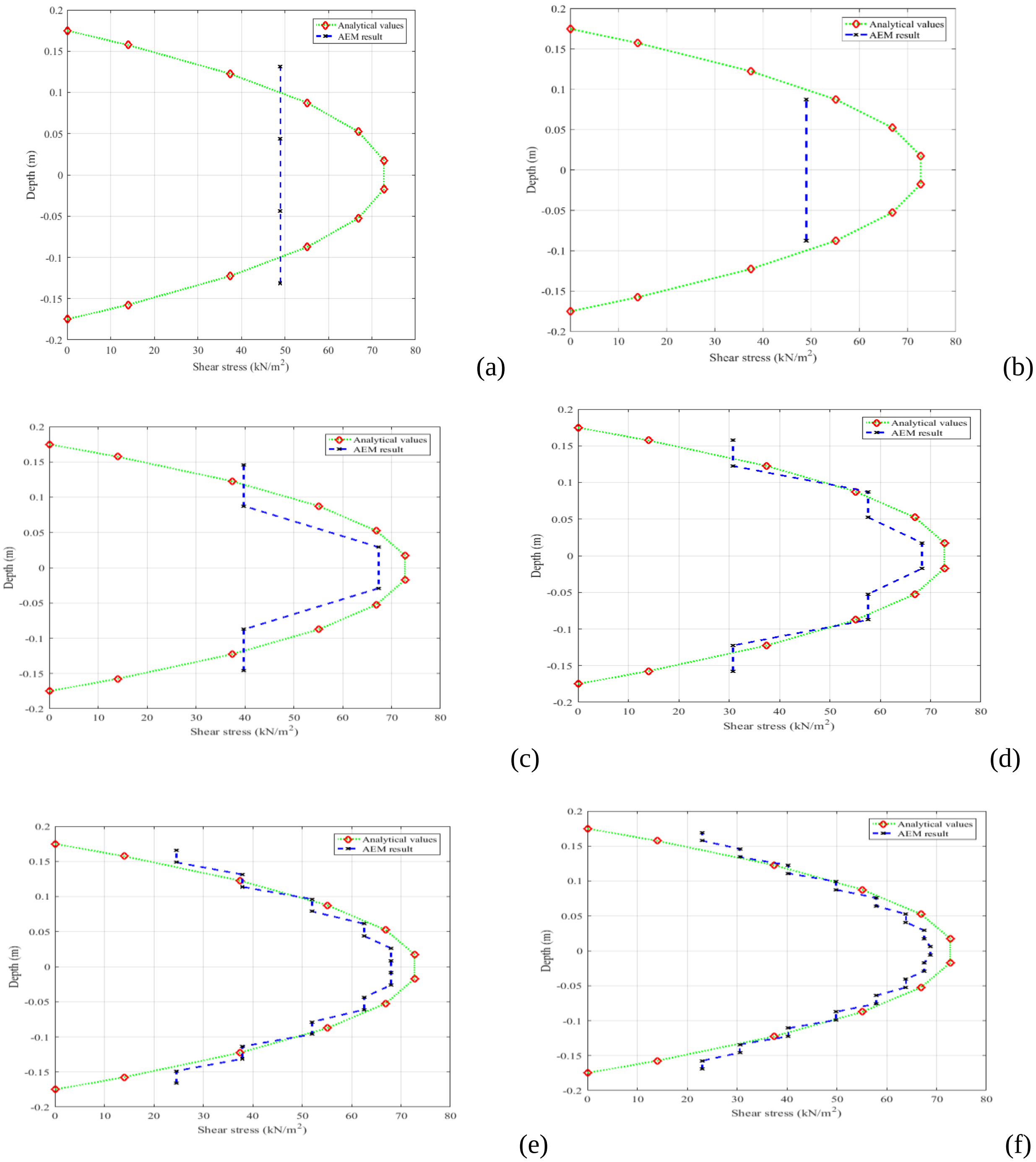


Figure 4.10 Shear stress distribution along depth

(a) NL 50, NT, 1 50 Elements **(b)** NL 50, NT, 2 100 Elements

(c) NL 50, NT 3, 150 Elements **(d)** NL 50, NT 5, 250 Elements

(e) NL 50, NT 10, 500 Elements **(f)** NL 50, NT 15, 750 Elements

*<u>**Where**</u>: NL: Number of elements along longitudinal direction

NT: Number of elements along transversal direction

On (a) discretization is only along longitudinal direction 50 elements shear stress distribution is constant and equal with (b) but (b) covers a greater depth when number of elements increased along the depth direction shear stress distribution is becoming a curve to be identical with the analytical result. For all analysis results are summed and summarized in Figure. 4.11 showing the effect of number of elements and type of discretization on shear stress distribution.

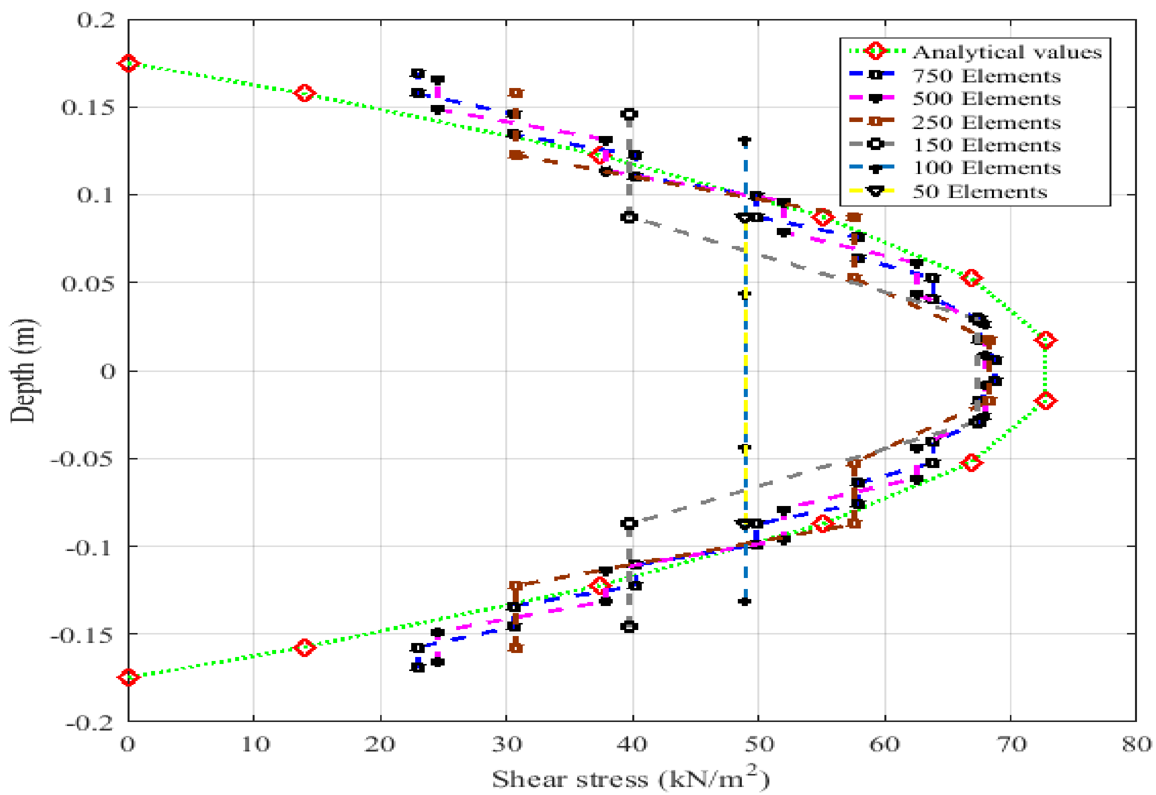


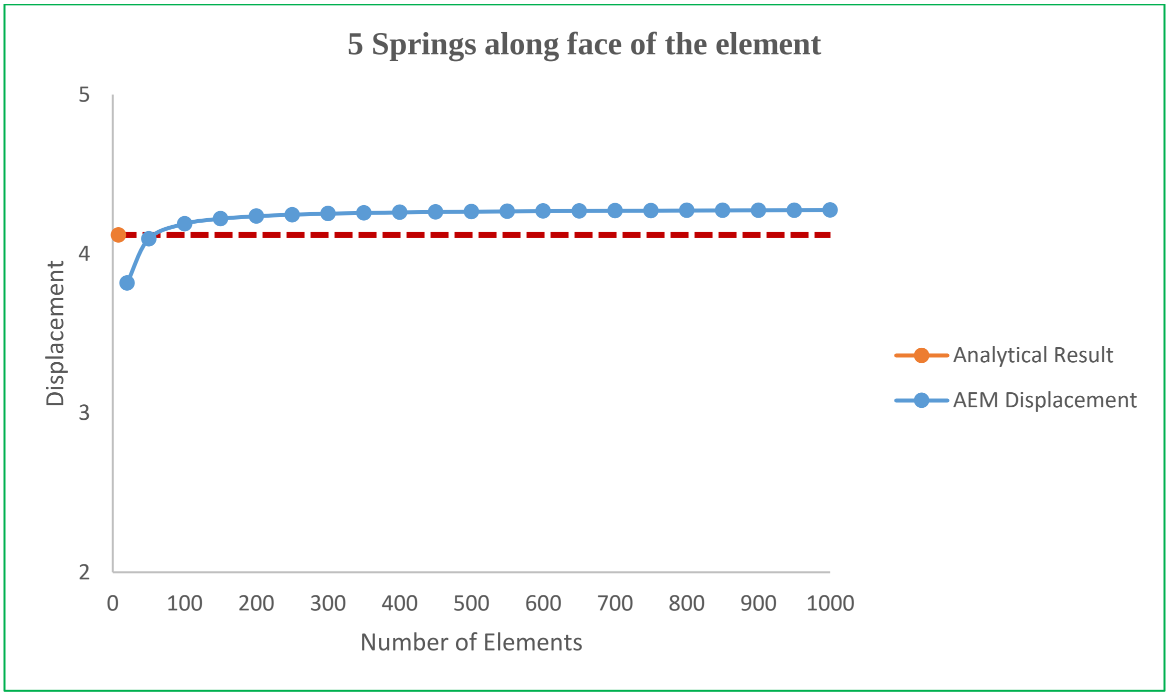
Figure 4.11 Summery of shear stress distribution along depth

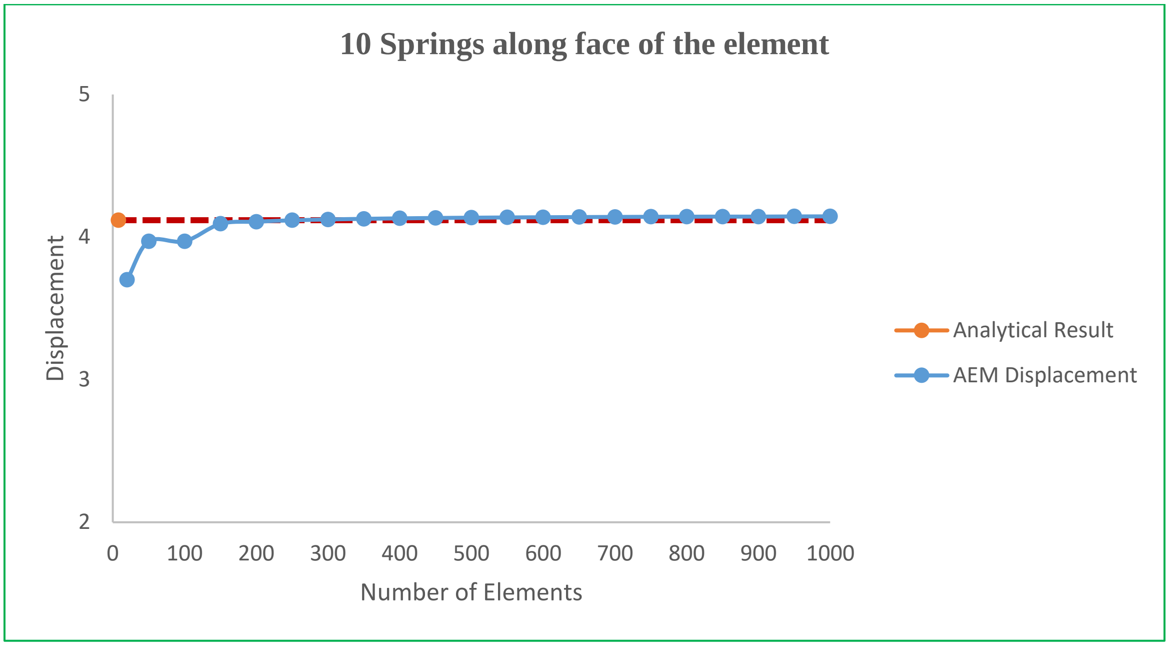
To increase accuracy of getting shear stress distribution along the depth smaller size elements and greater number of elements along depth direction should be used.

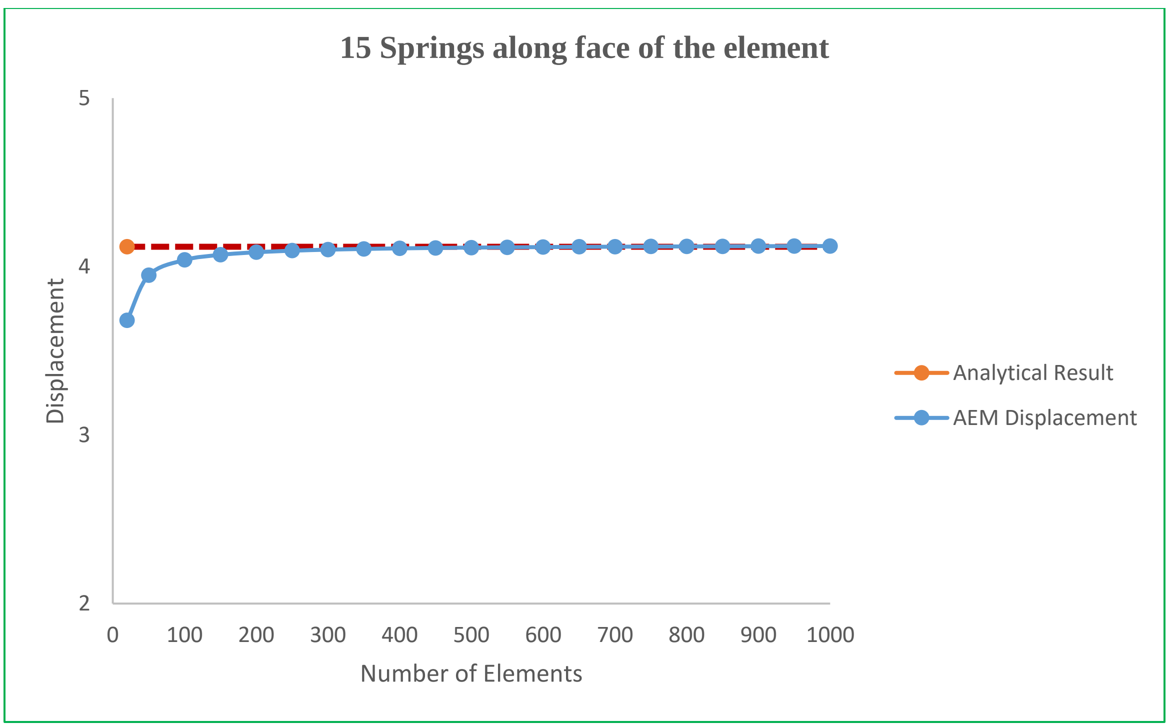
4.2.2 Effect of number of springs along face of elements

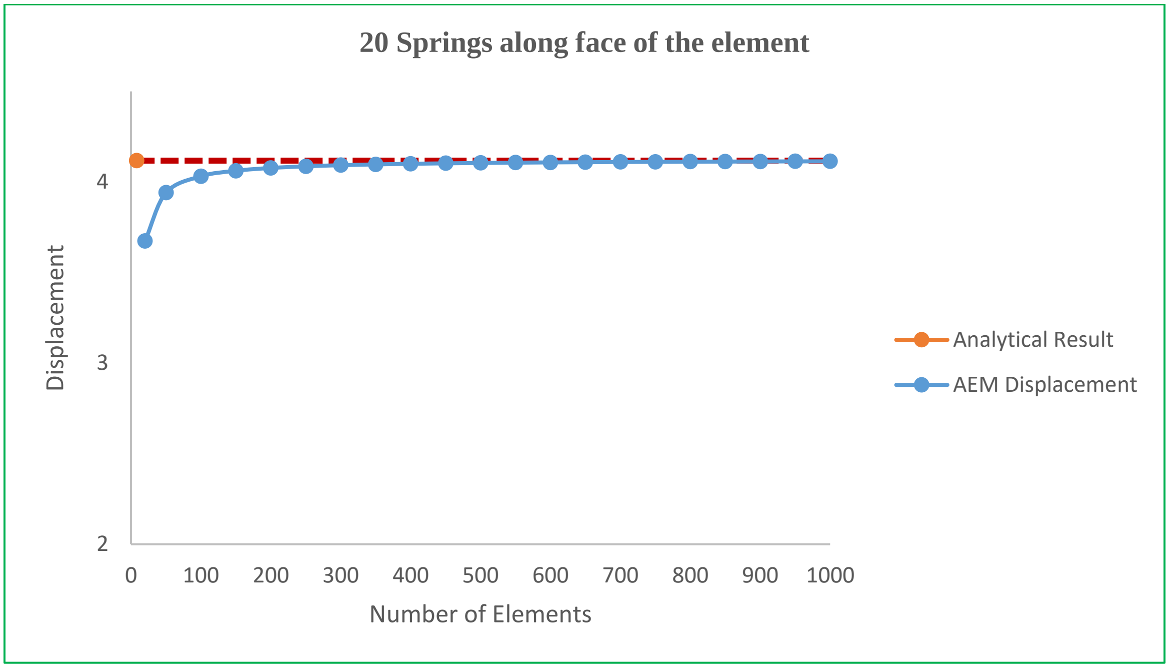
I. <u>Investigation of effect of number of springs discretization along the longitudinal direction</u>:

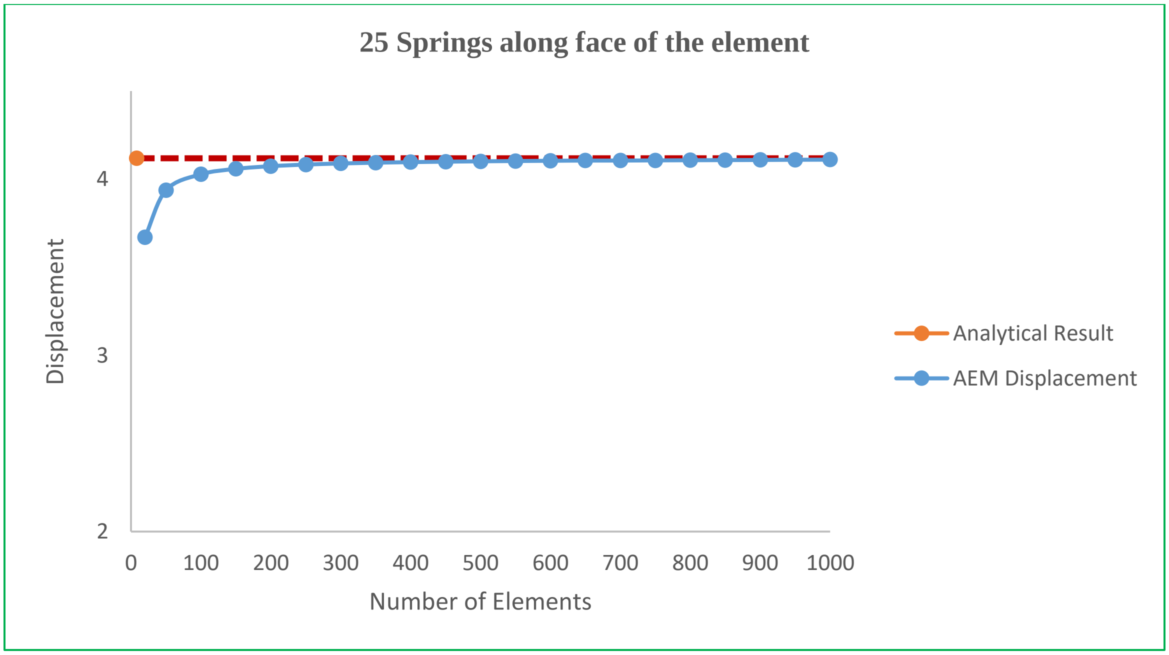
In AEM springs on element faces provide connection between adjacent elements. To investigation of the effect of number of springs on face of each element *5,10,15,20,25,30,35,40,45* and *50* number of springs on each face of an elements are selected. Series of analysis is performed on a 2D linear static structural analysis by utilizing AEM by varying the selected spring numbers.

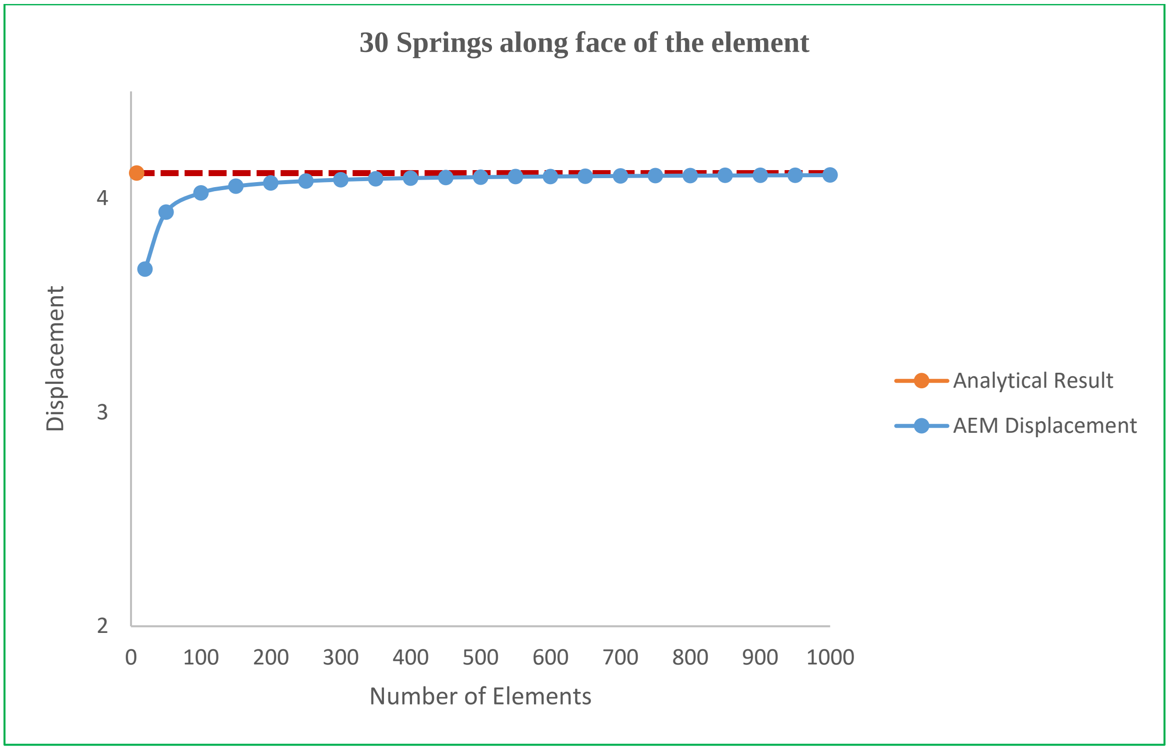


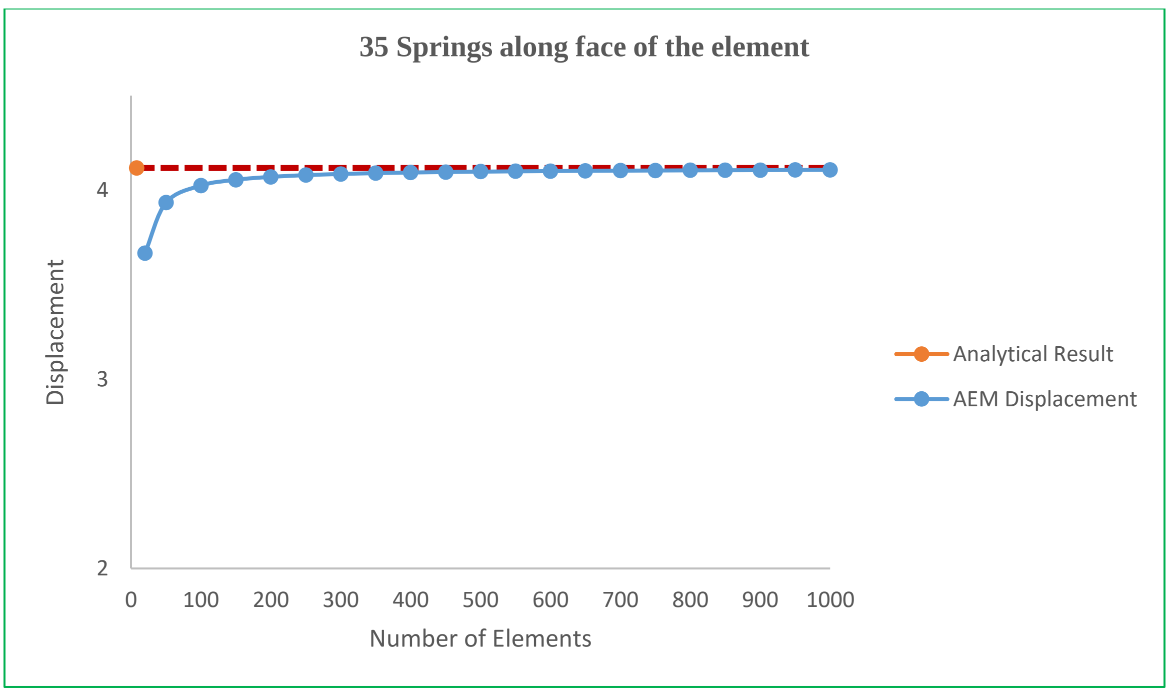

10 Springs along face of the element
Displacement
5
4
3
2
0
100
200
300
400
500
600
700
800
900
1000
Number of Elements
Analytical Result
AEM Displacement


15 Springs along face of the element
Displacement
5
4
3
2
0
100
200
300
400
500
600
700
800
900
1000
Number of Elements
Analytical Result
AEM Displacement

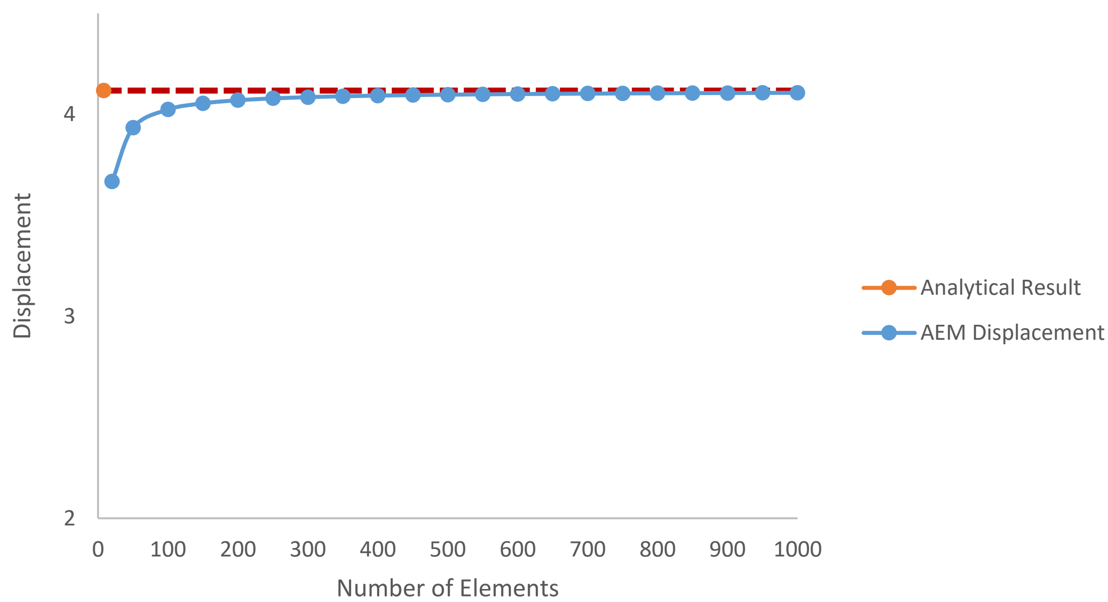

20 Springs along face of the element
Displacement
4
3
2
0
100
200
300
400
500
600
700
800
900
1000
Number of Elements
Analytical Result
AEM Displacement


25 Springs along face of the element
Displacement
4
3
2
0
100
200
300
400
500
600
700
800
900
1000
Number of Elements
Analytical Result
AEM Displacement

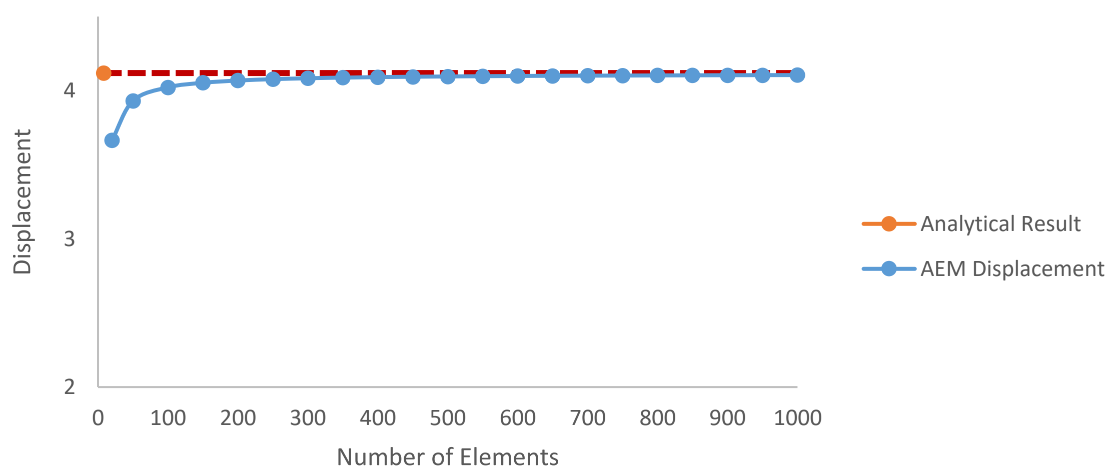

30 Springs along face of the element
Displacement
4
3
2
0
100
200
300
400
500
600
700
800
900
1000
Number of Elements
Analytical Result
AEM Displacement


35 Springs along face of the element
Displacement
4
3
2
0
100
200
300
400
500
600
700
800
900
1000
Number of Elements
Analytical Result
AEM Displacement

40 Springs along face of the element



45 Springs along face of the element



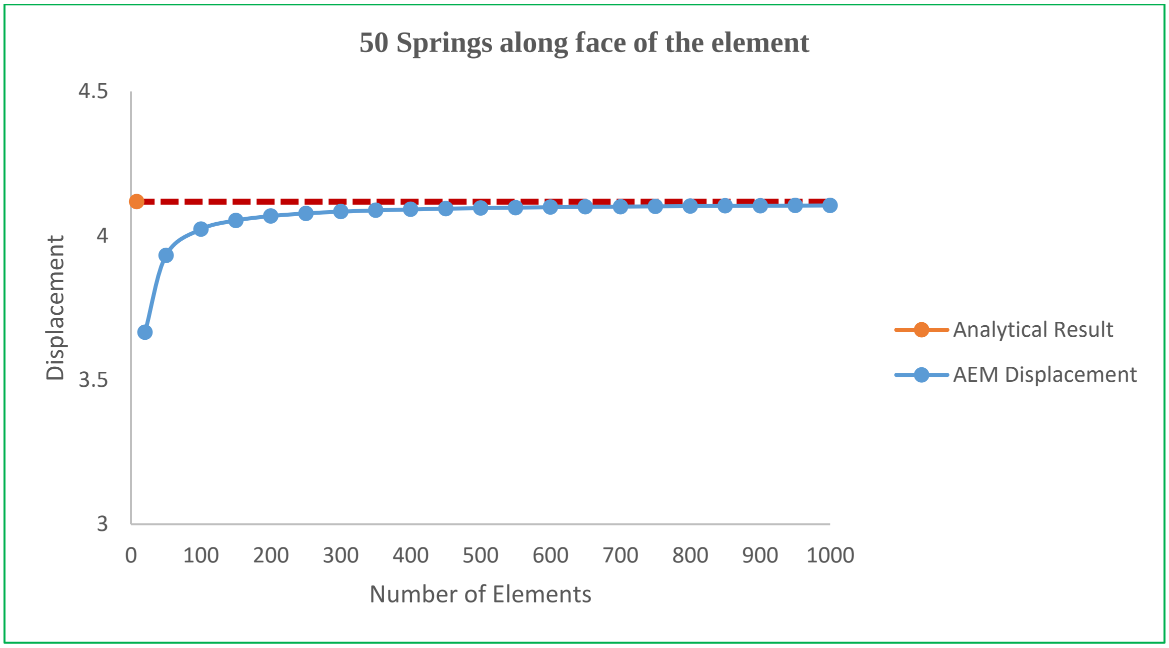


Figure 4.12 Displacement Vs Number of Elements for different number springs on each element face

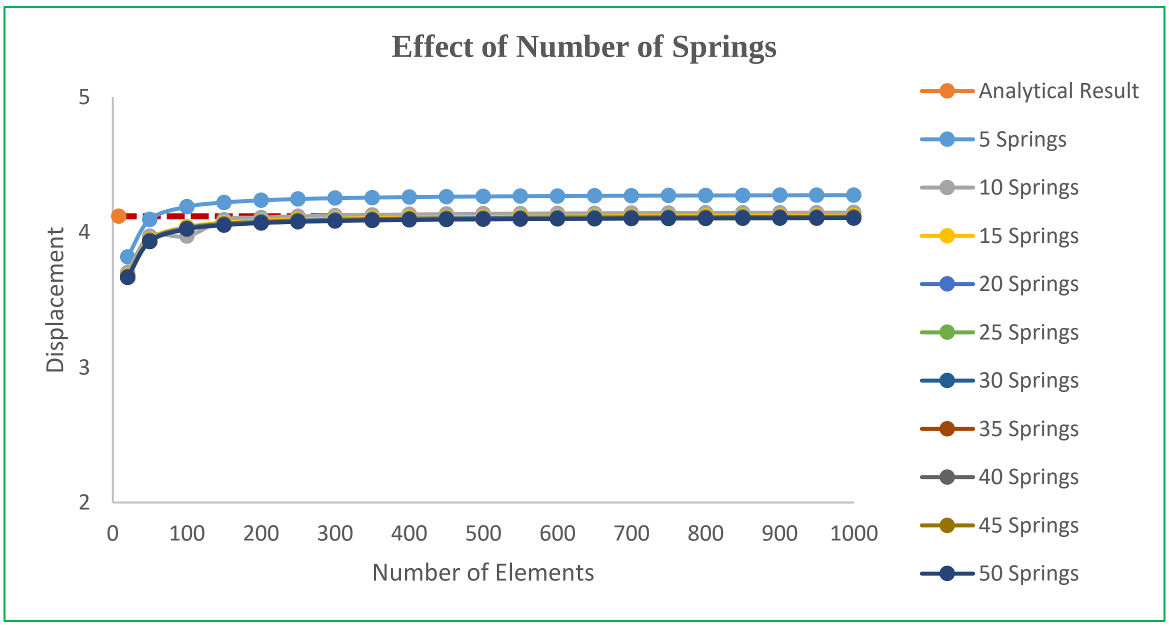


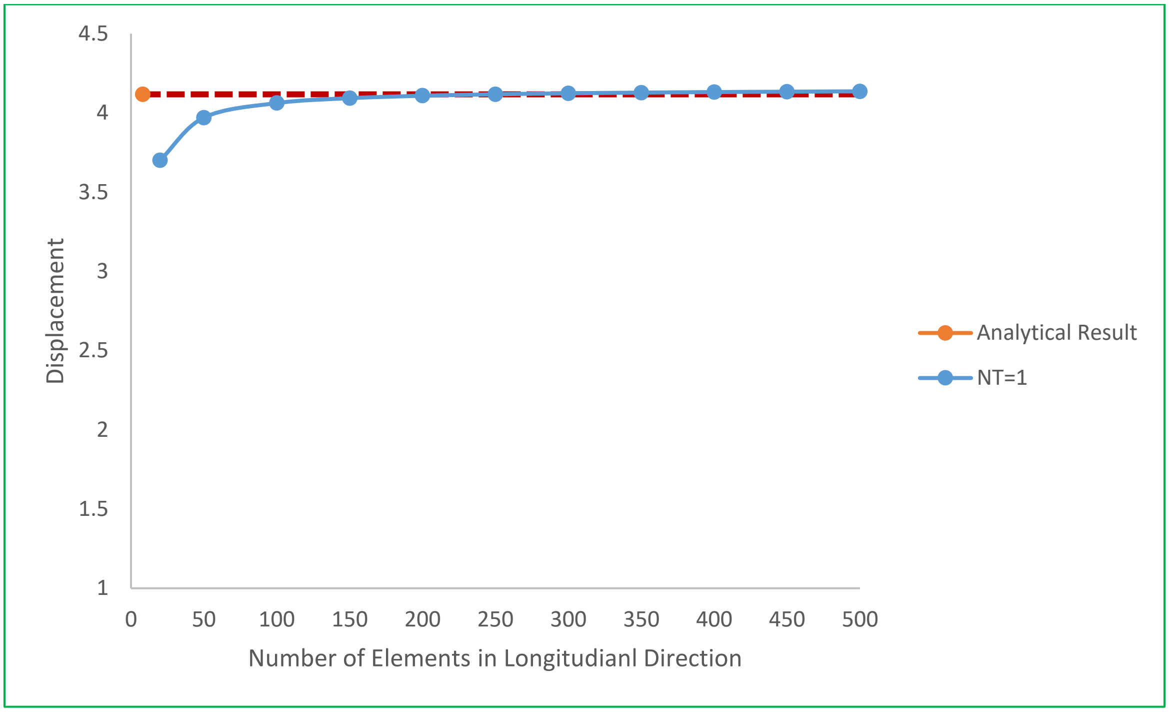
Figure 4.13 Combining Displacement Vs Number of Elements for different number springs

The effects of number of springs investigated for different number of springs, From the result it can be seen that when number of springs between elements is five it exaggerates the value which is above the analytical displacement but for springs number starting at ten and for above number of springs, the results are converged when 150 number of elements are taken along the length direction, the numerical result converges almost at the same number of elements. These indicates that numerical result does not vary much after the number of springs between the elements is ten. During the analysis while varying the spring number of springs the computational time does vary much, this indicate that increasing or decreasing the number of springs doesn't affect computational time. These indicate we can use a greater number of springs without affecting the computational time.

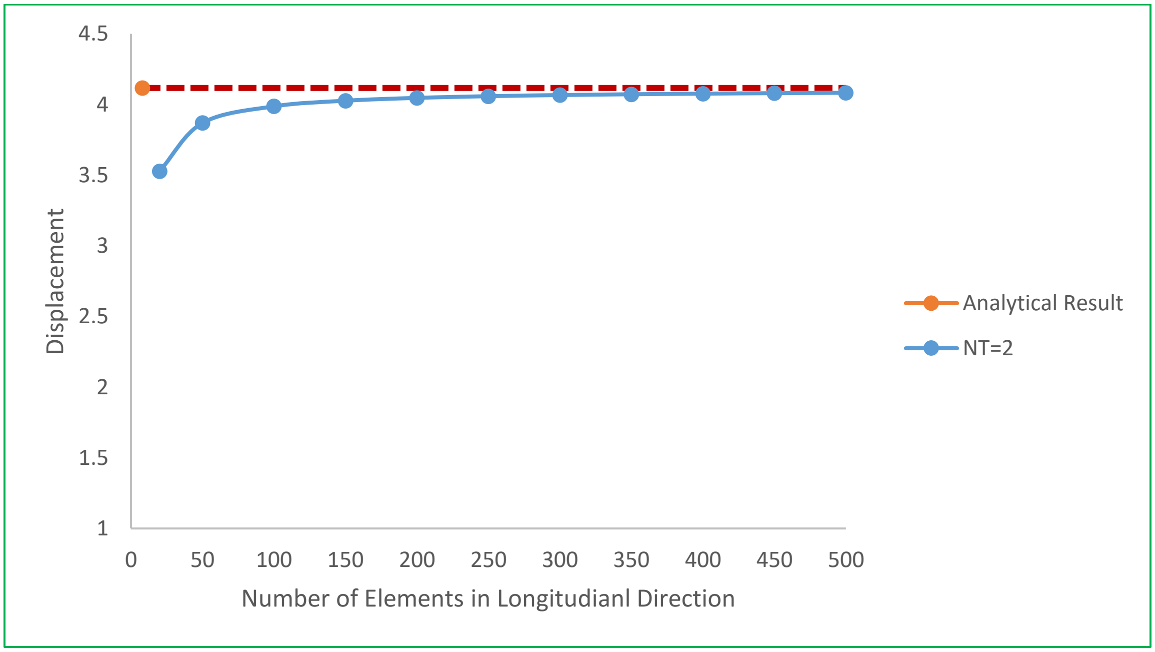
4.2.3 Effect of discretization on vertical displacement

II. Investigation of two-dimensional discretization: -

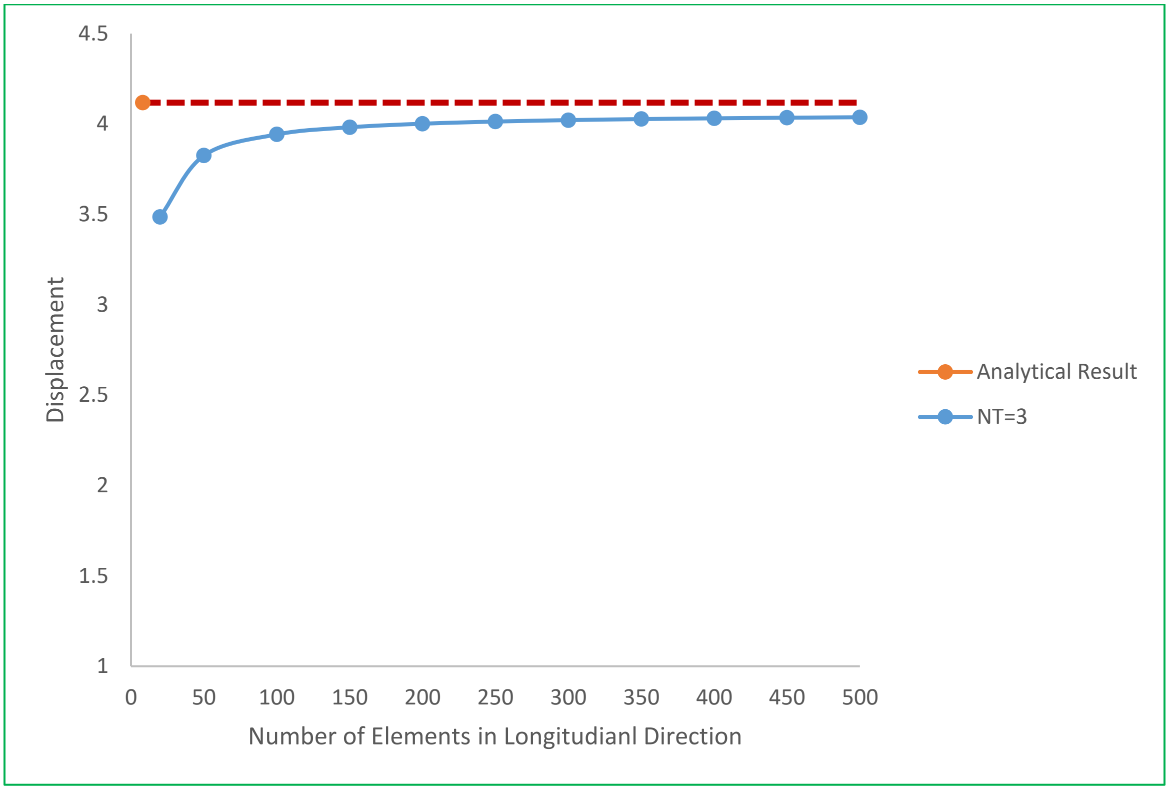
Investigation on the effect of two-dimensional discretization i.e. along both longitudinal and transverse direction is carried out by selection of ten number of springs on each face of element and 1,2,3 and 4 discretization of elements along the transversal direction. 2D numerical structural analysis is carried out, the numerical results for these selected criteria are plotted against number of elements along the longitudinal direction and compared with the analytical displacement to check convergency of the method.



4.5
4
3.5
3
2.5
2
1.5
1
Displacement
0
50
100
150
200
250
300
350
400
450
500
Number of Elements in Longitudianl Direction
Analytical Result
NT=1



4.5
4
3.5
3
2.5
2
1.5
1
Displacement
0
50
100
150
200
250
300
350
400
450
500
Number of Elements in Longitudianl Direction
Analytical Result
NT=2



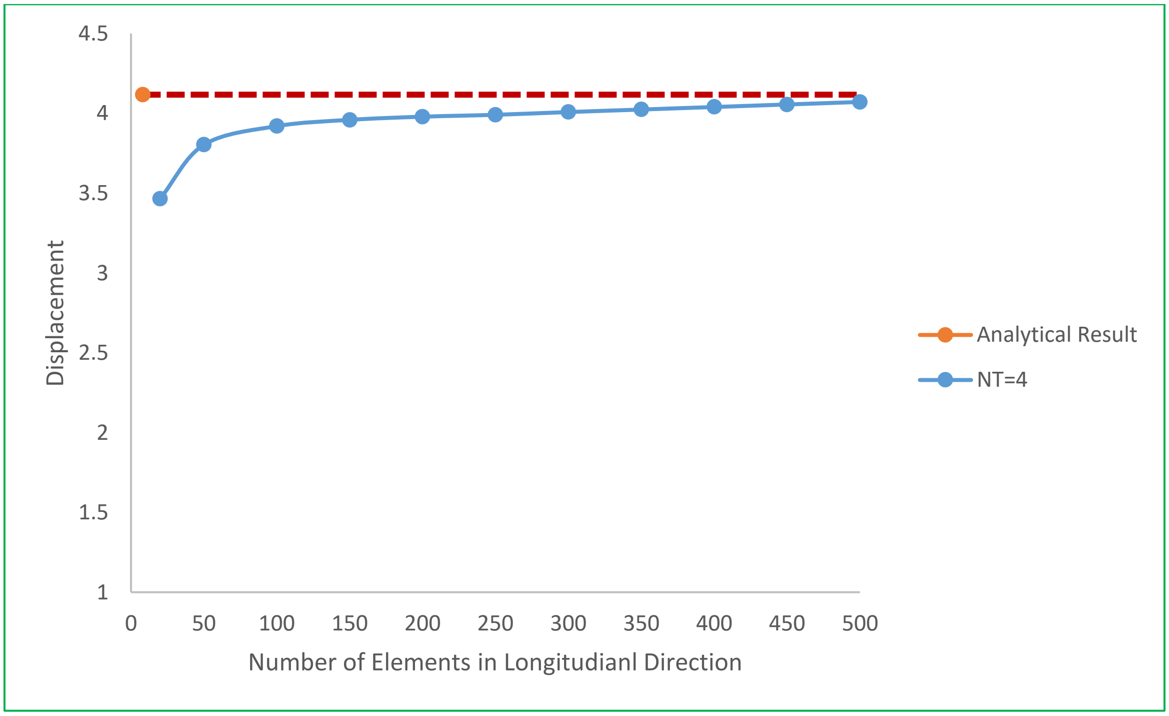


Figure 4.14 Displacement Vs Number of Elements in longitudinal direction for one-four elements along transverse direction

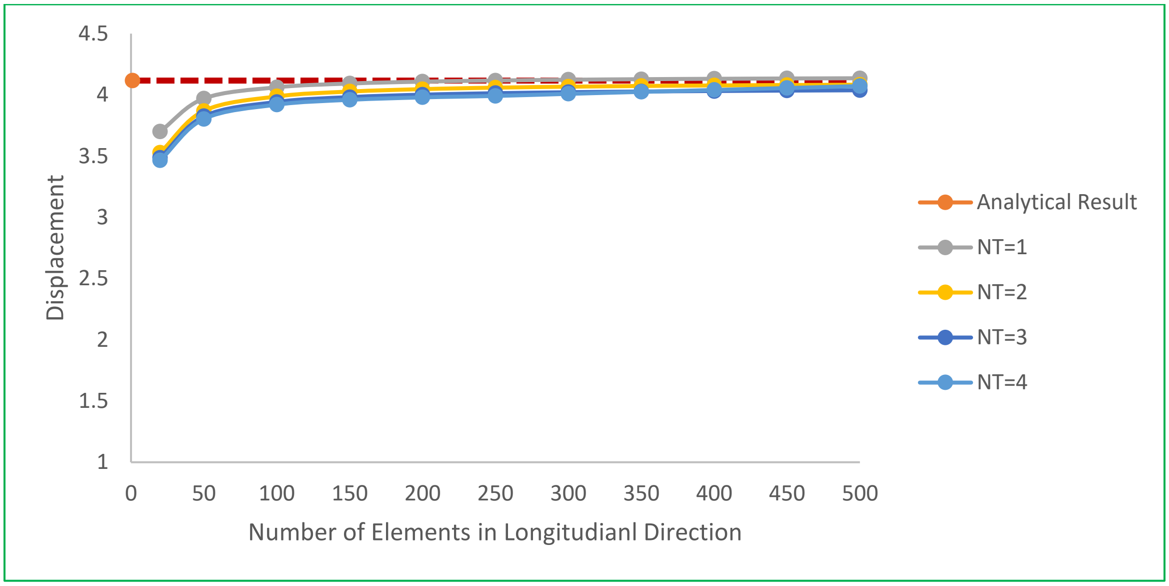


Figure 4.15 Combining Displacement Vs Number of Elements in longitudinal direction for different elements along transverse direction

To study the effects of two-dimensional discretization is conducted here, number of elements in longitudinal direction are taken from 50-500, to save the processing time elements along transversal direction taken 1-4 and numerical results are summarized in figure 4.12. From the result obtained it can be seen that, the more we discretize along the transverse direction it takes more numbers of elements to converge to analytical result. The overall agreement between the analytical result and numerical result is good. For the cantilever beam only discretizing along the length direction gives better result than discretizing along both directions. During the analysis while varying number of elements the computational time vary specially for discretization along the transverse direction, this indicate that number of elements and discretization type highly affect computational time.

4.2.3 Comparisons of AEM and FEM

This section presents the similarities and differences of the two numerical methods AEM and FEM, investigated on theoretically and numerically.

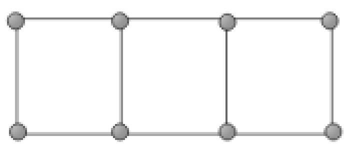
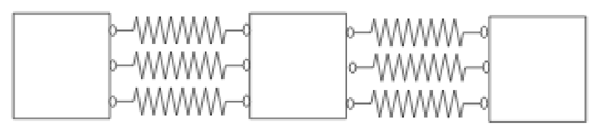
More than 75 years of development, Finite element method is the well-known and widely used numerical method, which is applicable to many fields including engineering and non-engineering to solve differential and integral equations, this paper only considers the use of FEM for structural analysis.

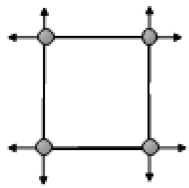
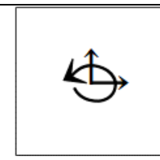
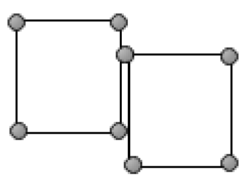
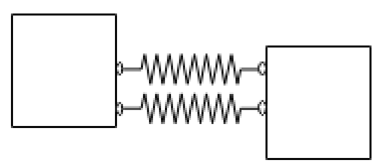
AEM is developed by professors Hatem TAGEL-DIN and Kimiro MEGURO. It combines features from finite element and discrete element methods. The primary benefit of this technique is that it can follow the structural collapse behavior going through all phases of loading.

AEM and FEM are both numerical methods where FEM is a general numerical method applied to numerous fields including engineering to solve differential equations, though AEM is simply restricted to structural analysis with a very simple modelling. Both methods have similar procedures after global stiffness matrix is developed. A comparison of AEM and FEM on their respective features is summarized on table below:

Table 4-2 Comparison of FEM and AEM on their respective features

Finite Element Method	**Applied Element Method**
It is a broadly utilized general numerical technique grown over 75 years, used to solve differential and integral equations and it is used in numerous fields of study.	A numerical technique that is developed not over 25 years, which is grown uniquely for structural analysis.

Involves modeling the structure using small interconnected elements called finite elements. These elements are continuum and deformable mediums and adopt all material and these elements are connected by nodes, DOF are assumed at the nodes, three quadrilateral elements shown by nodes is shown in figure below.  **Figure 4.16 Three quadrilateral elements connected in FEM**	The fundamental idea of the method is to divide the structure virtually with small elements. Elements can be in different shapes according to the geometry of the structure DOF's are assumed at the center of the element, these elements are strained by the springs (in normal and tangential direction) around their edge and makes the element rigid meaning size, shape of the element does not change under applied loading. Three elements connected via springs is shown below.  **Figure 4.17 Three elements connected with a pair of springs**
Requires some technical knowledge.	Understanding theory and simple modeling of AEM, doesn't need profoundly previous specialized technical knowledge about materials behavior.
Many types of finite element used for meshing, number and type of DOF depends on type of finite elements used for the analysis DOF's is located at the nodes position a quadrilateral element shown in Fig. below with 8 DOF's. 2 DOF's at each node.	DOF's are located at the center of each element

 **Figure 4.18 quadrilateral element with 8 DOF's**	 **Figure 4.19 AEM element with DOF's at the center**
Elements are connected by nodes; these nodes are not permitted to separate during analysis because in FEM partial connection is not allowed. If the elements move during apply loading connection will be lost between elements as shown in figure below.  **Figure 4.20 No connectivity when slip occurs in FEM**	Unlike the FEM, in AEM, the elements are connected through a series of springs on their surfaces These springs represent stresses, strains and connectivity between elements. Therefore, partial connectivity between the elements is permitted during the analysis and some of the springs can fail while others are effective and their failure will not cause any type of singularity in the model.  **Figure 4.21 Connectivity is still achieved even if ither is a slip**

In both methods degree of freedom has a lot of impact on the processing time, FEM has much degree of freedom than AEM. For instance, an 8-noded hexahedral element has 24 degrees of freedom three degrees of freedom at each node. DOF is greater than that of AEM because of this it takes more computational time.	3D elements in AEM have 6 degrees of freedom as shown in figure 2.1 (b). Less DOF less time taken during computation
Global stiffness matrix is computed by assembling individual stiffness matrix which are computed by considering all contribution of elements in the system.	Stiffness matrix is computed by considering all contribution of pair of springs.
The stiffness matrix is created by permitting a unit deformation in all directions of DOF's. The stiffness matrix is determined by integration in FEM, while, it is straightforwardly accessible in AEM. In FEM, numerical integration is essential when non-rectangular components are utilized. The straightforwardness of AEM is unequivocal from the fact that only springs are utilized for modeling any sort of structure.	
In some cases, size of elements are changed in structure from coarsely to finely discretized model a transition zone is needed in FEM to transmit from small elements to large elements.	Without the need of transition elements, small elements can be directly connected to larger elements. These reduces the number of elements at the same time reduce computation time. These makes this method uniquely suitable for collapse simulation.

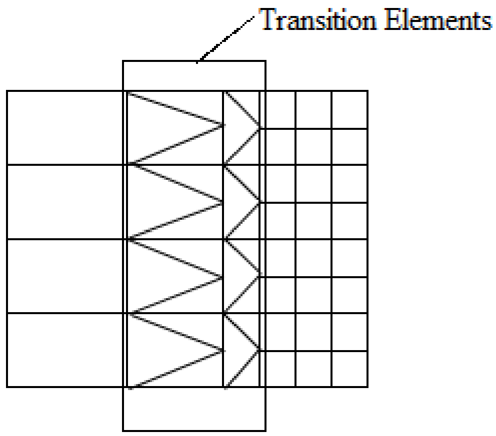
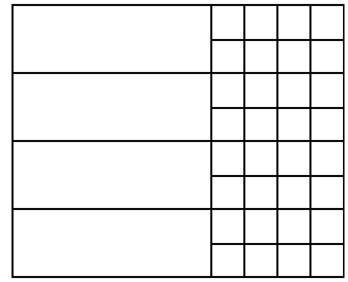
<table>
<tr>
<td>

Figure 4.22 Transition elements used in FEM</td>
<td>
Figure 4.23 Transition elements not required in AEM</td>
</tr>
<tr>
<td>All deformation is happening inside the elements.</td>
<td>Deformation and forces in element are represented by deformation and the forces in springs connecting the elements.</td>
</tr>
<tr>
<td>Progressive collapse analysis where the element separation is expected, element displacements must be treated independently to accurately track the behavior of phenomenon which the FEM is incapable of smoothly dealing with. each element when separation occurs.</td>
<td>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.</td>
</tr>
<tr>
<td>FEM is the only method able to explicitly consider shock waves caused by blast by performing a Multiphysics analysis.</td>
<td>AEM is currently not able to perform Multiphysics analysis (e.g., heat flux, radiation) and subsequent reaction of the structure, which is an area of future research</td>
</tr>
</table>

III. Convergence Test

Comparisons and validity of the two selected methodologies AEM and FEM is done in this section. Six random cantilever beams are selected as shown in Fig. 4.21 all beams have similar mechanical properties. The tip displacement of a cantilever beam is computed using different number of elements. AEM analysis done using numerical software “Scilab” and FEM analysis done using a finite element software “ANSYS 2020 R2” and numerical results of the tip displacement of a cantilever beam values compared with the analytical results.

Mechanical property of all the beams is similar, Modulus of elasticity and Poisson’s ratio of the material are 25000 N/mm^2 and 0.2 respectively. 10 number of springs are provided along all faces in the case of AEM.

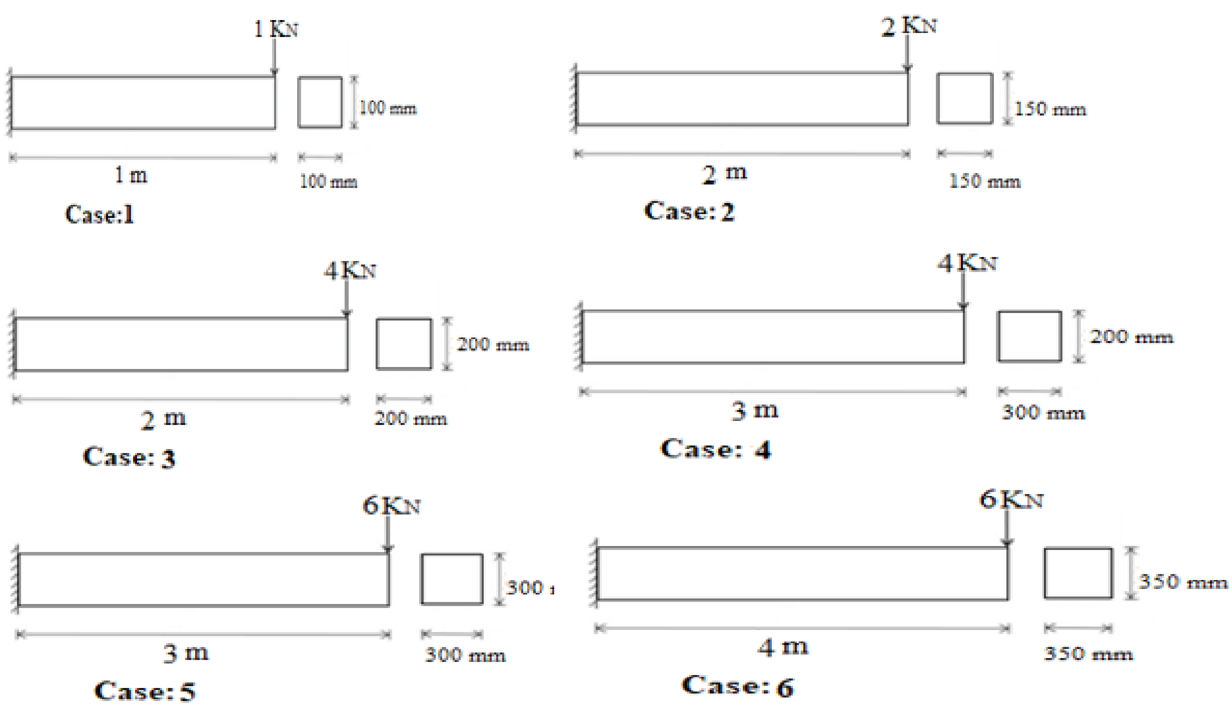
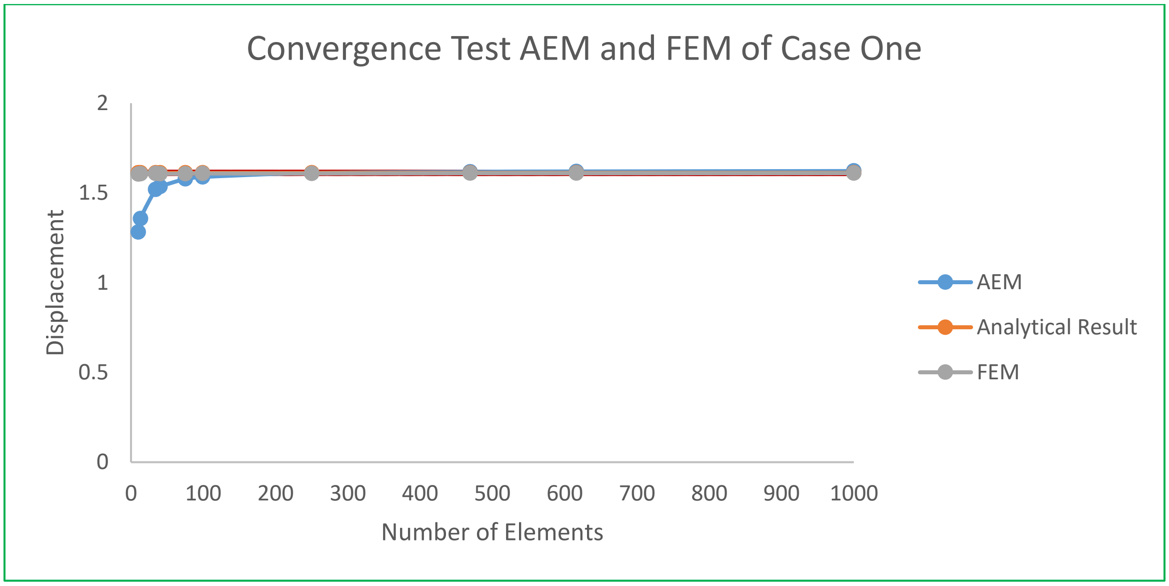
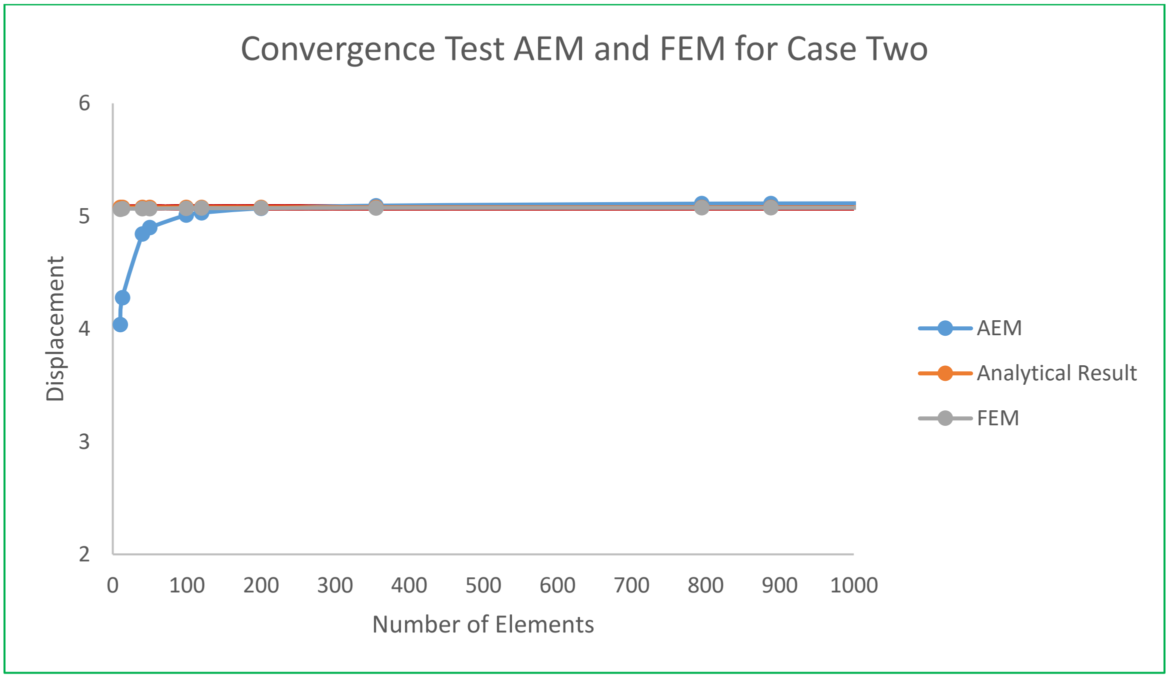


Figure 4.24 Random Cantilever Beam Samples

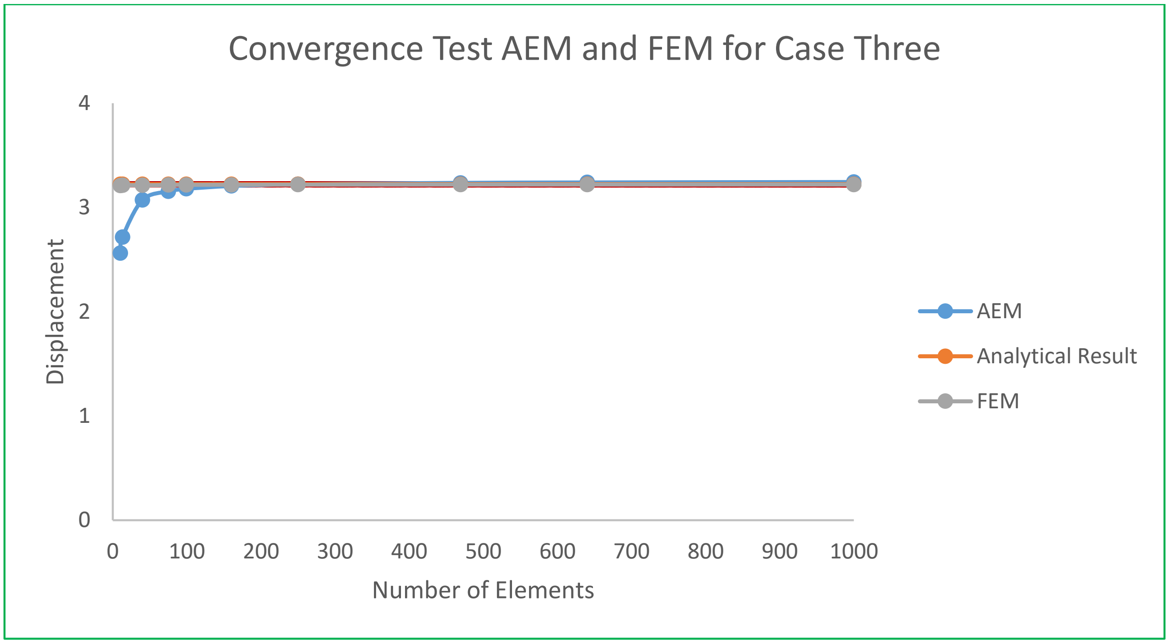
Case:1


Convergence Test AEM and FEM of Case One
Displacement
2
1.5
1
0.5
0
0
100
200
300
400
500
600
700
800
900
1000
Number of Elements
AEM
Analytical Result
FEM

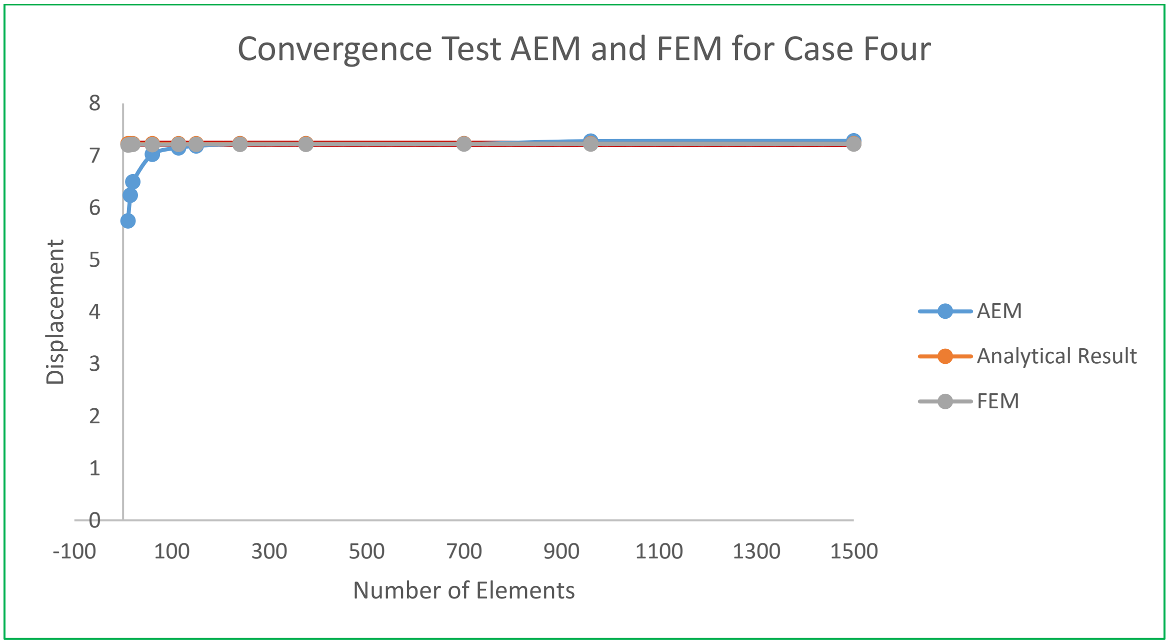
Case:2


Convergence Test AEM and FEM for Case Two
Displacement
6
5
4
3
2
0
100
200
300
400
500
600
700
800
900
1000
Number of Elements
AEM
Analytical Result
FEM

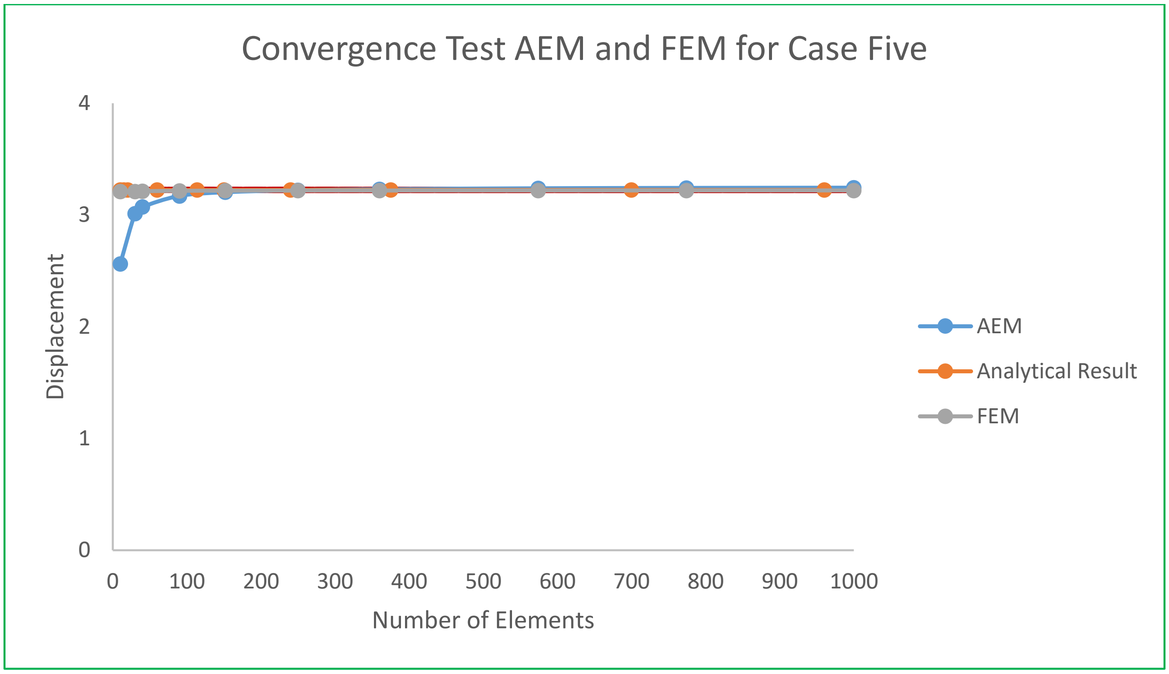
Case:3


Convergence Test AEM and FEM for Case Three
Displacement
0
1
2
3
4
Number of Elements
0
100
200
300
400
500
600
700
800
900
1000
AEM
Analytical Result
FEM

Case:4


Convergence Test AEM and FEM for Case Four
Displacement
0
1
2
3
4
5
6
7
8
Number of Elements
-100
100
300
500
700
900
1100
1300
1500
AEM
Analytical Result
FEM

Case:5



Case:6

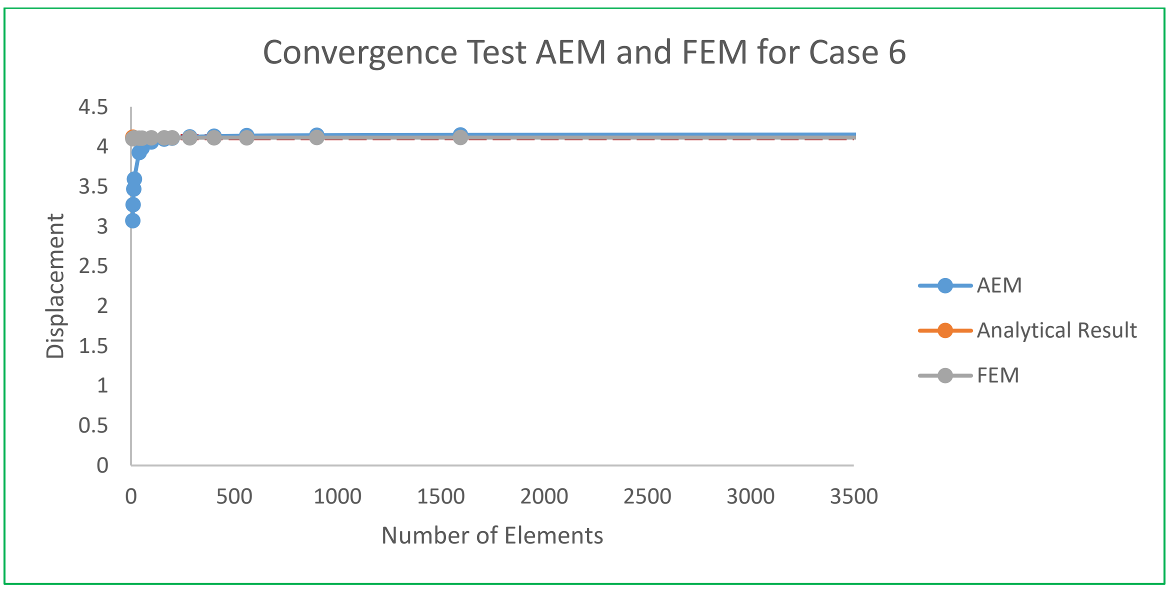


Figure 4.25 Displacement Vs Number of Elements for AEM and FEM numerical results for the six cases

From Figure 4.25, it is interesting to note that good agreement was observed on all randomly selected beams when comparing the analytical results with that of AEM and FEM numerical results for the same number of elements.

CHAPTER FIVE

5.0 Conclusions and Recommendations

5.1 Conclusions

FEM and AEM are numerical methods, other than age distinction and applications i.e FEM is developed for more than 75 years and it has wide range of application to many fields of studies, AEM developed for not more than 25 years is simply restricted to structural analysis.

In collapse modeling considering separation of structural elements is very important, it is challenging in FEM because it is based of continuous idealization of elements. AEM combines advantages of continuum and discrete element methods, which makes it a suitable method for collapse analysis with its simplicity in modeling do not need an earlier specialized technical knowledge about material behavior. In contrast to the FEM, deformations are not calculated within the elements but in connecting springs. Elements which are assumed rigid DOF's are found at each center of element and connected by a pair of springs, by the total contribution of springs stiffness matrix is developed. by contribution of all element's stiffness matrix is developed. After development of stiffness matrix in each method, global stiffness is found by assembling stiffness matrices and the procedure remains same after this. AEM is currently not able to perform Multiphysics analysis and subsequent reaction of the structure, which is an area of future research, but It has been proved that AEM is able to achieve satisfying results at blast loading and collapse analysis using the only AEM software package ELS.

The aim of this research is to lead to a better understanding of AEM, it involves a literature review, Overview of the AEM, validation of the AEM for analyzing cantilever beam, an investigation of the effects of key parameters including number of elements, number of springs connecting adjacent element, one-dimensional and two-dimensional discretization.

The findings from these investigations can lead to a better understanding of the effect of these various key parameters and their significant impact on the analysis of structures using AEM. The conclusions here are subjected to the assumptions and limitation the Applied Element Method.

From the background study of the AEM and from the validation study, the following conclusions were reached:

1. AEM is a very simple and efficient numerical method used for structural analysis.
2. It was found that using 10 contact springs or more contact springs to connect the two adjacent faces of the elements does not affect the accuracy of the results.
3. Good agreement was observed when comparing the analytical result with the numerical results of AEM and FEM.
4. For analysis of a beam, on determining displacement discretizing in one-dimension gives better results than discretizing in two-dimension. But for determining shear stress distribution elements along depth direction should be increased in addition to longitudinal direction.
5. In AEM, increasing the number of elements leads to increasing the computation time required to assemble global stiffness matrix and time required to determine the displacement because number of elements are associated with DOF.

6. Increasing the number of springs does not affect the computational time because number of springs does not associate with DOF's, can utilize large number of springs without altering the computational time.

5.2 Recommendations

This study mainly focused on 2D structural analysis using numerical methods, Emphasis is given to the recently new discrete element method Applied Element Method. In particular, analysis of a cantilever beam was done and parametric studies was investigated to see the effects of each key factors including Number of elements, Number of springs and Convergence test with the well-known FEM were examined in the study.

Recommendation for future works:

For future studies, a number of recommendations can be made as follows:

- This study can be extended by considering three-dimensional structural analysis and comparing results with analytical and FEM results and static and dynamic structural analysis by considering different types of structural elements.
- Further investigation using AEM, because it can predict strains, stress, bending moment, shear force, natural frequency and mode shapes with reasonable accuracy and relatively short computation time.
- "Extreme Loading on Structures" The only AEM software package if it is accessible many features of AEM can be studied.

References

A.S. Chepurnenko, A. S. V. C., 2018. Applied Element Method In The Solution of Plane Problems In The Theory of Creep. *Materials Physics and Mechanics,* p. 6.

AlHafian, S., 2013. Seismic Progressive Collapse of Reinforced Concrete Frame Structures Using The Applied Element Method. p. 310.

Anon., n.d. *Applied Element Method.* [Online]
Available at: https://www.appliedelementmethod.org/

Bargi, A. S. a. K., 2015. Use of Applied Element Method for Structural Analysis. *KSCE Journal of Civil Engineering,* p. 10.

Christoph Grunwalda, A. A. K. B. S. E. M. R., 2018. Reliability of Collapse Simulation – Comparing Finite and Applied Element Methods. *Elsevier*, p. 14.

Christy D, L. & Nagarajan, D. P. P. M., 2019. Simplified Analysis of Structures Using Applied Element Method. *Songklanakarin Journal of Science and Technology,* p. 33.

Clough, R. W., 1952. Original Formulation of The Finite Element Method. *Elsevier Science,* p. 13.

Clough, S. b. P. R. W., 2004. Early History of The Finite Element Method from The View Point of a Pioneer. *International Journal for Numerical Methods in Engineering,* p. 5.

D. Lincy Christy, T. M. M. P. a. P. N., 2020. A Comparison of Applied Element Method and Finite Element Method for Elastostatic Problems. *Structural Integrity Assessment*, p. 12.

D.Cook, R., 1994. *Finite Element Modeling for Stress Analysis.* Madison, s.n., p. 330.

D.Logan, D., 2012. *First Course in Finite Element Method.* Stamford: Global Engineering.

Daniele Malomo, P. C. R. P. A. p., 2018. *The Applied Element Method and The Modelling of Both In-Plane Resposnse of URM Walls.* s.l., s.n.

Erhunmwun, I., 2017. Review on Finite Element Method. *JASEM ISSN*, p. 5.

Faraji, J. J. C. ·. S., 2013. *Fundamentals of Structural Engineering.* New York: Springer.

Goodier, S. a. J., 1951. *Theory of Elasticity.* New York, Toronto, London: McGRAW-HILL BookCompany INC..

H. Daganic, D. V. a. S. L., 2018. An Overview of Methods for Blast Load Testing and Devices for Pressure Measurnment. *Hindawi Advanes in Civil Engineering*, p. 20.

Hakuno, K. M. a. M., 1989. Fracture Analysis of Concrete Structures By The Modified Distinct Element Method. *JSCE No. 410/I-12,* Volume 6, p. 12.

Hakuno, K. M. a. M., 1994. Application of The Extended Distinct Element Method for Collapse Simulation of A Double Deck Bridge. *J. Struct. Mech. Earthquake Eng. No. 483/I-26,* Volume 10, p. 11.

Hrennikoff, A. P., 1940. Plane Stress and Bending of Plates by Method of Articulated Framework. p. 201.

Jainu Karthik, D. v. a. A. S. K., 2019. Analysis and Impact of Blast Load on Structural Elements. *International Journal of Innovative Technology and Exploring Engineering,* Volume 8, p. 6.

Kassimali, A., 2011. *Structural Analysis.* USA: Cengage.

Kawin Worakanchana, P. M. R. G. N. a. K. M., 2008. 3-D Applied Element Method for Static Non-Linear Simulation of PP-Band Retrofitted Masonry. *Bulletin of ERS, No. 41,* p. 11.

Lincy Christy D., T. M. M. P. P. N., 2018. *Analysis of Concrete Beams Using Applied Element Method.* NIT Calicut, India, IOP, p. 8.

M.J.Nollet, A. K. a., 2008. *Application of the Applied Element Method to The Seismic Vulnerability Evaluation of Existing Buildings.* Quebec, QC, s.n.

Marin Lupoae, C. B., 2009. *Use of Applied Element Method to Simulate The Collapse of A Building.* Bucharest, s.n.

Meguro, H. T.-D. a. K., 2000. Nonlinear Simulation of RC Structures Using Applied Element Method. *Structural Eng./Earthquake Eng., JSCE,* Volume 17, p. 12.

Meguro, H. T.-D. A. K., 2015. Analysis of Small Scale RC Building Subjected to Shaking Table Tests Using Applied Element Method. *12WCEE 200,* p. 9.

Meguro, K., 2001. *Applied Element Method: A New Efficient Tool for Design of Structure Considering its Failure Behaviour.* Bangkok, s.n.

Mohammed El-desoqi, M. E. H. S., 2020. Progressive Collapse Assessment of Precast Reinforced Concrete Beams Using Applied Element Method. *Case Studies in Construction Materials,* p. 24.

Nollet, A. K. a. M., 2008. Application of the Applied Element Method to The Seismic Vulnerability Evaluation of Existing Buildings. *CSCE 2008 Annual Conference,* p. 11.

Osama El-Mahdy, E.-S. E.-K. H. A. A. E.-G., 2017. Aplication of AEM in Progressive Collapse Dynamics Analysis of R.C. Structures. *Civil Engineering,* p. 19.

R., C., 1942. *Variational Methods for The Solution of Problems of Equilibrium and Vibrations.* Washington, D.C, s.n., p. 23.

Sushma Pulikanti, M. A. H. a. R. P. K., 2012. Amplication Studies of Local Soils Using Applied Element Method. *International Journal of Earth Sciences and Engineering,* p. 12.

Szab, B. A., 1979. Some Recent Developments in Finite Element Analysis. *Camp. & Mnlhs. wifh Apple,* p. 17.

Tagel-Din, B. K. M. a. H., 2001. Applied Element Simulation of RC Structures Under Cyclic Loading. *Journal of Structural Engineering,* p. 11.

Tagel-Din, H. &. M., 2002. Applied Element Method Used for Large Displacement Structural Analysis. *Structural Eng./Earthquake, JSCE,* p. 10.

Tagel-Din, K. M. a. H., 2000. Applied Element Method for Structural analysis: Theory and application for linear materials. *Structural Eng. / Earthquake Eng., JSCE,* Volume 17, p. 16.

Tagel-Din, K. M. a. H., 2000. Applied Element Method for Structural Analysis: Theory and Application for Linear Materials. *Structural Eng./Earthquake Eng., JSCE,* Volume 17, p. 16.

Tagel-Din, K. M. a. H., n.d. *A New Simplified and Efficient Technique for Fracture Behaviour of Concrete Structures.* Germany, s.n., p. 10.

Tahmilci, F., 2007. Analysis of Blast Loading Effect on Regular Steel Building Structures. p. 147.

Vikas Gohel, P. V. a. D. J., 2012. *Analysis of Frame Using Applied Element Method (AEM).* India, SciVerse ScienceDirect, p. 9.

Wiley, 2018. *Fundamentals of Finite Element Analysis.* Croydon: CPI Group (UK) Ltd.

APPENDIX-A

In this paper codes were developed using a numerical software called Scilab, these programs are used for statical structural analysis for a cantilever beam. They can be used as a basis for future studies to develop more codes using AEM. The verification of the codes on its accuracy was evaluated by comparing its numerical results with the analytical and FEM results.

Sample Scilab code for 1D-Discritization along length direction with 10 springs connecting each face of an element.

```
clear

//Discretization Only along the length direction
// Cantilever Beam Analysis Using AEM

//Define Units Used

// Force: N
// Length: mm
// Moment: N.mm

// Define number of elements
   n=450
   ny=10
   ns = n-1
   //Where:
// n: number of elements
//ns: number of group of springs
//ny: number of sprigs in each element face

//Given (mm) :-

//Geometric Properties: -
t=350;
D=350;
L=4000;
d=D/ny;
a=L/n;
x=a/2;

// Where:
// t: Thickness
// a: C/C distance between elements
```

```
// L: Length of beam
// D: Depth of beam
// d: Distance between springs
// x: Point of contact with respect to centroid along the X-axis

// Material Properties: -

v=0.2;
E=25000;

//Shear Modulus G (N/mm^2)

G= E/(2*(1+v));

//Normal and Shearing Stiffness Constants: -

//Normal Stiffness (N)

Kn= (E*d*t) / a;

//Shearing Stiffness (N)

Ks= (G*d*t) / a;

// Where: -
// G: Shear Modulus+
// v= Poisson ratio
// E= Young's modulus

function K=springstiffness(Kn, Ks, x, y)
  K(1,1) = Kn;
  K(2,1) = (0);
  K(3,1) = -Kn*y;
  K(4,1) = -Kn;
  K(5,1) = (0);
  K(6,1) = Kn*y;
  K(2,2) = Ks;
  K(3,2) = Ks*x;
  K(4,2) = (0);
  K(5,2) = -Ks;
  K(6,2) = Ks*x;
  K(3,3) = (Kn*y^2)+(Ks*x^2);
  K(4,3) = Kn*y;
  K(5,3) = -Ks*x;
  K(6,3) = (-Kn*y^2)+(Ks*x^2);
```

```
   K(4,4) = Kn;
   K(5,4) = (0);
   K(6,4) = -Kn*y;
   K(5,5) = Ks;
   K(6,5) = -Ks*x;
   K(6,6) = (Kn*y^2)+(Ks*x^2);
for i = 1:6
   for j = 1:6
      K(i,j)= K(j,i)
    end
end
endfunction

//..............TEN SPRINGS IN EACH FACE...........//
//Spring 1:

y1=(D/2)-(d/2);

y=y1;

K1 = springstiffness(Kn,Ks,x,y);

//Spring 2:

y2=y1-d;

y=y2;

K2 = springstiffness(Kn,Ks,x,y);

//Spring 3:

y3=y2-d;

y=y3;

K3 = springstiffness(Kn,Ks,x,y);

//Spring 4:

y4=y3-d;

y=y4;

K4 = springstiffness(Kn,Ks,x,y);
```

```
//Spring 5:
y5=y4-d;

y=y5;

K5 = springstiffness(Kn,Ks,x,y);
//............................
//Spring 6:

y6=y5-d;

y=y6;

K6 = springstiffness(Kn,Ks,x,y);

//Spring 7:

y7=y6-d;

y=y7;

K7 = springstiffness(Kn,Ks,x,y);

//Spring 8:

y8=y7-d;

y=y8;

K8 = springstiffness(Kn,Ks,x,y);

//Spring 9:
y9=y8-d;

y=y9;

K9 = springstiffness(Kn,Ks,x,y);

//Spring 10:
y10=y9-d;

y=y10;

K10 = springstiffness(Kn,Ks,x,y);
```

```
//............................
//Assembling Stiffness On Each Face
// Define element stiffness matrix
k = zeros(6,6);
k(1:6,1:6)=
K1(1:6,1:6)+K2(1:6,1:6)+K3(1:6,1:6)+K4(1:6,1:6)+K5(1:6,1:6)+K6(1:6,1:6)+K7(1:6,1:6)+K
8(1:6,1:6)+K9(1:6,1:6)+K10(1:6,1:6);

//............................
// Hard coded size of global stiffness matrix
K = zeros(3*ns+3,3*ns+3);
size(K)

// Hard coded loop for assembling each element stiffness matrix
for i = 1:ns
   j = 3*i-2;
   m = 3*i+3;
   K(j:m,j:m) = K(j:m,j:m) +k;
end

// Display the global matrix

K;
size(K)

//Applying Boundary Conditions
//DOF 1,2,3 are removed because of fixed support

K(3,:) = [];
K(:,3) = [];

K(2,:) = [];
K(:,2) = [];

K(1,:) = [];
K(:,1) = [];

size(K)

//Load Vector

F=6000;

f=[zeros((n*3-5),1);-F; -(0.5*F*a)];
```

```
size(f)
//Displacement

Dtip = inv(K)*f;
```

APPENDIX-B

Sample Scilab code for 2D-Discritization with 10 springs connecting each face of an element.

```
clear

// Cantilever Beam Analysis Using AEM
//Define Units Used

// Force: N
// Length: mm
// Moment: N.mm

//Given (mm) :-
//Geometric Properties: -
t=350;
L=4000;
D=350;

// Where:
// t: Thickness
// a: C/C distance between elements
// L: Length of beam
// D: Depth of beam
// d: Distance between elements
// x: Point of contact with respect to centroid along the X-axis

// Material Properties: -

v=0.2
E=25000

//Shear Modulus, G (N/mm^2)

G= E/(2*(1+v))

//Discretization:

// Define number of elements per one row in length direction
NL = 500
// Define number of elements per one column in depth direction
ND = 2

//Stiffness Along Length Direction:
```

```
a=L/NL
x=a/2
DL=D/ND
d=DL/10
//Normal and Shearing Stiffness Constants: -

//Normal Stiffness (N)

Kn= (E*d*t) / a

//Shearing Stiffness (N)

Ks= (G*d*t) / a

// Where: -
// G: Shear Modulus
// v= Poisson ratio
// E= Young's modulus

function K=springstiffness(Kn, Ks, x, y)
   K(1,1) = Kn;
   K(2,1) = (0);
   K(3,1) = -(Kn*y);
   K(4,1) = -Kn;
   K(5,1) = (0);
   K(6,1) = (Kn*y);
   K(2,2) = Ks;
   K(3,2) = (Ks*x);
   K(4,2) = (0);
   K(5,2) = -Ks;
   K(6,2) = (Ks*x);
   K(3,3) = ((Kn*y^2)+(Ks*x^2));
   K(4,3) = (Kn*y);
   K(5,3) = -(Ks*x);
   K(6,3) = ((-Kn*y^2)+(Ks*x^2));
   K(4,4) = Kn;
   K(5,4) = (0);
   K(6,4) = -(Kn*y);
   K(5,5) = Ks;
   K(6,5) = -(Ks*x);
   K(6,6) = ((Kn*y^2)+(Ks*x^2))
for i = 1:6
   for j = 1:6
      K(i,j)= K(j,i)
    end
```

```
end
endfunction

//..............TEN SPRINGS IN EACH FACE...........//
//Spring 1:

y1=(DL/2)-(d/2);

y=y1;

K1 = springstiffness(Kn,Ks,x,y)

//Spring 2:

y2=y1-d;

y=y2;

K2 = springstiffness(Kn,Ks,x,y)

//Spring 3:

y3=y2-d;

y=y3;

K3 = springstiffness(Kn,Ks,x,y)

//Spring 4:

y4=y3-d;

y=y4;

K4 = springstiffness(Kn,Ks,x,y)

//Spring 5:
y5=y4-d;

y=y5;

K5 = springstiffness(Kn,Ks,x,y)
//............................
//Spring 6:
```

```
y6=y5-d;

y=y6;

K6 = springstiffness(Kn,Ks,x,y)

//Spring 7:

y7=y6-d;

y=y7;

K7 = springstiffness(Kn,Ks,x,y)

//Spring 8:

y8=y7-d;

y=y8;

K8 = springstiffness(Kn,Ks,x,y)

//Spring 9:
y9=y8-d;

y=y9;

K9 = springstiffness(Kn,Ks,x,y)

//Spring 10:
y10=y9-d;

y=y10;

K10 = springstiffness(Kn,Ks,x,y)

//..............................

//Assembling Stiffness On Each Face
// Define element stiffness matrix
k= zeros(6,6);
k(1:6,1:6)=
K1(1:6,1:6)+K2(1:6,1:6)+K3(1:6,1:6)+K4(1:6,1:6)+K5(1:6,1:6)+K6(1:6,1:6)+K7(1:6,1:6)+K
8(1:6,1:6)+K9(1:6,1:6)+K10(1:6,1:6)
```

```
//Stiffness Along Depth Direction:

a=D/ND
Dd=L/NL
d=Dd/10
x=a/2

//Normal and Shearing Stiffness Constants: -

//Normal Stiffness (N)

Kn= (E*d*t) / a

//Shearing Stiffness (N)

Ks= (G*d*t) / a

// Where: -
// G: Shear Modulus
// v= Poisson ratio
// E= Young's modulus



function K=springstiffness(Kn, Ks, x, y)
   K(1,1) = Kn;
   K(2,1) = (0);
   K(3,1) = -(Kn*y);
   K(4,1) = -Kn;
   K(5,1) = (0);
   K(6,1) = (Kn*y);
   K(2,2) = Ks;
   K(3,2) = (Ks*x);
   K(4,2) = (0);
   K(5,2) = -Ks;
   K(6,2) = (Ks*x);
   K(3,3) = ((Kn*y^2)+(Ks*x^2));
   K(4,3) = (Kn*y);
   K(5,3) = -(Ks*x);
   K(6,3) = ((-Kn*y^2)+(Ks*x^2));
   K(4,4) = Kn;
   K(5,4) = (0);
   K(6,4) = -(Kn*y);
   K(5,5) = Ks;
```

```
   K(6,5) = -(Ks*x);
   K(6,6) = ((Kn*y^2)+(Ks*x^2))
for i = 1:6
   for j = 1:6
      K(i,j)= K(j,i)
    end
end
endfunction

//.............TEN SPRINGS IN EACH FACE...........//
//Spring 1:

y1=(Dd/2)-(d/2);

y=y1;

K1 = springstiffness(Kn,Ks,x,y)

//Spring 2:

y2=y1-d;

y=y2;

K2 = springstiffness(Kn,Ks,x,y)

//Spring 3:

y3=y2-d;

y=y3;

K3 = springstiffness(Kn,Ks,x,y)

//Spring 4:

y4=y3-d;

y=y4;

K4 = springstiffness(Kn,Ks,x,y)

//Spring 5:
y5=y4-d;
```

```
y=y5;

K5 = springstiffness(Kn,Ks,x,y)
//...........................
//Spring 6:

y6=y5-d;

y=y6;

K6 = springstiffness(Kn,Ks,x,y)

//Spring 7:

y7=y6-d;

y=y7;

K7 = springstiffness(Kn,Ks,x,y)

//Spring 8:

y8=y7-d;

y=y8;

K8 = springstiffness(Kn,Ks,x,y)

//Spring 9:
y9=y8-d;

y=y9;

K9 = springstiffness(Kn,Ks,x,y)

//Spring 10:
y10=y9-d;

y=y10;

K10 = springstiffness(Kn,Ks,x,y)

//............................

//Assembling Stiffness On Each Face
```

```
// Define element stiffness matrix
kd= zeros(6,6);
kd(1:6,1:6)=
K1(1:6,1:6)+K2(1:6,1:6)+K3(1:6,1:6)+K4(1:6,1:6)+K5(1:6,1:6)+K6(1:6,1:6)+K7(1:6,1:6)+K
8(1:6,1:6)+K9(1:6,1:6)+K10(1:6,1:6)

//Stiffness along transverse direction
//Transformation Matrix
A=90;
 l=cosd(A)
 m=sind(A)

T=[l m 0 0 0 0;
  -m l 0 0 0 0;
   0 0 1 0 0 0;
   0 0 0 l m 0;
   0 0 0 -m l 0;
   0 0 0 0 0 1]
k1=T'*kd*T

// Hard coded number of springs per one row
n = NL-1
// Hard coded total number of vertical springs
tnvs = (ND-1)*NL

// Hard coded size of global stiffness matrix
K = zeros(3*n+3,3*n+3);
size(K)
// Define element horizontal stiffness matrix (Kh)
Kh = k

// Define element vertical stiffness matrix (Kv)
Kv = k1

// Hard coded loop for assembling each element stiffness matrix
for i = 1:n
   j = 3*i-2;
   m = 3*i+3;
   K(j:m,j:m) = K(j:m,j:m) +Kh;
end
Kj = K

for i = 1:ND-1
   K = blockdiag(K,Kj)
end
```

```
for i = 1:tnvs
   x = 3*i-2;
   y = 3*i;
   v = 3*i-2;
   w = 3*i;
   s = 3*i+3*(NL-1)+1;
   t = 3*i+3*(NL-1)+3;
   K(x:y,x:y) = K(x:y,x:y) +Kv(1:3,1:3);
   K(v:w,s:t) = K(v:w,s:t) +Kv(1:3,4:6);
   K(s:t,v:w) = K(s:t,v:w) +Kv(4:6,1:3);
   K(s:t,s:t) = K(s:t,s:t) +Kv(4:6,4:6);
end
// Display the global matrix

K;
//Applying Boundary Conditions
//DOF's are removed because of fixed support

K((NL*3)+3,:) = [];
K(:,(NL*3)+3) = [];

K((NL*3)+2,:) = [];
K(:,(NL*3)+2) = [];

K((NL*3)+1,:) = [];
K(:,(NL*3)+1) = [];

K(3,:) = [];
K(:,3) = [];

K(2,:) = [];
K(:,2) = [];

K(1,:) = [];
K(:,1) = [];

size(K)

//Load Vector

F=6000;

f=zeros((NL*ND)*3,1);
f(((NL*3)-2),1)=0;
```

```
f(((NL*3)-1),1)=-F;
f((NL*3),1)=-(0.5*F*a);

f((NL*3)+3,:) = [];
f((NL*3)+2,:) = [];
f((NL*3)+1,:) = [];
f(3,:) = [];
f(2,:) = [];
f(1,:) = [];

size(f)

//Displacement

Dtip = inv(K)*f
```

[This page intentionally left blank]