The Islamic University – Gaza
Postgraduate Deanship
Faulty of Engineering
Structural Engineering Department

الجامعة الإسلامية – غزة
عمادة الدراسات العليا
كلية الهندسة
قسم الإنشاءات

NEW STRAIN-BASED TRIANGULAR AND RECTANGULAR FINITE ELEMENTS FOR PLANE ELASTICITY PROBLEMS

SUBMITTED BY
ENG. SALAH M. TAYEH

SUPERVISED BY
DR. ATTIA I. MOUSA
Associate Professor, Dean of Faculty of Engineering
Islamic University – Gaza

A Thesis submitted in partial fulfillment of the requirements
for the Degree of Master of Science in Civil / Structural Engineering

JULY 2003

بسم الله الرحمن الرحيم

لَا يُكَلِّفُ اللَّهُ نَفْسًا إِلَّا وُسْعَهَا لَهَا مَا كَسَبَتْ وَعَلَيْهَا مَا اكْتَسَبَتْ رَبَّنَا لَا تُؤَاخِذْنَا
إِن نَّسِينَا أَوْ أَخْطَأْنَا رَبَّنَا وَلَا تَحْمِلْ عَلَيْنَا إِصْرًا كَمَا حَمَلْتَهُ عَلَى الَّذِينَ مِن قَبْلِنَا
رَبَّنَا وَلَا تُحَمِّلْنَا مَا لَا طَاقَةَ لَنَا بِهِ وَاعْفُ عَنَّا وَاغْفِرْ لَنَا وَارْحَمْنَا أَنتَ مَوْلَانَا
فَانصُرْنَا عَلَى الْقَوْمِ الْكَافِرِينَ

صدق الله العظيم

سورة البقرة الآية – ٢٨٦ -

ABSTRACT

In this thesis, the Finite Element Method of structural analysis is used to investigate and compare the performance of several strain-based elements. Two new strain-based triangular and rectangular finite elements in Cartesian coordinates system, for two dimensional elasticity problems are developed. Each of these elements has three degrees of freedom per node. The "Strain Based Approach" is used to develop and formulate these two dimensional finite elements. In this approach, finite elements are formulated based on assumed polynomial strains rather than displacements.

Two main computer programs are developed to analyze the new finite elements. To test the performance of these elements, they are used to solve two common plane elasticity problems. The problems considered included are: the problem of a plane deep cantilever beam fixed at one end and loaded by a point load at the free end; and the problem of a simply supported beam loaded at the mid-span by a point load.

The finite element solutions obtained for these problems are compared with the analytical values given by the elasticity solutions.

Results obtained using the new triangular and rectangular elements are also compared to those of the well-known constant strain triangular element (CST) and the bilinear rectangular element (BRE) respectively.

In all cases, convergence curves for deflection at specific points within each problem are plotted to show that acceptable levels of accuracy. Furthermore, convergence curves for bending stress at points on the upper surface and shear stress at points on the neutral axis are plotted; again convergence is ensured.

TO MY PARENTS

AND

MY WIFE

I DEDICATE THIS WORK

ACKNOWLEDGMENTS

The work presented in this thesis was carried out in the Department of Civil and Structural Engineering at the Islamic University of Gaza, under the supervision of Dr. Attia Mousa, Dean of Faculty of Civil Engineering.

The researcher wishes to take this opportunity to express his acknowledgments and gratitude to Dr. Mousa for his continued guidance and encouragement throughout the course of this work.

My thanks extend to the lecturers at the Civil Engineering Department for their effort and their role in launching the M. Sc. program in the field of Structural Engineering.

I am particularly grateful to my parents, whom, without their support I would have not been able to complete this work.

Finally, I present my thanks for all people who helped me in completing this work..

LIST OF ABBREVIATIONS (NOTATIONS)

Symbol	**Notation**
x, y	Cartesian coordinates
T	Thickness of the element
E	Young's Modulus of Elasticity
ν	Poisson's ratio
U	Displacement in the x direction
V	Displacement in the y direction
ϕ	In-plane rotation
ε_x, ε_y	Direct strains in the x and y directions, respectively
γ_{xy}	Shear strain
σ_x, σ_y	Normal stresses in the x and y directions, respectively
τ_{xy}	Shear stress
$[C]$	Displacement transformation matrix
$[B]$	Strain matrix
$[D]$	Rigidity matrix
$[K^e]$	Element stiffness matrix
$[Q]$	Strain energy matrix
$\{\delta^e\}$	Element nodal displacement vector
Π	Total Potential Energy
DoF	Degree of Freedom
CST	Constant Strain Triangle
BRE	Bilinear Rectangular Element
SBTREIR	Strain-Based Triangular Element with In-plane Rotation
SBREIR	Strain-Based Rectangular Element with In-plane Rotation

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

1. CHAPTER ONE: INTRODUCTION

1.1 INTRODUCTION

The conventional analytical approaches to solution of plane elasticity problems are based on determining the necessary equations governing the behavior of the structure by taking into account equi1ibrium and compatibi1ity within the structure as in the methods developed in the theory of elasticity [4], [32] & [35]. The solution to these equations exists only for special cases of loading and boundary conditions. Due to the complex nature of the shape of the structure, loading pattern, irregularities in geometry or material, the analytical solution normal1y becomes difficult and even impossible to solve such problems for displacements, stress or strains within the structure. The need for some other technique, such as suitable numerical methods for tackling the more complex structures with arbitrary shapes, loading and boundary conditions, is then essential.

Several approximate numerical methods have evolved over the years. One of the common methods is the Finite Difference scheme in which an approximation to the governing equations is used. The solution is formed by writing difference equations for a grid points. The solution is improved as more points are used. With this technique, some fairly difficult problems can be treated, but for example, for problems of irregular geometries or unusual specification of boundary conditions, the solution becomes more complex and difficult to obtain. On the other hand, the Finite Element Method (FEM), can take care of all these complex problems, and hence has become more widespread in finding solutions to complex structural and non-structural problems.

The Finite Element Method (FEM), or Finite Element Analysis (FEA), is based on the idea of building a complicated object with simple blocks, or, dividing a complicated object into small and manageable pieces. Application of this simple idea can be found everywhere in everyday life as well as in engineering. FEM is a powerful method for the analysis of continuous structures including complex geometrical configurations, material properties, or loading. These structures are idealized as consisting of one, two or three-dimensional elements connected at the nodal points, common edges, or surfaces. An important category is the "two dimensional plane elasticity problems". These problems are characterized by the following assumptions:

- two dimensions are large, the third is small,
- the structure is plane
- the loads act parallel to the plane.

These structures are described by the mid plane and the thickness distribution. Because of the special type of loading, the general three-dimensional behavior of a continuum can be reduced to two dimensions by the assumption of constant distributed stresses or strains throughout the thickness. The English term "plate" only reflects the geometry of the structure whereas the German term "Scheibe" additionally refers to the fact that only membrane action is present with no bending or twisting. The following terms might alternatively be used:

- in-plane loaded plate (plane stress)
- membrane structure
- plane stress / strain structure

The two-dimensional plate elements (that will be studied in this thesis) are extremely important for:

(1) Plane stress analysis which is defined as the state of stress in which the normal stress and the shear stresses directed perpendicular to the plane are assumed to be zero. This includes problems such as plates with holes, fillets or other changes in geometry that result in stress concentrations.

(2) Plane strain analysis which is defined as the state of strain in which the direct and shear strains normal to the x-y plane are assumed to be zero. This includes problems such as long underground box culverts subjected to uniform load acting constantly along its length, or a dam subjected to the hydrostatic horizontal loading along its length [15].

1.2 HISTORY OF FINITE ELEMENT METHOD

The modern development of the finite element method in the field of structural engineering dates back to 1941 and 1943, when its key features were published by Courant [4], Hrenikoff [10] and McHenry [15]. The work of Courant is particularly significant because of its concern with problems governed by equations applicable to structural mechanics and other situations. He proposed setting up the solution of stresses in the variational form. Then he introduced piecewise interpolation functions (shape functions) over triangular sub-regions making the whole region to obtain the approximate numerical solution.

In 1947, Levy [13] developed the flexibility method (force method) and in 1953 he suggested that the use of the displacement method could be a good alternative for the analyzing statically redundant aircraft wings [14]. This method became popular only later after the invention of the high-speed computers.

In 1954, Argyris and Kelsey [12] gave a very general formulation of the stiffness matrix method based on the fundamental energy principles of elasticity. This illustrated the importance of the energy principles and their role in the development of the finite element method.

In 1956, Turner, Clough, Martin and Topp [34] presented the first treatment of the two dimensional elements. They derived the stiffness matrices for triangular and rectangular elements based on assumed displacements and they outlined the procedure commonly known as the Direct Stiffness Method for assembling the total stiffness matrix of the structure. This is regarded as one of the key contributions in the discovery of the finite element method.

The technology of finite elements has advanced through a number of indistinct phases in the period since the mid 1950's. The formulation of the triangular and rectangular elements for plane stress has motivated the researchers to continue and establish element relationships for solids, plates in bending and thin shells.
In 1960's, linear strain triangular element was developed [33]. This element has 6 nodes with 2 degrees of freedom per node. The derivation of the stiffness matrix for this element was difficult. The isoparametric formulation was then developed [11] in which both the element geometry and displacements are defined by the same interpolation functions. This formulation was then applied to two and three dimensional stress analysis where higher order triangular and rectangular plane elements were developed. Also, brick elements were developed for three dimensional stress analysis. Elements created can be non rectangular and have curved sides.
By the early 1970's, this method was further developed for use in the aerospace and nuclear industries where the safety of the structures is critical. Since the rapid decline in the cost of computers, FEM has been developed to an incredible precision. Currently, there exist commercial finite element packages that are capable of solving the most sophisticated problems for static as well as dynamic loading, in a wide range of structural as well as non-structural applications.
Before reviewing the available finite element solutions for two dimensional structures, a brief introduction to the finite element method is presented showing a description of the procedure for obtaining the stiffness matrix of the general (triangular or rectangular) plane element.

1.3 PROCEDURE OF THE FINITE ELEMENT METHOD

In continuum problem of any dimension, the field variable (whether it is pressure, temperature, displacement, stress or some other quantity) possesses infinite values because it is a function of each generic point in the body or solution region. Consequent1y, the problem is one with an infinite number of unknowns. The finite element discretization procedure reduces the problem to one of finite numbers of unknowns by dividing the solution region into elements and by expressing the unknown field variable in terms of assumed approximating function within each element. The approximation functions are defined in terms of the values of the field variables at specified points called nodes or nodal points. Nodes usually lie on the element boundaries where adjacent elements are considered to be connected. The nodal values of the field variable and the approximation for the elements completely define the behavior of the field variables within the elements. For the finite element representation of a problem, the nodal values of the field variables become the new unknowns. Once these unknowns are found, the functions define the field variable throughout the assemblage of elements.

Clearly, the nature of the solution and the degree of approximation depend not only on the size and the number of the elements used, but also on the selected approximation functions.

An important feature of the finite element method that sets it apart from other approximate numerical methods is the ability to formulate solutions for individual elements before putting them together to represent the entire problem. Another advantage of the finite element method is the variety of ways in which one can formulate the properties of individual elements. The most common approach to obtaining element properties is called "displacement approach" and the method can be summarized as described below.

1.3.1 Idealization of the Structure

The idealization governs the type of the element that must be used in the solution of the structure. A variety of element shapes can be used, and with care, different element shapes may be employed in the same solution. Indeed when analyzing, for example, an elastic shell that has different types of components such as stiffener beams, it is necessary to use different types of elements in the same solution.

1.3.2 Formulation of Element Stiffness Matrix

The evaluation of the stiffness matrix for a finite element is the most critical step in the whole procedure because it controls the accuracy of the approximation. This step includes the choice of:

- The number of nodes and the number of nodal degrees of freedom that determines the size of the stiffness matrix. An element may contain corner nodes, side nodes and/or interior nodes. The degrees of freedom are usual1y referred to the displacements and their first-order partial derivatives at a node but very often include second or higher order partial derivatives.
- The theory that determines the stress-strain and strain-displacement relationships to be used in deriving the element matrices.
- The displacement functions (simple polynomial) or interpolation functions in terms of the coordinate variable and a number of constants (equal to the total number of degrees of freedom in the element). The displacement functions are then chosen to represent the variation of the displacements within each element.

By using the principle of virtual work or the principle of minimum potential energy, a stiffness matrix relating the nodal forces to the nodal displacements can be derived. Hence, the choice of suitable displacement functions is the major factor to be considered in deriving element stiffness matrices.

1.3.3 General Procedure for Derivation of Element Stiffness Matrix

A stiffness matrix expresses the relation between the nodal loads applied to the element, and the nodal displacements. Such a relation can be derived from consideration of geometry, relations in the theory of elasticity and the conditions of equilibrium [21]. Thus, if $|\Delta|$ is the vector containing the displacement functions of the element, $|\sigma|$ is the vector containing the stresses and $|\varepsilon|$ is the corresponding strains, and if:

$$\{\Delta\}=[\,f\,]\{A\} \qquad \textbf{Eq. 1-1}$$

where $[\,f\,]$ is the matrix containing the coordinate variables (x, y, etc.) and $|A|$ is the vector of constant terms (a1, a2, … etc.) of the displacement function $|\Delta|$ respectively, we can apply equation (1-1) to the nodes of the element to relate the displacement within the element to its nodal displacements then we obtain:

$$\{\delta\}=[\,C\,]\{A\} \qquad \textbf{Eq. 1-2}$$

From which $|A|=[\,C\,]^{-1}|\delta|$ **Eq. 1-3**

where $|\delta|$ is the vector containing the nodal degrees of freedom of the element and $[\,C\,]$ is the transformation matrix resulting from substitution the coordinates of each nodal point into the $[\,f\,]$ matrix. Therefore, by substituting equation (1-3) into equation (1-1), the latter becomes:

$$|\Delta|=[\,f\,][\,C\,]^{-1}|\delta| \qquad \textbf{Eq. 1-4}$$

Now, we can express the strains by using the fact that the strains are the derivatives of the displacements and by using equation (1-4):

$$\{\varepsilon\}=[\,B\,][\,C\,]^{-1}\{\delta\} \qquad \textbf{Eq. 1-5}$$

where $[\,B\,]$ is called the strain matrix and it contains the necessary derivatives of $[\,f\,]$ corresponding to the strain-displacement relationship. Also from Hook's la w, the stresses within the element can be expressed as

$$\{\sigma\}=[\,D\,]\{\varepsilon\} \qquad \textbf{Eq. 1-6}$$

Thus from (1-6) and (1-5)

$$\{\sigma\}=[\,D\,][\,B\,][\,C\,]^{-1}\{\delta\} \qquad \textbf{Eq. 1-7}$$

where $[\,D\,]$ is called the rigidity matrix that contains the material properties.

In order to relate the applied nodal loads to the nodal displacements, the nodal loads should first be related to the internal stresses, using the conditions of equilibrium, and then to the nodal displacements using equations (1-5) & (1-7). Using the principle of minimization of the potential energy, we can derive the stiffness of the element. The total potential energy, Π, of an element is the difference between the strain energy stored by the internal stresses, U, and the potential energy of the applied loads Ω,

$$\Pi = U - \Omega \qquad \textbf{Eq. 1-8}$$

An expression for the strain energy may be written as:

$$U = \frac{1}{2}\iiint_{vol} |\varepsilon|^{T}\,|\sigma|\,dvol \qquad \textbf{Eq. 1-9}$$

Using equations 1-5 and 1-6, we get

$$= \frac{1}{2}\int |\delta|^{T}\left[C^{-1}\right]^{T}[B]^{T}\;[D][B][C]^{-1}\,|\delta|\,dvol \qquad \textbf{Eq. 1-10}$$

while the potential energy of the applied loads Ω (including the contribution of body forces and surface traction forces) is written as

$$\Omega = \{\delta\}^{T}\{P\} \qquad \textbf{Eq. 1-11}$$

So we have

$$\Pi = \frac{1}{2}\{\delta\}^{T}\int \left[C^{-1}\right]^{T}[B]^{T}\;[D][B][C]^{-1}\,dvol\;\{\delta\} - \{\delta\}^{T}\{P\} \qquad \textbf{Eq. 1-12}$$

where $|\delta|$ has been taken out of the integral, as it is independent of the general x-y coordinates. Now, by differentiating Π and equating it to zero, we get

$$\frac{d\Pi}{d\delta} = \int \left[C^{-1}\right]^{T}[B]^{T}\;[D][B][C]^{-1} dvol\;\{\delta\} - \{P\} = 0$$

or, $\{P\} = \left[K^{e}\right]\{\delta\}$

where

$$\left[K^{e}\right] = \left[C^{-1}\right]^{T}\left[\int [B]^{T}\,[D][B]\;dvol\;\right][C]^{-1} \qquad \textbf{Eq. 1-13}$$

is the element stiffness matrix.

1.3.4 Assembly of the Overall Structural Matrix and the Solution Routine

To obtain a solution for the overall system modeled by a network of elements, we must first "assemble" all the elements' stiffness matrices. In other words, we must combine the matrix equations expressing the behavior of the entire structure. This is done using the principle of superposition; it is also called the "*direct stiffness method*". The basis for the assembly procedure stems from the facts that:

- At a node where elements are interconnected, the value of the field variable (generated displacements) is the same for each element sharing that node so that the structure remains together and no tearing or overlap occur anywhere in the structure.
- Equilibrium is satisfied at each node. i.e. the sum of all the internal nodal forces meeting at a node must be equal to the externally applied forces at that node.

Therefore, to implement these two facts a simple computer program can be written and used for the assembly of any number of elements. The resulting stiffness matrix of the element (and hence that of the total structures) is symmetrical and singular matrix. So, the resulting simultaneous equations can be solved, after the introduction of the boundary conditions to the specific problem to obtain the nodal displacements and these are then used for the calculation of the stresses.

1.3.5 Convergence Criteria

A characteristic of the finite element method is that the results should approach the exact values as more and more elements are used. With good displacement functions, convergence towards the exact value will be much faster than with poor functions, thus resulting in a reduction of the modeling and computing time and effort. In order to achieve the convergence towards the exact value of the required variables, the displacement functions chosen should try to represent the true displacement distribution as close as possible and should have certain properties, known as "convergence criteria". These are conditions to guarantee that exact answers, within the approximation made, will be approached as more and more elements are used to model the arbitrary structure. These criteria are discussed next.

A- Rigid-Body Modes

The approximation function should allow for rigid-body displacement and for a state of constant strain within the element. For example, the one-dimensional displacement function $u = a_1 + a_2 x$ satisfies this criterion because the constant (a_1) allows for rigid body displacement (constant motion of the body without straining) while the term (a_2 x) allows for a state of constant strain ($ex = du/dx = a_2$). This simple polynomial is then said to be "complete" and is used for the one-dimensional bar element. Completeness of the chosen displacement function is a necessary condition for convergence to the exact values of displacements and stresses [15], [35]. The inclusion of rigid body modes is necessary for equilibrium of the nodal forces and moments and hence the satisfaction of global equilibrium in the structure being analyzed [21].

B- Constant Strain

In order for the solution to converge to the actual state of strains, the approximation function should also allow for a state of constant strain within the element. The state of the constant strain in the element can occur if the elements are chosen small enough.

This requirement is obvious for structures subjected to constant strains because as elements get smaller, nearly constant strain conditions prevail in them. As the mesh becomes finer, the element strains are simplified, and in the limit they will approach their constant values.

C- <u>Inter-element Compatibi1ity</u>

The concept of compatibility means that the displacements within the elements and across the boundaries are continuous. It has been shown that the use of complete and compatible displacement functions is a basic guarantee to obtain converging solutions as more elements and nodes are used (the mesh size is refined).

Note that the above criteria are mathematically included in the statement of "Functional Completeness".

1.3.6 Mesh Size Design

The finite element analysis (FEA) uses a complex system of points called nodes that make up a grid called the mesh. Nodes are assigned at a certain density throughout the material depending on the anticipated stress levels at particular areas. Regions that will receive larger amounts of stress usually have a higher node density than those with little or no stresses.

In order to conduct a finite element analysis, the structure must be first idealized into some form of a mesh. Meshing is the procedure of applying a finite number of elements to the FEA model. The art of a successful application of the meshing task lies in the combined choice of element types and the associated mesh size. If the mesh is too coarse, then the solution will not give correct results. Alternatively, if the mesh is too fine, the computing time and effort can be out of proportion of the results obtained. A coarse mesh is sufficient in areas where the stress is relatively constant while a fine mesh is required where there are high rates of changes of stress and strain. Local mesh refinement may be used at area of maximum stress states.

To ensure that convergent results of the FEA solution are obtained, the following modeling guidelines should be considered in the design of the mesh:

A- Number of Degree of Freedom

It is the essence of the finite element method that increasing the total number of degree of freedom (calculated as the number of nodes times the number of DoF per node) has the basic influence on the convergence of the FEA solution towards a true solution.

B- Aspect Ratio

The aspect ratio is defined as the ratio between the longest and shortest element dimensions. Acceptable ranges for aspect ratio are element and problem dependent. It is generally known that the accuracy of the solution deteriorates as larger aspect ratio is used.

The limit of the aspect ratio is affected by the order of the element displacement function, the numerical integration pattern for stiffness, the material behavior (linear of nonlinear) and the anticipated solution pattern for stresses or displacements. Elements with higher-order displacement functions and higher-order numerical integration are less sensitive to large aspect ratios. Elements in regions of material non-linearity are more sensitive to changes in aspect ratio than those in linear regions. However, fixed numerical limits are given such as 3:1 for stresses and 10:1 for deflections [9].

C- Element Distortions (Skewing)

Distortions of elements or their out-of-plane warping are important considerations. Skewing is usually defined as the variation of element vertex angles from 90º for quadrilaterals and from 60º for triangles. Warping occurs when all the nodes of three-dimensional plates or shells do not lie on the same plane or when the nodes on a single face of a solid deviate from a single plane [9]. These concepts are illustrated in Figure 1-1 below.

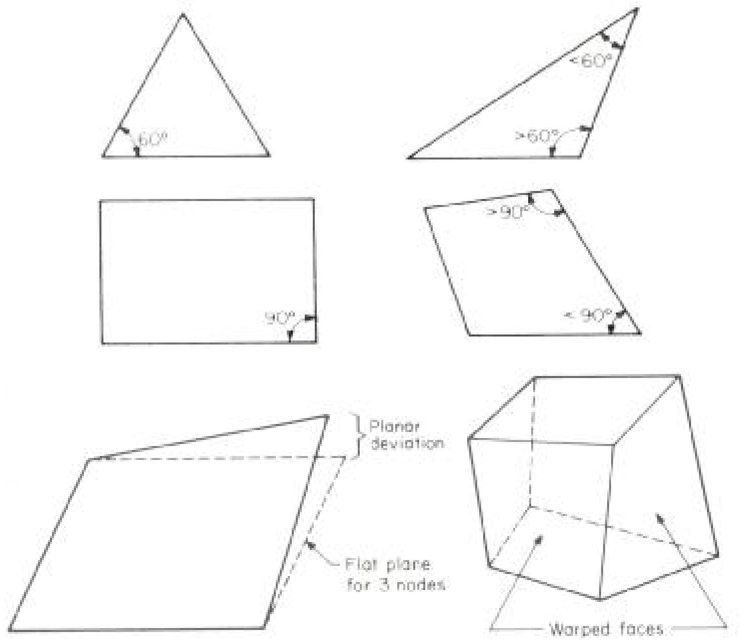


Figure 1-1: Skewed And Warped Elements (Poor Shaped)

1.4 FINITE ELEMENT APPLICATIONS OF TWO-DIMENSIONAL STRUCTURES

The finite element solution of two-dimensional structures is made by dividing the structure into a mesh of elements, mainly triangular and/or rectangular elements. There are two main types of elements; the basic elements are those having only corner nodes such as the constants strain triangular element (CST) and the bilinear rectangular element (BRE). The more advanced elements are those who have additional mid-side nodes such as the linear strain triangular element (LST) and the (quadratic iso-parametric element). The use of mid-side nodes allows quadratic variation of strains and hence faster rate of convergence. Another advantage of the use of higher order elements is that curved boundaries of irregularly shaped structures can be approximated more closely than by the use of simple straight-sided elements. The goal of developing all these elements is to introduce more degrees of freedom into the solution and hence get more accurate results as well as better modeling of various structure boundaries.

The following is a brief description of the basic features of the commonly used elements for two-dimensional problems [15].

1.4.1 Constant Strain Triangular Element (CST)

The constant strain element is the basic and most common element used for the solution of plane elasticity problems. This element has two degrees of freedom (U and V displacements) at each of its three corner nodes, thus it has a total of six degrees of freedom.

The element is based on independent displacement fields in the x and y directions, i.e. the constants appearing in the expression for the displacement in one direction do not appear in the expression for the other direction as follows:

$$U = a_1 + a_2\, x + a_3\, y$$
$$V = a_4 + a_5\, x + a_6\, y$$

The associated strains for this element are given by

$$\varepsilon_x = \frac{\partial U}{\partial x} = a_2$$
$$\varepsilon_y = \frac{\partial V}{\partial y} = a_6$$
$$\gamma_{xy} = \frac{\partial U}{\partial y} + \frac{\partial V}{\partial x} = a_3 + a_5$$

It is obvious that the values of strain components, and so the stresses, are constant and do not vary throughout the element, hence came the name. This means that satisfactory convergence towards the exact solution can only be ensured by using a large number of elements [9], [15] & [35].

1.4.2 Liner Strain Triangular Element (LST)

This element has six nodes, i.e. three corner nodes as well as three mid-side nodes. This is considered as a development over the CST [9], [15] & [35]. It is a higher order triangular element that has twelve degrees of freedom. Its strains vary linearly with x and y coordinates. The element is based on independent displacement fields in the x and y directions as follows:

$$U = a_1 + a_2\, x + a_3\, y + a_4\, x^2 + a_5 x\, y + a_6\, y^2$$
$$V = a_7 + a_8\, x + a_9\, y + a_{10}\, x^2 + a_{11} x\, y + a_{12}\, y^2$$

The associated strains for this element are given by

$$\varepsilon_x = \frac{\partial U}{\partial x} = a_2 + 2\,a_4\, x + a_5\, y$$
$$\varepsilon_y = \frac{\partial V}{\partial y} = a_9 + a_{11}\, x + 2\,a_{12}\, y$$
$$\gamma_{xy} = \frac{\partial U}{\partial y} + \frac{\partial V}{\partial x} = (a_3 + a_8) + (a_5 + 2a_{10})\, x + (2\,a_6 + a_{11}) y$$

1.4.3 Bilinear Rectangular Element (BRE)

This rectangular element is also a basic and common element used for the solution of plane elasticity problems. This element has two degrees of freedom (U and V displacements) at each of its four corner nodes, thus it has a total of eight degrees of freedom. The element is based on independent displacement fields in the x and y directions as follows:

$$U = a_1 + a_2\, x + a_3\, y + a_4\, x\, y$$
$$V = a_5 + a_6\, x + a_7\, y + a_8\, x\, y$$

The associated strains for this element are given by

$$\varepsilon_x = \frac{\partial U}{\partial x} = a_2 + a_4\, y$$
$$\varepsilon_y = \frac{\partial V}{\partial y} = a_7 + a_8\, x$$
$$\gamma_{xy} = \frac{\partial U}{\partial y} + \frac{\partial V}{\partial x} = a_3 + a_4\, x + a_6 + a_8\, y$$

The strain components, and hence the stresses, vary linearly in both the x and y directions, hence come its name. This means that satisfactory convergence towards the exact solution can be achieved better than the CST mentioned above [9], [15] & [35].

1.4.4 Quadratic Isoparametric Rectangle Element

This element has eight nodes, i.e., four corner nodes as well as four mid-side nodes. This is considered as a development over the BRE. It is a higher order rectangular element that has sixteen degrees of freedom [9], [15] & [35].

All of the above-mentioned elements are based on assumed polynomial displacements. Another approach called the "Strain Based Approach" exists where assumed polynomial strains are used for deriving the displacement fields. This new approach is applied in the present work for deriving new displacement fields of triangular and rectangular elements with three degrees of freedom pre node, which are the two translations and the in-plane rotation.

1.5 SCOPE OF THE CURRENT THESIS

The purpose of the work presented in this thesis is to derive two new strain-based elements for two dimensional elasticity problems.

In Chapter 4, a new strain based triangular element having three degrees of freedom per node is derived. The element is then applied to solve two problems in plane elasticity, i.e., a deep cantilever beam problem and a simply supported beam problem. Convergence of the solutions for deflection and stresses is studied with mesh refinement. The performance of this element is compared to that of the available displacement based Constant Strain Triangle element, CST and the exact elasticity solutions.

In Chapter 5, a new strain based rectangular element having three degrees of freedom per node is derived. The element is then applied to solve the same problems mentioned above. Also, convergence of the solutions using this element is studied. The performance of this element is compared to that of the available displacement based Bilinear Rectangular Element, BRE and the exact elasticity solutions.

In Chapter 6, the performance of these elements is compared for the results of deflection and stresses.

2. CHAPTER TWO: STRAIN BASED APPROACH

2.1 INTRODUCTION

A new approach to develop the stiffness matrix of finite elements was proposed by Ashwell, Sabir and Roberts [1] & [3]. In this approach, polynomial strain components are assumed. Then the displacement fields are obtained by integration of strain components according to the relevant strain/displacement relations. The displacement fields have two essential components. The first component relates to the zero-strain rigid body mode of displacement while the second is due to the straining of the element, which can be represented by assuming independent polynomial terms of strains in so far as it is allowed by the compatibility equations governing the changes in the direct stresses and shear stresses.

A main feature of the strain-based approach is that the resulting components of displacements are not independent as in the usual displacement approach but are linked. This linking is present in the exact terms representing rigid body modes and the approximate terms within the context of the finite element method representing the straining of the element.

Another feature of this approach is that the method allows the in-plane components of the displacements to be presented by higher order terms without increasing the number of degrees of freedom beyond the essential external degrees of freedom.

2.2 REASONS FOR DEVELOPMENT OF STRAIN BASED ELEMENTS

The reasons for seeking an element based on generalized strain functions rather than displacements were given by the authors who developed this approach [3]. They wrote that:

> "Firstly if we wish to minimize the contribution of strain energy to the potential energy of an element we should seek variations of strain which are as smooth as possible. This consideration follows from the fact that the strain energy is calculated from squares and products of the strain, and imposing local variations on an initially smooth distribution without altering the local mean values increases the value of the squares when they are integrated over the element.
>
> Secondly the equations relating displacements and generalized strains are coupled in such away that some strains are functions of more than one displacement thus making displacements independent of one another will not make the strains independent of each other. In addition, since two rules in the convergence criteria and directly concerned with strains they would be easier to satisfy with assumed strains rather than displacement functions".

2.3 HISTORY OF STRAIN BASED APPROACH

2.3.1 Strain Based Curved Elements

The development of displacement fields by the use of the Strain Based Approach was first applied to curved elements. It was revealed that to obtain satisfactory converged results, the finite elements based on independent polynomial displacement functions require the curved structures to be divided into a large number of elements.

Ashwell, Sabir and Roberts [1] & [3] showed that when assumed independent polynomial displacement fields are used in the analysis of curved elements, the structure needs to be divided into large number of elements in order to get satisfactory converged results. However, when assumed independent polynomial strain fields are used, converged results are obtained even when less divisions of the structure are used, i.e., the strain based elements showed faster convergence. Therefore, they continued to develop a new class of simple and efficient finite elements for various types of problems based on assumed independent strains rather than independent displacement fields.

2.3.2 Strain Based Shell Elements

- Sabir and Ashwell [1] presented a strain-based rectangular cylindrical element that has twenty degrees of freedom. It uses only external geometrical nodal displacements (three linear displacement and two rotations). It includes all rigid body displacements exactly and satisfies the constant strain condition in so far as that condition applies to cylindrical shells.
- Because the rectangular elements which have been developed can not be used for modeling shells having irregular curved boundaries, Sabir and Charchafchi [25] used the strain approach to develop a quadrilateral element and Sabir used this element to investigate the problem of stress concentration in cylinders with circular and elliptical holes [24] and the problem of normally intersecting cylinders [28].
- Sabir et al also used the strain approach to develop element for arches deforming in the plane [2] and out of the plane containing the curvature [27] and took the opportunity to show that higher order strain based elements can be obtained and also can be condensed to the only essential external degrees of freedom. This statical condensation at the element level was shown to produce further improvement in the convergence of the result.

- In 1985, Sabir and Ramadanhi [26] developed a simple curved finite element based on shallow shell equations. The element is rectangular in plane and has three principal curve lines and has the only essential external degrees of freedom. The element presented is based on assumed strains rather than displacement fields. The element was tested by applying it to analysis of cylindrical [26], spherical [29] and hyperbolic parabolic shell and it gave high degree of accuracy. Convergence of the results for deflection as well as stresses was more rapid when compared with other finite elements based on assumed displacements.
- Sabir and El-Erris [8] also developed a new curved strain-based hyperbolic parabolic shell element similar to that developed by Sabir and Ramadanhi but having the in plane rotation as a sixth degree of freedom. The results obtained by the use of this element were shown to converge more rapidly for a variety of problems even when compared with high order elements.
- Sabir and El-Erris developed a curved conical shell finite element suitable for general bending analysis of conical shells. This element is simple and possesses all the necessary requirements for less computational effort. The element has 20 degrees of freedom and satisfies the exact representation of rigid body modes of displacement. The convergence characteristics of the element were tested by applying to the bending analysis of conical shells and it was shown that results of acceptable level of accuracy are obtained when few elements are used.
- Djoudi [6] developed a curved triangular shallow shell element. The element has only the five essential degrees of freedom at each node and is based on assumed strains. Several examples of shells with different loading and boundary conditions were considered and the results obtained were shown to be satisfactory for most problems.
- The strain approach was also used in developing several strain based shell elements were developed by Mousa, A. [17] to [22]. These elements include conical, cylindrical and spherical shell elements. Also, two groups of doubly curved triangular elements were developed, the first group included

three elements that have five degrees of freedom at each node while the second group included two elements that have six degrees of freedom at each node. These elements gave high accuracy of results for displacements and components of stresses when they were used in complete analysis of a variety of complex structures such as:

- Complex type of shell roof referred to as fluted roof, with studying the effect of using stiffening beams on deflections.
- Cylindrical storage tank (made up of cylindrical and conical shell components) that exhibits large concentration of stresses at the junction of the two components with studying the effect of using stiffening ring beams on stresses at the junction.
- Doubly curved hyperbolic parabolic dam with constant or variable thickness,
- Doubly curved spherical shell roof in the form of a four-corner star with studying the effect of using stiffening beams on deflections.

2.3.3 Strain Based Two Dimensional Elements

The strain-based approach was further extended by Sabir [23] to the two dimensional plane elasticity problems. A new family of such elements was developed. These elements satisfy the requirements of strain-free rigid body mode of displacement and the compatibility within the element.

§ A triangular in-plane element was developed having the two basic degrees of freedom. The resulting displacement fields of this element are given by

$$U = a_1 - a_3\, y + a_4 x + \frac{a_6}{2}\, y$$
$$V = a_2 + a_3\, x + a_5\, y + \frac{a_6}{2}\, x \qquad \textbf{Eq. 2-1}$$

However, this element gave no improvement since its performance was found to be exactly the same that of the displacement based Constant Strain Triangular element.

§ A basic rectangular element was developed and tested by applying it to the two dimensional analysis of a beam and plate with holes. Another version of this element satisfying the above requirements as well as satisfying equilibrium equations was also developed and tested. These elements have the two essential translational degrees of freedom at each of the corner nodes. The resulting displacement fields of this element are given by

$$\begin{aligned} U &= a_1 - a_3\,y + a_4\,x + a_5\,x\,y - \frac{a_7}{2}y^2 + \frac{a_8}{2}y \\ V &= a_2 + a_3\,x - \frac{a_5}{2}x^2 + a_6\,y + a_7\,x\,y + \frac{a_8}{2}x \end{aligned} \qquad \textbf{Eq. 2-2}$$

This element was applied to solution of several plane elasticity problems such as a deep cantilever beam and a simply supported beam and was found to give good results and fast rate of convergence.

§ The displacement fields of the above mentioned rectangular element developed by Sabir was applied by Sfendji [31] to a triangular element with four nodes (three corners and a mid-hypotenuse node) and using the statical condensation of two such triangular elements. Based on the same rectangular element, he also derived two new rectangular elements satisfying the equilibrium equations for plane stress and plane strain respectively.

The displacement fields of these elements are given by

$$\begin{aligned} U &= a_1 - a_3\,y + a_4\,x + a_5\,x\,y - \frac{a_7}{2}(x^2 + y^2) + \frac{a_8}{2}y - \frac{a_{10}}{2}\left(\frac{1-\nu}{2\nu}x^2 - y^2\right) \\ V &= a_2 + a_3\,x - \frac{a_5}{2}(x^2 + y^2) + a_6\,y + a_7\,x\,y + \frac{a_8}{2}x - \frac{a_9}{2}\left(\frac{1-\nu}{2\nu}y^2 - x^2\right) \end{aligned} \qquad \textbf{Eq. 2-3}$$

for plane stress and

$$\begin{aligned} U &= a_1 - a_3\,y + a_4\,x + a_5\,x\,y - \frac{a_7}{2}(\nu x^2 + y^2) + \frac{a_8}{2}y - \frac{a_{10}}{2}\left(\frac{1-\nu}{2}x^2 - y^2\right) \\ V &= a_2 + a_3\,x - \frac{a_5}{2}(x^2 + \nu y^2) + a_6\,y + a_7\,x\,y + \frac{a_8}{2}x - \frac{a_9}{2}\left(\frac{1-\nu}{2}y^2 - x^2\right) \end{aligned} \qquad \textbf{Eq. 2-4}$$

for plane strain.

§ A sector finite element was developed in polar coordinates by Djoudi [6]. This element has three degrees of freedom at each node (two essential degrees of freedom and the in-plane rotation). It was applied to the two analysis of rotationally symmetric curved beam subject to end shear.

2.4 GENERAL PROCEDURE FOR DERIVING DISPLACEMENT FIELDS FOR STRAIN-BASED ELEMENTS

The following are general guidelines of deriving displacement fields for strain-based elements.

- The displacement fields are required to satisfy the requirement of strain-free rigid body mode of displacement and straining of the element.
- To get the first part of the displacement fields corresponding to rigid body movement of the element, we begin by writing the governing strain/displacement relationships to the considered element type (2 dimensional, 3 dimensional, flat, curved, etc) and make them equal to zero.
- The resulting equations are applicable to any type of finite elements of that type regardless of the total number of degrees of freedom per node.
- Depending on the number of nodes and the number of degrees of freedom per node in the considered element, it is generally essential that the total number of degrees of freedom in the element (and hence the number of constants used in defining the displacement fields within the element) equals the number of nodes times the number of degrees of freedom per node.
- Some of the required constants will have already been used to describe the strain-free rigid body mode of displacement as described above.
- To get the remaining part of the displacement field, corresponding to the straining of the element, an expression is assumed for each of the strain components utilizing the remaining number of constants.
- The assumed expression should be checked to ensure that if they are twice differentiated, they satisfy the general compatibility equation of strains. These expressions should include constant terms corresponding to the state

of constant strain that ensures the convergence of the solution with mesh refinement.

- The parts of the strain/displacement relationships involving direct strains are integrated to give expressions of direct displacements (for example U and V) that will include integration constants.
- These expressions are substituted in the equations that link them to other parts of the strain/displacement relationships (for example, the shear strain equals the derivatives of U and V). The terms corresponding to each of the coordinate variables (for example x and y) are collected and separated. This will give the values of integration constants.
- By combining the displacement functions due to the rigid body motion and those due to the straining of the element, we get the final expression of the displacement fields.
- To compare the strain-based elements with the commonly used displacement-based elements, it is noted that the displacement fields of the strain-based element are linked through the terms representing both the rigid body mode as well as the straining of the element, i.e. most of the constant terms appear in expression for each of the displacement fields.
- After calculating the displacement fields, the element stiffness matrix can be calculated using the general expression

$$\left[K^e\right]=\left[C^{-1}\right]^T.\int[B]^T.[D].[B].d(vol).\left[C^{-1}\right] \qquad \textbf{Eq. 2-5}$$

where the transformation matrix, [C] is calculated by substituting the value of displacement variables (x, y, etc.) at each node. For example, for elements with two degrees of freedom per node, [C] is calculated as:

$$[C]=\begin{bmatrix} U_1\ @\,(x_1,y_1) \\ V_1\ @\,(x_1,y_1) \\ U_2\ @\,(x_2,y_2) \\ V_2\ @\,(x_3,y_2) \\ --- \\ --- \\ --- \\ U_n\ @\,(x_n,y_n) \\ V_n\ @\,(x_n,y_n) \end{bmatrix}$$

, where n is the total number of nodes.

Matrix $[B]$ is the strain matrix involving the coordinate variables attached to each of the constant terms in the expression for the various strain components. And $[D]$ is the rigidity matrix relating the strains to stress and using the material properties (mainly modulus of elasticity and Poisson's ratio in structural solid mechanics and plane elasticity problems).

3. CHAPTER THREE: COMPUTER PROGRAMS

3.1 USED COMPUTER SOFTWARE

Two main programs were used for the derivation of the new finite elements and computer implementation of the analysis. A description of each program is given below.

3.1.1 MathCAD Software

MathCAD is a powerful mathematics software for technical calculations. The program is used to implement almost all kinds of mathematical operations in an easy way. The program interface is just like an open page so the user can write anywhere in this page. The basic feature of the program is that it implements symbolic mathematics such as simplifying expressions, differentiation, integration, collecting variables, factoring terms, etc.

In this thesis, MathCAD is used to derive the displacement fields for triangular and rectangular elements as will be detailed in the next chapters. In this regard, the strains are assumed then the remaining symbolic analysis are made until the complete expressions for displacement fields are completed and then the adequacy of the resulting transformation matrix is checked by applying it to an arbitrarily oriented element. The use of MathCAD makes it easy to change the assumptions for strains and see the resulting displacement fields immediately.

3.1.2 MatLab Software

MatLab is an interactive system for doing numerical computations. The program got its name from the fact that it is a "Matrix Laboratory" because it is based on matrix manipulations, which produce high efficiency in calculations. Each entry into the MatLab is treated as a matrix, even if it is a single number. Computation speed of the program is incredible so it is very suitable to be used to solve the large number of simultaneous equations resulting from finite element analyses. For example, the inverse of a very large stiffness matrix is calculated using one command in contrast to the old programming languages where that task usually requires a large amount of programming effort.

3.2 IMPLEMENTATION OF THE FEM ANALYSIS

Special purpose computer programs were developed using MatLab to generate the mesh data including node coordinates and element connectivity for the cases of triangular and rectangular elements within each problem domain.

Furthermore, another program was developed to solve the problems using each element type, i.e. the existing CST and BRE elements as well as the new triangular and rectangular elements, which will be developed in the following chapter of this thesis.

Input data was written in a separate file to organize the data and facilitate the process of generating the nodal coordinates and element connectivity (mesh definition). Output of the program was also received into a separate file. The output included the input data, nodal forces, nodal displacements and element stresses at the nodes of each element.

Samples of the used computer programs are shown in "Appendix A" including:

A-1: MatLab Code for Generation of Triangular Mesh in Rectangular Domain

A-2: MatLab Code for Generation of Rectangular Mesh in Rectangular Domain

A-3: MatLab Code for the Strain Based Triangular Element with In-plane Rotation (SBTREIR)

A-4: MatLab Code for the Strain Based Rectangular Element with In-plane Rotation (SBREIR)

A-5: Sample Input File (Rectangular Element)

A-6: Sample Output File (Rectangular Element)

3.2.1 Outline of the Major Program Steps

The following is a summary of the basic stages that one has to go through when implementing the MatLab developed computer program.

- Start
- Ask for input file name to read input data
- Ask for output file name to write results
- Read input file:
  - Total number of nodes in the structure
  - Total number of elements in the structure
  - Node coordinates
  - Element connectivity data
  - Material Data: Modulus of elasticity, Poisson's ratio, thickness
  - Nodal forces
  - Nodal fixation data (boundary conditions)
- Plot the structure to ensure correctness of data
- Open output file and prepare headings to write results
- Write the input data into the output file
- Prepare sampling points and weights to be used in numerical integration
- Calculate the rigidity matrix for (plane stress / plane strain)
- Start calculating element stiffness matrix:
  - Read element coordinates
  - Calculate the transformation matrix [C]
  - Assign zero matrix for the element stiffness matrix,

- For each of the sampling points:
  - Calculate the value shape functions and their derivatives.
  - Calculate the strain matrix [B].
  - Calculate the value of the stiffness matrix and accumulate it.

§ Assemble the global stiffness matrix of the structure [GK]

§ Apply boundary conditions to stiffness matrix and nodal force vector {F}

§ Solve for global nodal displacements {GD} = inv [GK].{F}

§ Write nodal displacements to the output file

§ Prepare headings to write stresses into the output file

§ Start calculating the element stresses:

- Read element coordinates again
- Extract element nodal displacements {d} from the global displacements [GD]
- Calculate the transformation matrix [C] again
- Calculate the rigidity matrix [D]
- Calculate the strain matrix [B]
- Calculate the element stresses = [D].[B].inv[C].{d}
- Write element stresses to the output file

§ End

3.2.2 Programs Flow Chart

The following figure shows the usual flow chart of processes involved in the implementation of the finite element analysis.

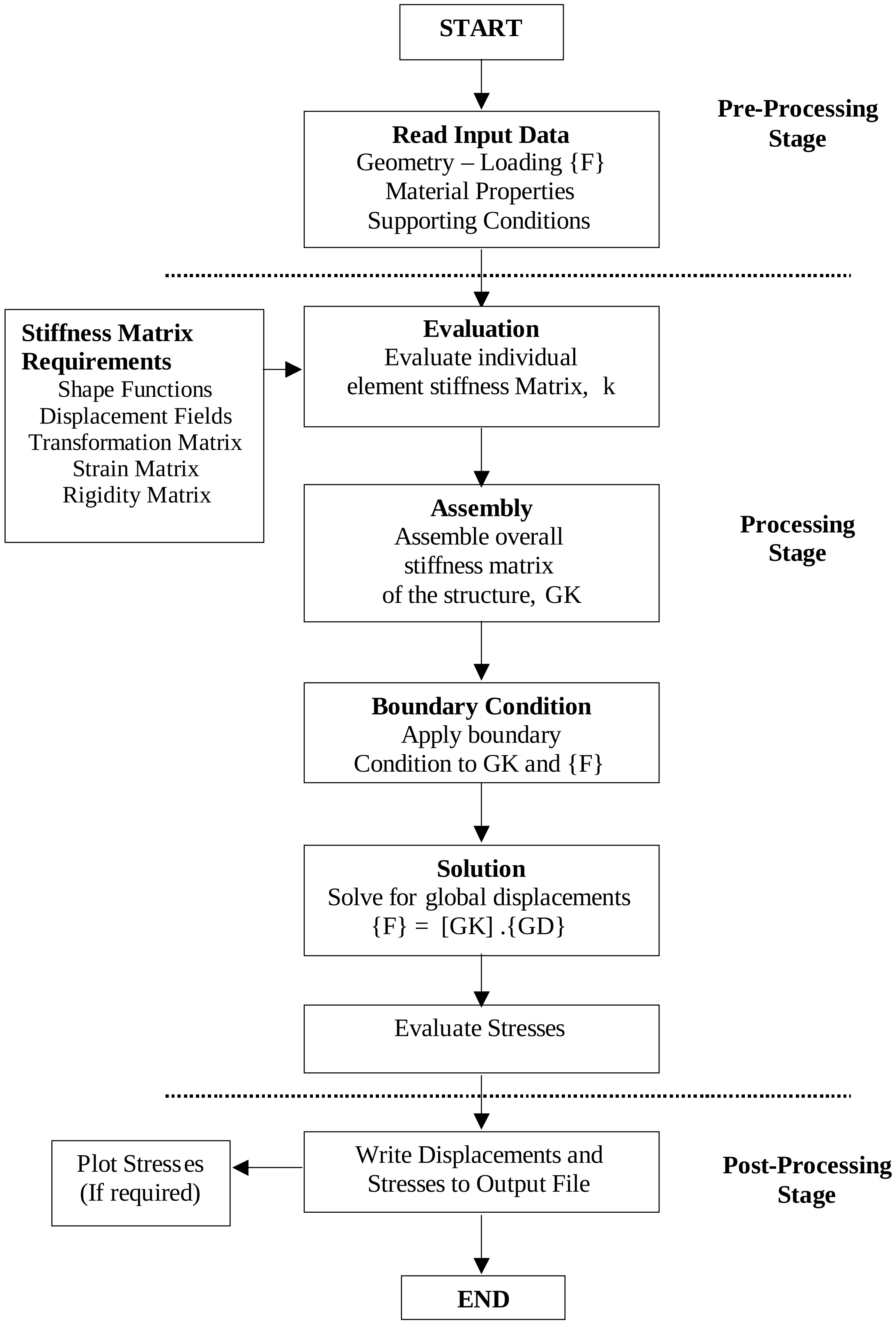


Figure 3-1: Finite Element Analysis Flow Chart

4. CHAPTER FOUR: DEVELOPMENT OF NEW STRAIN-BASED TRIANGULAR ELEMENT

4.1 INTRODUCTION

In this chapter, the strain-based approach is extended to investigate the ability of deriving displacement fields for a new strain based triangular element with the inclusion of the third degree of freedom at each node, which is the in-plane rotation (also called drilling degree of freedom).

The performance of the new triangular element is investigated by applying it to the solution of two common plane elasticity problems. These problems are: the problem of a plane deep cantilever beam fixed at one end and loaded by a point load at the free end and the problem of a simply supported beam loaded at the mid-span by a point load.

The results obtained by the developed triangular strain based element are compared to those given by the well known Constant Strain Triangle (CST) and the analytical solutions for deflection and stresses as detailed below.

4.2 DERIVATION OF DISPLACEMENT FIELDS OF NEW TRIANGULAR ELEMENT WITH IN-PLANE ROTATION

The following outlines the assumptions and steps to derive the new strain-based triangular element.

- The new triangular element has three corner nodes with three degrees of freedom at each node as shown in the figure below.
- The displacement fields must satisfy the requirement of strain-free rigid body mode of displacement and straining of the element.

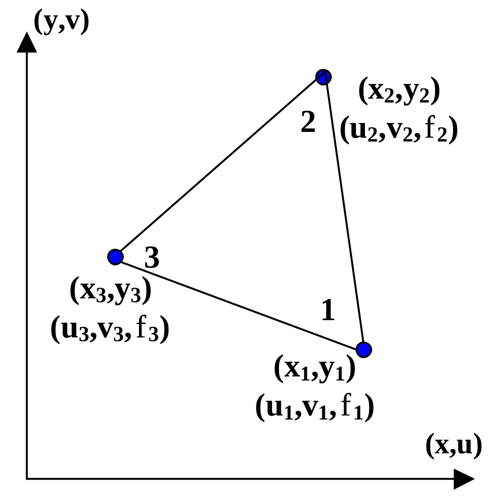


Figure 4-1: Coordinates and Node Numbering for Triangular Element with 3 DOF per node

- To get the first part of the displacement fields for rigid-body mode (U_1 and V_1), we begin by writing the strain/displacement relationships for plane elasticity and make them equal to zero. These relationships are given by:

$$\varepsilon_x = \frac{\partial U}{\partial x} \qquad \varepsilon_y = \frac{\partial V}{\partial y} \qquad \gamma_{xy} = \frac{\partial U}{\partial y} + \frac{\partial V}{\partial x} \qquad \textbf{Eq. 4-1}$$

where

U and V : are the displacements in the x and y directions respectively.

ε_x and ε_y : are the direct axial strains in the x and y directions respectively.

γ_{xy} : is the shear strain.

Next these stains are set equal to zero and then integrated, they will give

$$\varepsilon_x = \frac{\partial U_1}{\partial x} = 0 \quad ===> U_1 = a_1 + f_1(y)$$

$$\varepsilon_y = \frac{\partial V_1}{\partial y} = 0 \quad ===> V_1 = a_2 + f_2(x)$$

$$\gamma_{xy} = \frac{\partial U_1}{\partial y} + \frac{\partial V_1}{\partial x} = 0 \quad ===> f_1'(y) + f_2'(x) = 0$$

where $f_1'(y)$ and $f_2'(x)$are constants that can be taken as

$f_1'(y) = -a$ and $f_2'(x) = a_3$

hence,

$f_1(y) = -a_3\,y$ and $f_2(x) = a_3\,x$

thus,

$$U_1 = a_1 - a_3 y$$
$$V_1 = a_2 + a_3 x \qquad \textbf{Eq. 4-2}$$

where

U_1 and V_1 : are the displacements in the x and y directions respectively corresponding to rigid-body mode of displacement.

a_1 and a_2 : represent the translations of the element in the x and y directions respectively.

a_3 : represent the rigid-body in plane rotation of the element.

It is noted that equations Eq. 4-2 are applicable to any type of two dimensional plane finite elements regardless of the total number of degrees of freedom per node.

- Depending on the number of nodes and the number of degrees of freedom per node in the considered element, it is generally essential that the total number of degrees of freedom in the element (and hence the number of constants used in defining the displacement fields within the element) equals the number of nodes times the number of degrees of freedom per node.
- Three constants have already been defined while the remaining six have to be used to describe the straining of the element. A first attempt to do so is to assume that

$$\varepsilon_x = a_4 + a_5 y$$
$$\varepsilon_y = a_6 + a_7 x \qquad \textbf{Eq. 4-3}$$
$$\gamma_{xy} = a_8 x + a_9 y$$

- This arrangement of strains does not contain a constant term in the expression for γ_{xy} and it is expected that it wouldn't give good solutions. Also it was found that it leads to a singular displacement transformation matrix and hence it can't be used to derive a stiffness matrix.
- Several other arrangements were tried to avoid this problem. A good arrangement that gives non-singular transformation matrix is found to be as follows:

$$\varepsilon_x = a_4 + a_5 y + a_9 \frac{y^2}{4}$$
$$\varepsilon_y = a_6 + a_7 x - a_9 \frac{x^2}{4} \qquad \textbf{Eq. 4-4}$$
$$\gamma_{xy} = a_8 + a_9 (x + y) - a_5 \frac{x^2}{4} + a_7 \frac{y^2}{4}$$

§ We observe that, if the terms of this equation are twice differentiated, they satisfy the general compatibility equation of strains, namely:

$$\frac{\partial^2 \varepsilon_x}{\partial y^2} + \frac{\partial^2 \varepsilon_y}{\partial x^2} = \frac{\partial^2 \gamma_{xy}}{\partial x\, \partial y} \qquad \textbf{Eq. 4-5}$$

§ The constants a_4, a_6 and a_8 are the terms corresponding to state of constant strain that ensures the convergence of the solution with mesh refinement. The constants a_5, a_7 and a_9 are the terms corresponding to the strain behavior.

§ To get the second part of the displacement fields for straining mode (U_2, V_2), we first integrate the fist two equations as follows.

$$U_2 = a_4 x + a_5\, x\, y + a_9 \frac{x\, y^2}{4} + f(y)$$
$$V_2 = a_6 y + a_7\, x\, y - a_9 \frac{y\, x^2}{4} + f(x) \qquad \textbf{Eq. 4-6}$$

§ To get the functions f(x) and f(y), we substitute their derivatives in the third equation then separate the resulting expressions for x and y respectively as follows:

$$\gamma_{xy} = \frac{\partial U_2}{\partial y} + \frac{\partial V_2}{\partial x}$$
$$a_8 + a_9 (x + y) - a_5 \frac{x^2}{4} + a_7 \frac{y^2}{4} = a_5 x + f'(x) + a_7 y + f'(y) \qquad \textbf{Eq. 4-7}$$

$$f(x) = \int [\frac{a_8}{2} + a_9 x - a_5 \frac{x^2}{4} - a_5 x]\, dx \ = \ a_8 \frac{x}{2} + a_5 (-\frac{x^3}{12} - \frac{x^2}{2}) + a_9 \frac{x^2}{2}$$
$$f(y) = \int [\frac{a_8}{2} + a_9 y + a_7 \frac{y^2}{4} - a_7 y] dy \ = \ a_8 \frac{y}{2} + a_7 (\frac{y^3}{12} - \frac{y^2}{2}) + a_9 \frac{y^2}{2} \qquad \textbf{Eq. 4-8}$$

§ The in-plane rotation can be calculated using the relation:

$$\varphi = \frac{1}{2}\left(\frac{\partial V}{\partial x} - \frac{\partial U}{\partial y}\right) \qquad \textbf{Eq. 4-9}$$

- Now, f(x) and f(y) are substituted in U_2, V_2. By adding the expressions for U_1 and V_1 to U_2 and V_2 then calculating the in-plane rotation, the complete expressions for the displacement fields are obtained as:

$$\begin{aligned}
U &= a_1 - a_3 y + a_4 x + a_5\, x\, y + a_7\left(\frac{y^3}{12} - \frac{y^2}{2}\right) + a_8 \frac{y}{2} + a_9\left(\frac{y^2}{2} + \frac{xy^2}{4}\right) \\
V &= a_2 + a_3\, x + a_5\left(-\frac{x^2}{2} - \frac{x^3}{12}\right) + a_6 y + a_7 xy + a_8 \frac{x}{2} + a_9\left(\frac{x^2}{2} - \frac{x^2 y}{4}\right) \qquad \textbf{Eq. 4-10} \\
\Phi &= a_3 + a_5\left(-x - \frac{x^2}{8}\right) + a_7\left(y - \frac{y^2}{8}\right) + a_9 (x - y - xy)/2
\end{aligned}$$

- It is noted that we obtained quadratic and cubic terms (x^2, x^3, y^2 & y^3) without increasing the number of nodes beyond the three corner nodes. This is not achieved in the known constant strain triangular element (CST). It is expected that this increase in the degree of the polynomials will result in more accurate solutions using this element; as will be shown in the subsequent sections.
- Having obtained the displacement fields, the stiffness matrix of the triangular element can be evaluated using the general expression

$$\left[K^e\right] = \left[C^{-1}\right]^T . \int [B]^T . [D] . [B] . d(vol) . \left[C^{-1}\right] \qquad \textbf{Eq. 4-11}$$

where, the transformation matrix [C] is calculated as

$$[C] = \begin{bmatrix} U_1 \,@\, (x_1, y_1) \\ V_1 \,@\, (x_1, y_1) \\ \Phi_1 \,@\, (x_1, y_1) \\ U_2 \,@\, (x_2, y_2) \\ V_2 \,@\, (x_2, y_2) \\ \Phi_2 \,@\, (x_2, y_2) \\ U_3 \,@\, (x_3, y_3) \\ V_3 \,@\, (x_3, y_3) \\ \Phi_3 \,@\, (x_3, y_3) \end{bmatrix}$$

the strain matrix [B] for this element is

$$[B]=\begin{bmatrix} 0 & 0 & 0 & 1 & y & 0 & 0 & 0 & \frac{y^2}{2} \\ 0 & 0 & 0 & 0 & 0 & 1 & x & 0 & -\frac{x^2}{2} \\ 0 & 0 & 0 & 0 & -\frac{x^2}{4} & 0 & \frac{y^2}{4} & 1 & x+y \end{bmatrix}$$

and [D] is the rigidity matrix given by

$$[D]=\frac{E}{(1-\nu^2)}\begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix} \text{for the state of plane stress}$$

and

$$[D]=\frac{E}{(1+\nu)(1-2\nu)}\begin{bmatrix} 1-\nu & \nu & 0 \\ \nu & 1-\nu & 0 \\ 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix} \text{for the state of plane strain.}$$

In the subsequent sections, this element will be called Strain Based Triangular Element with In-Plane Rotation, (**SBTREIR**).

4.3 PROBLEMS CONSIDERD

The performance of the new strain based triangular element derived in the previous section is applied to solve a deep cantilever problem and a simply supported beam problem as detailed below.

4.4 DEEP CANTILEVER BEAM PROBLEM

The first problem is a deep cantilever beam loaded by a point load at the free end. The beam have length L=10m, height H=4m, and thickness t=0.0625m. The material properties: modulus of elasticity and Poisson's ratio are taken as E=100,000 KPa and ν=0.20 respectively. The point load at the free end of the beam is taken as P =100 KN. In order to achieve full fixity at the built-in end of the cantilever beam, all the nodes occurring at that end are assumed to be restrained in the x and y directions as well as the in-plane rotation. Dimension and locations of the investigated points within the cantilever beam are shown in Figure 4-2 below.

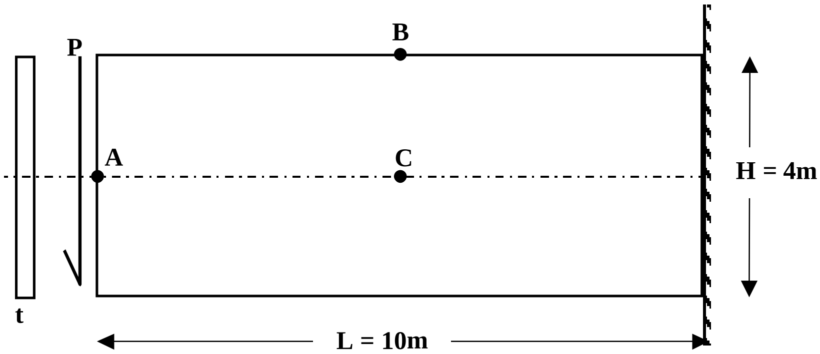


Figure 4-2: Dimensions and Locations of Considered Points for the Deep Cantilever Beam

4.4.1 Used Mesh Size

Several mesh sizes were used in the solution of the problem with increasing the total number of triangular elements. The adopted aspect ratio is 1:1 in almost all cases. The following table shows the number of elements, number of nodes and the aspect ratio of each mesh size.

Table 4-1: Mesh Size and Aspect Ratio of the Deep Cantilever Beam Using Triangular Elements

Elements in Short Side (L=4m)		Elements in Long Side (L=10m)		Aspect Ratio	Mesh Size	Total no. of Triangular Elements	Total no. of Nodes
No.	Dimension (m)	No.	Dimension (m)				
2	2.000	5	2.000	1 : 1	2x5	20	18
4	1.000	10	1.000	1 : 1	4x10	80	55
5	0.800	12	0.833	1: 1.042	5x12	120	78
6	0.667	15	0.667	1 : 1	6x15	180	112
8	0.500	20	0.500	1 : 1	8x20	320	189

A sample mesh size is illustrated in the figure below.

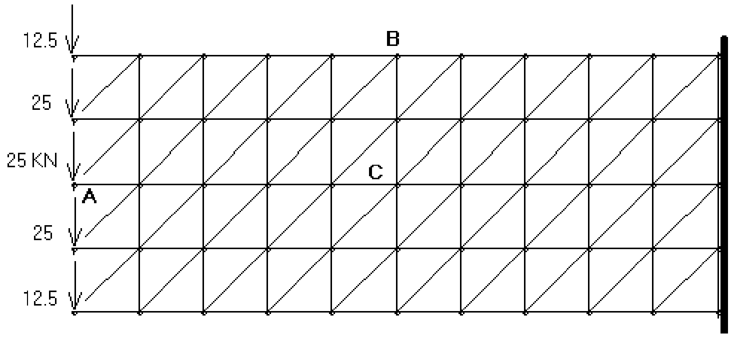


Figure 4-3: Sample Triangular Mesh of the Deep Cantilever Beam Problem

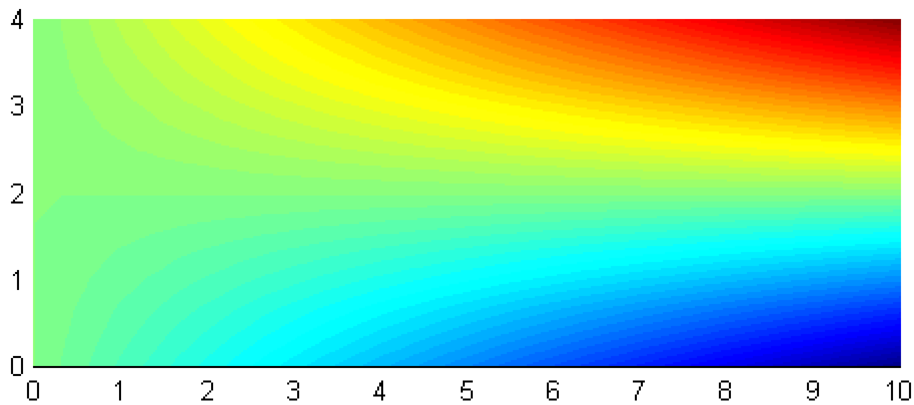
4.4.2 Analytical Solution

Based on the given geometry and material properties, the analytical solution for deflection and stresses at the specified points are calculated as follows:

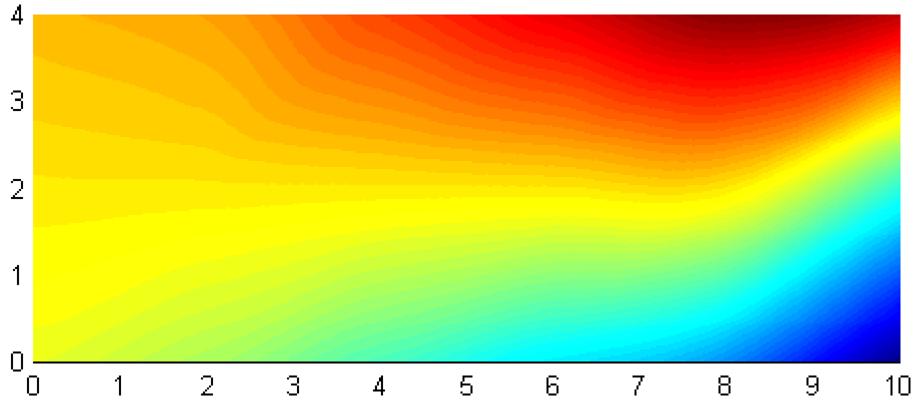
- Vertical deflection at the free end, point “A”= 1.105 mm.
- Bending stress at middle of upper face, point “B”= 3000 KPa.
- Shear stress at middle of centerline, point “C”= 600 KPa.
- In-Plane rotation at the free end, point “A”= 0.156 rad.

4.4.3 Convergence Results

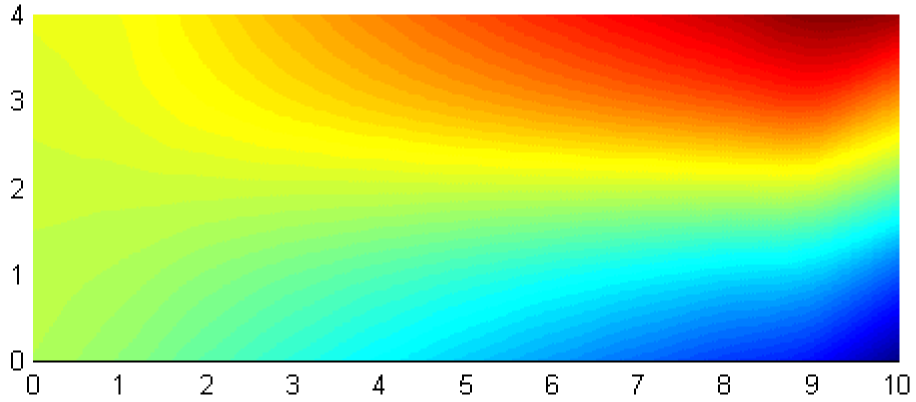
The problem was solved using the developed computer program (described in Chapter 3 and Annex A) for each of the mesh sizes listed in Table 4-1 above. Convergence of the overall pattern of bending stress in the cantilever beam is shown below for the analytical solution as well as the triangular elements (CST and SBTREIR) using each mesh size (Red: tension, Blue: compression).



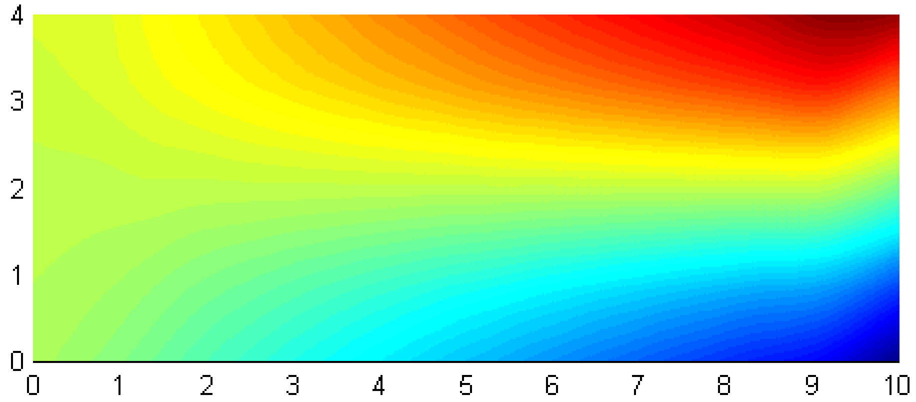
Bending Stress Pattern – Analytical Solution



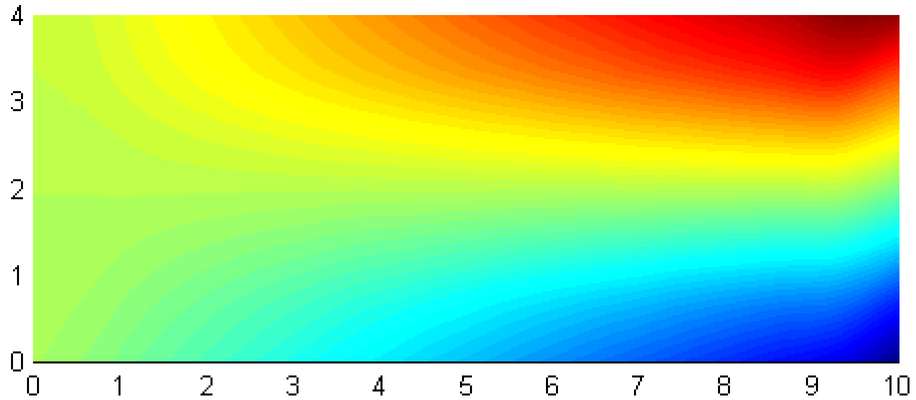
CST Mesh Size 2x5



CST Mesh Size 4x10



CST Mesh Size 5x12



CST Mesh Size 6x15

Figure 4-4: Bending Stress Pattern in the Deep Cantilever Beam Using CST

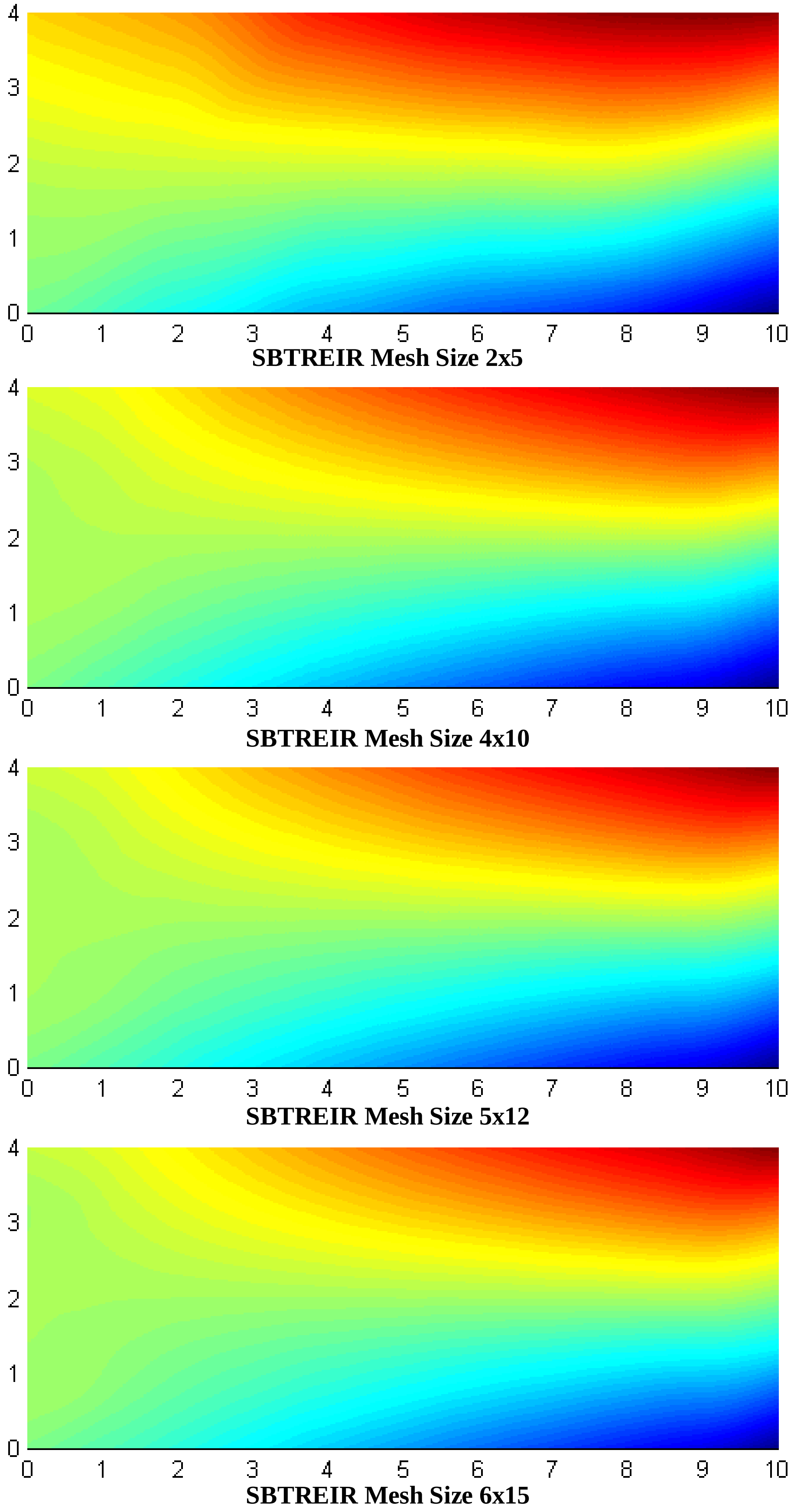


Figure 4-5: Bending Stress Pattern in the Deep Cantilever Beam Using SBTREIR

Table 4-2 shows a summary of the used meshes and results for vertical deflection, bending stress and shearing stress at the specified points within the deep cantilever beam as a percentage of the exact solutions for the CST and the new SBTREIR.

Table 4-2: Results of the Deep Cantilever Beam Problem Using Triangular Elements (CST and SBTREIR)

Mesh Size	No. of Elements	No. of Nodes	Vertical Deflection at A		Bending Stress at B		Shearing Stress at C	
			CST	SBTREIR	CST	SBTREIR	CST	SBTREIR
2 x 5	20	18	57.21%	71.12%	36.53%	70.18%	63.17%	63.11%
4 x 10	80	55	83.45%	89.44%	67.78%	87.48%	85.47%	97.79%
5 x 12*	120	78	88.50%	92.68%	75.00%	90.65%	86.83%	94.73%
6 x 15*	180	112	91.86%	95.56%	80.17%	93.18%	93.28%	99.18%
8 x 20	320	189	95.29%	98.25%	86.11%	96.00%	96.08%	99.59%
Analytical Solutions			**1.105**		**3000.00**		**600.00**	

* Note: In the case that any of the required points does not lie on a node, (as in the mesh sizes of 5x12 and 6x15 in this problem), results of bending stress and/or shearing stress are averaged from the nearest nodes to the location of the required point.

For the deep cantilever beam problem, we notice that the new triangular element, SBTREIR gives higher accuracy results than the CST for the cases of vertical deflection, bending stress and shear stress. For both elements, the convergence of the solution to the analytical value is ensured as more elements and nodes are used (mesh refinement).

The overall stress pattern converges to the analytical pattern in the solutions of the two triangular elements.

Figures 4-6 to 4-9 show graphical comparison between the results obtained by the SBTREIR element, the CST element and the analytical solutions.

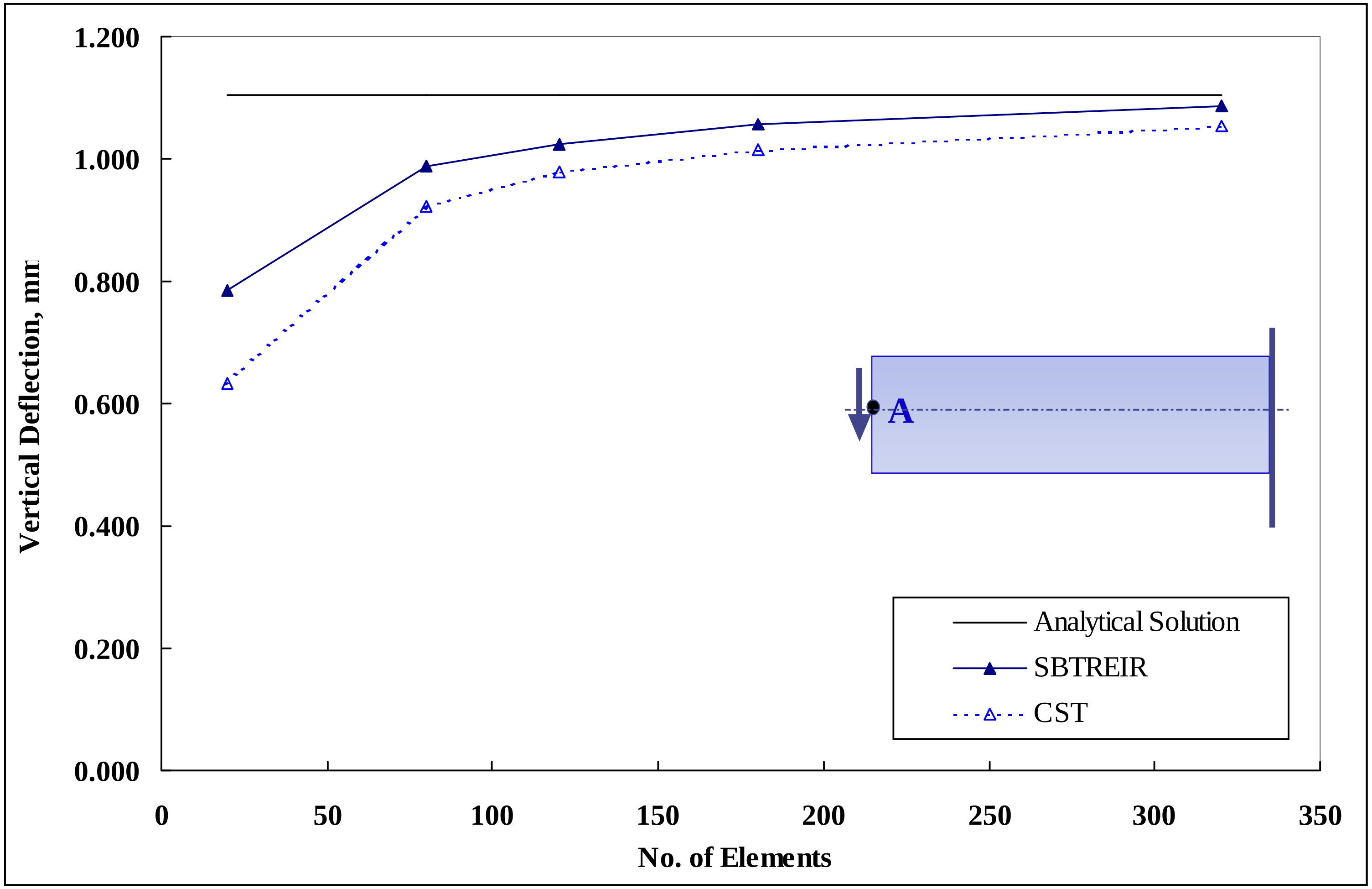


Figure 4-6: Vertical Deflection at "A", (mm) in the Deep Cantilever Beam Using Triangular Elements

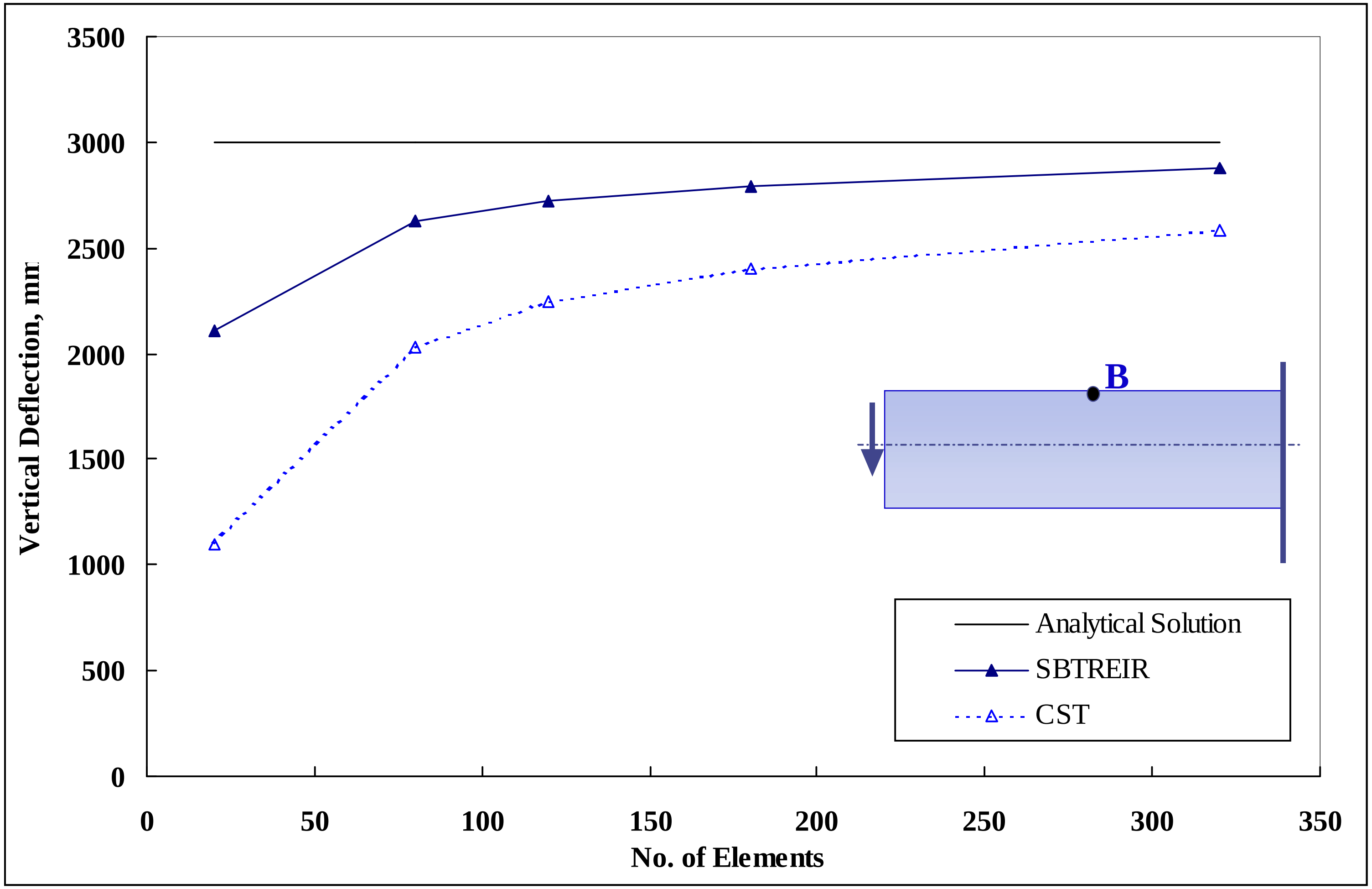


Figure 4-7: Bending stress at "B", (KPa) in the Deep Cantilever Beam Using Triangular Elements

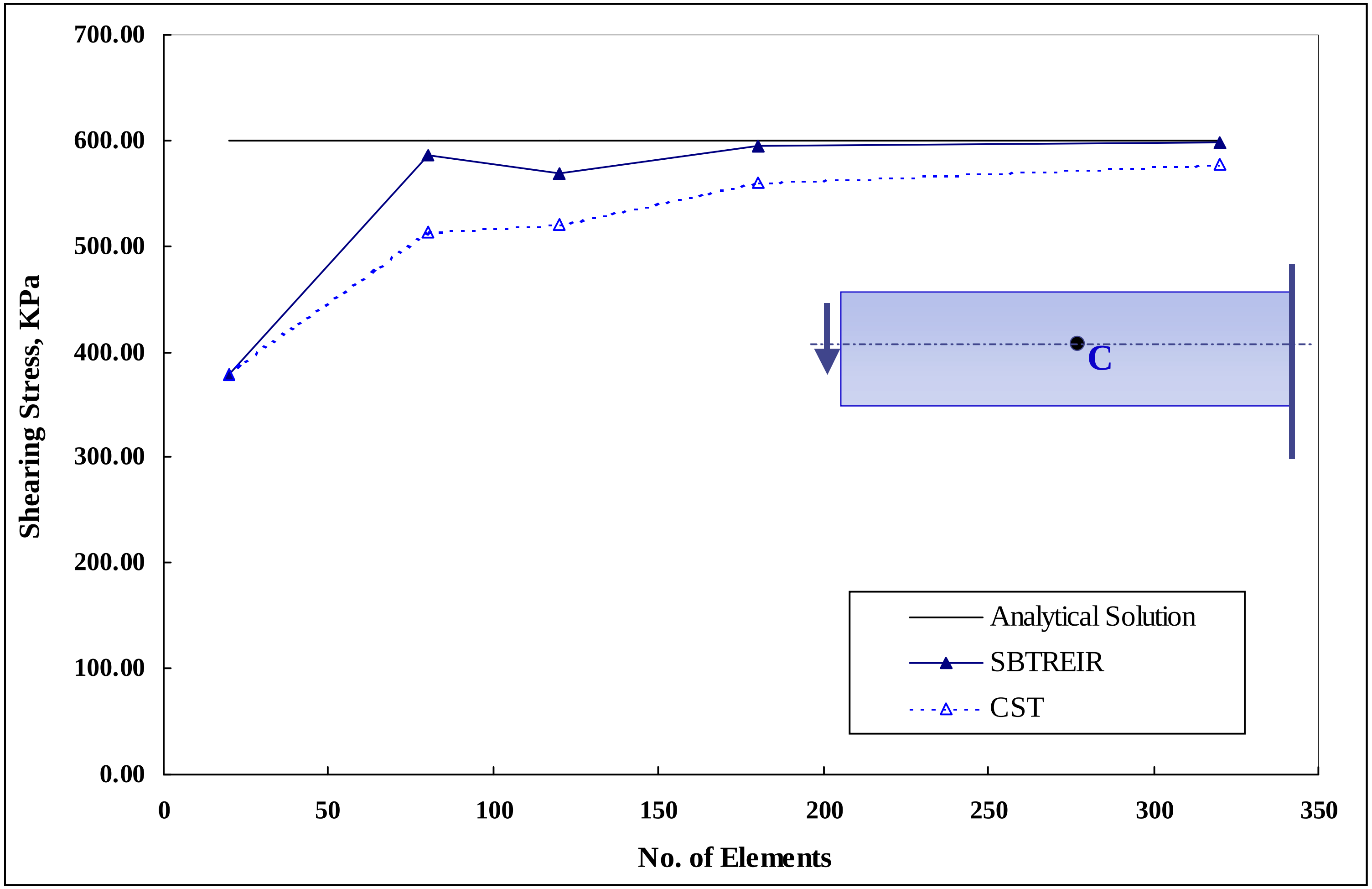


Figure 4-8: Shearing Stress at "C", (KPa) in the Deep Cantilever Beam Using Triangular Elements

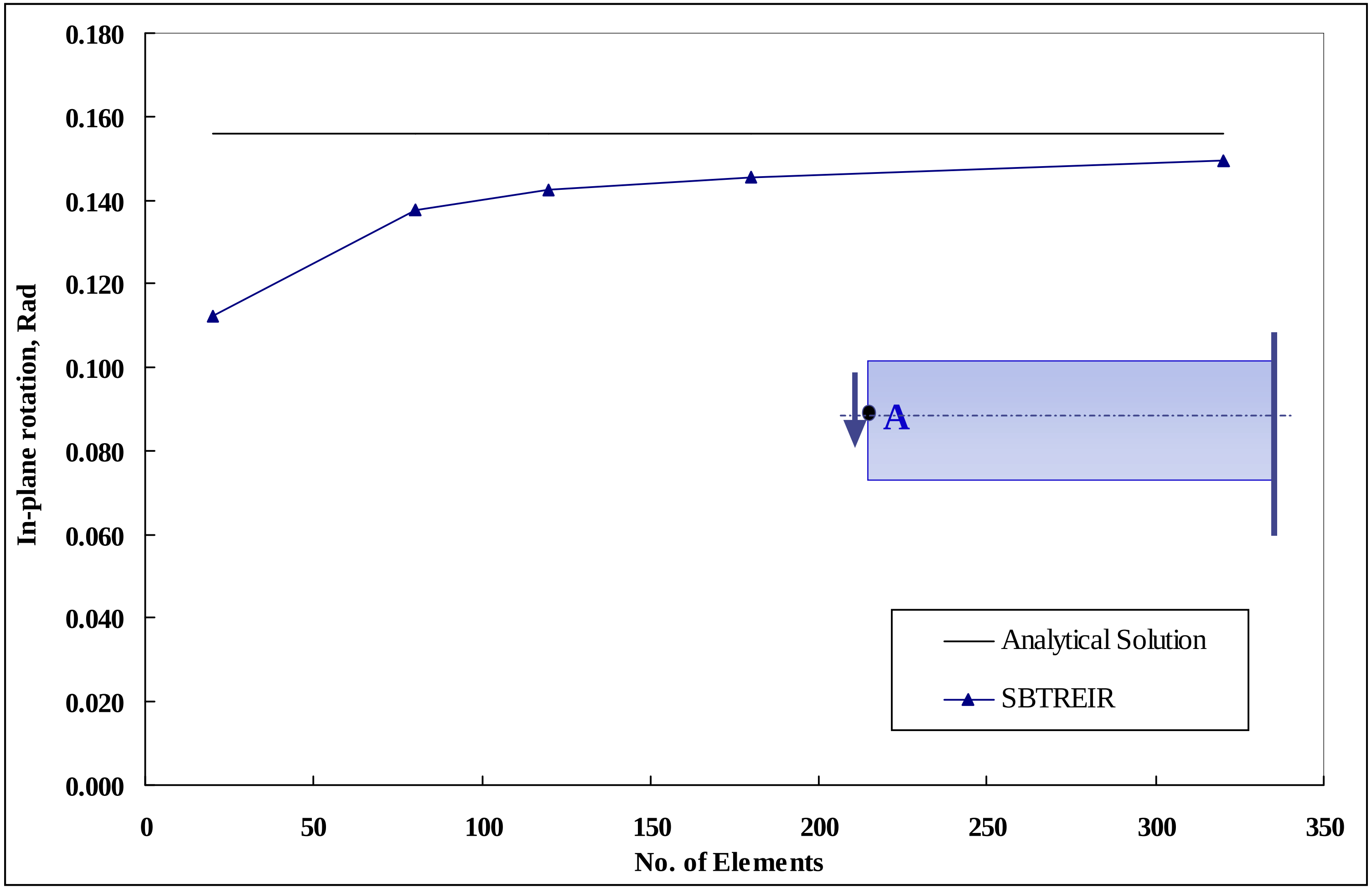


Figure 4-9: In-Plane Rotation at “A”, (Rad) in the Deep Cantilever Beam Using Triangular Elements

4.5 SIMPLY SUPPORTED BEAM PROBLEM

The second problem used to test the performance of the new element is that of a simply supported beam loaded by a point load at the middle of the upper surface of the beam. The beam have length, L=4m, height H=1m, and thickness t=0.5m. The material properties are taken as E=20,000 KPa and ν=0.20. The point load at the midspan end of the beam is taken as P=4.2 KN. The locations of the investigated points within the simply supported beam are shown in Figure 4-7 below.

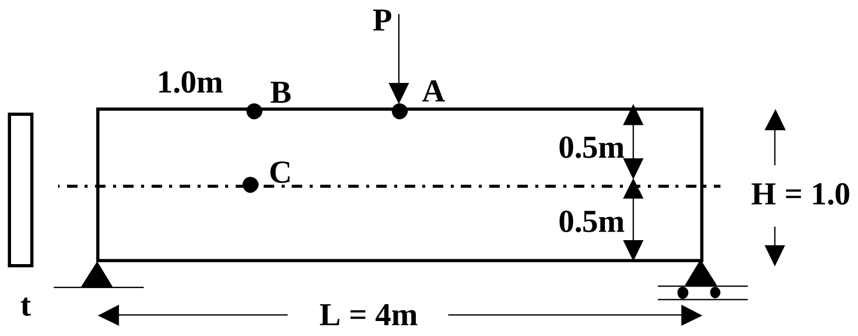


Figure 4-10: Dimensions and Considered Points for the Simply Supported Beam

4.5.1 Used Mesh Size

The following table shows the number of elements, number of nodes and the aspect ratio of each mesh size.

Table 4-3: Mesh Size and Aspect Ratio of the Simply Supported Beam Using Triangular Elements

Elements in Short Side (L=1m)		Elements in Long Side (L=4m)		Aspect Ratio	Mesh Size	Total no. of Triangular Elements	Total no. of Nodes
No.	Dimension (m)	No.	Dimension (m)				
1	1.000	4	1.000	1 : 1	1 x 4	8	10
2	0.500	8	0.500	1 : 1	2 x 8	32	27
3	0.333	12	0.333	1 : 1	3 x 12	72	52
4	0.250	16	0.250	1 : 1	4 x 16	128	85
5	0.200	20	0.200	1 : 1	5 x 20	200	126

A sample mesh size is illustrated in the figure below.

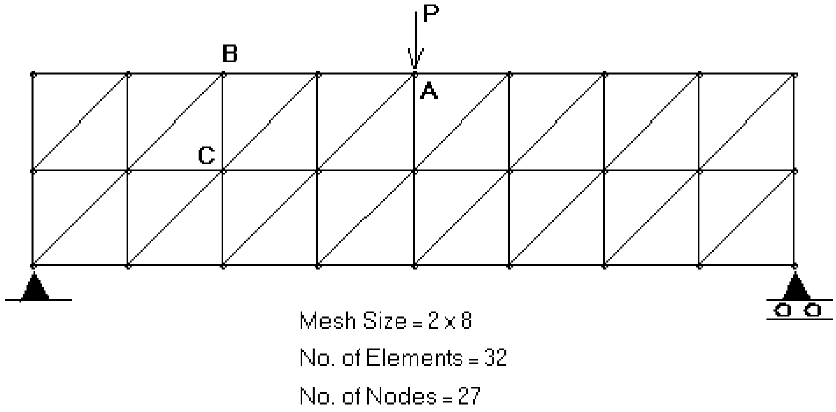


Figure 4-11: Sample Triangular Mesh of the Simply Supported Beam Problem

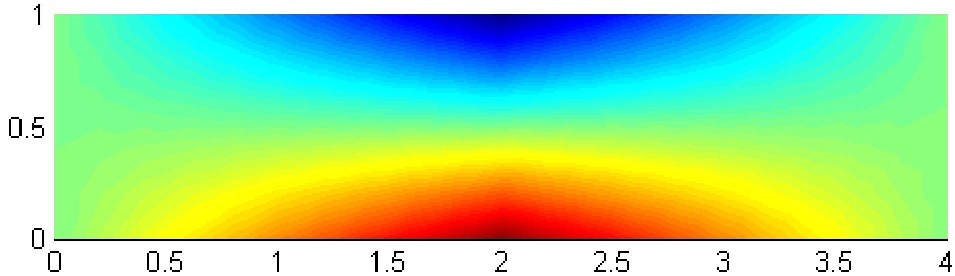
4.5.2 Analytical Solution

Based on the given geometry and material properties, the analytical solution for deflection and stresses at the specified points are calculated as follows:

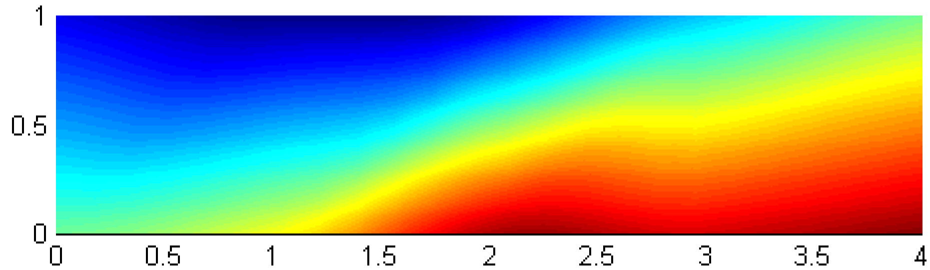
- Vertical deflection at point “A”= 6.72 mm.
- Bending Stress at point “B”= 25.2 KPa.
- Shear Stress at point “C”= 6.3 KPa.

4.5.3 Convergence Results

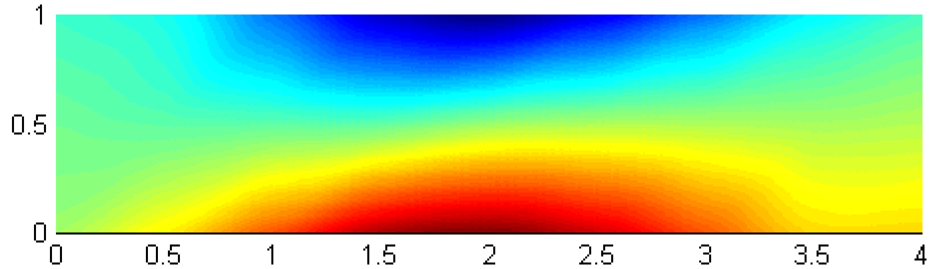
The problem was solved using the developed computer program (described in Chapter 3 and Annex A) for each of the mesh sizes listed in Table 4-3 above. Convergence of the overall pattern of bending stress in the simple beam is shown below for the analytical solution as well as the triangular elements (CST and SBTREIR) using each mesh size (Red: tension, Blue: compression).



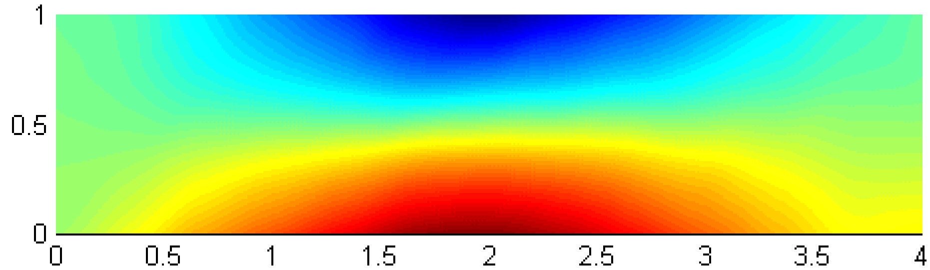
Bending Stress Pattern – Analytical Solution



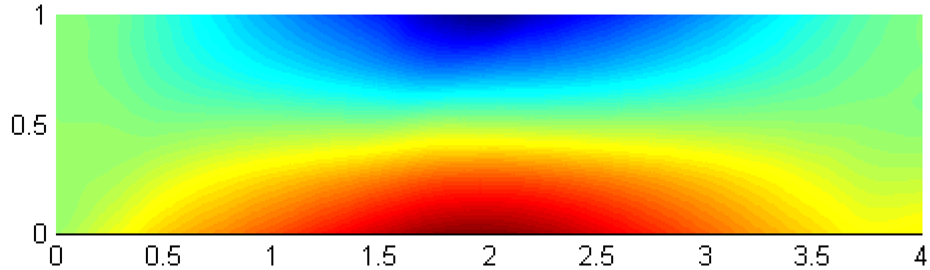
CST Mesh Size 1x4



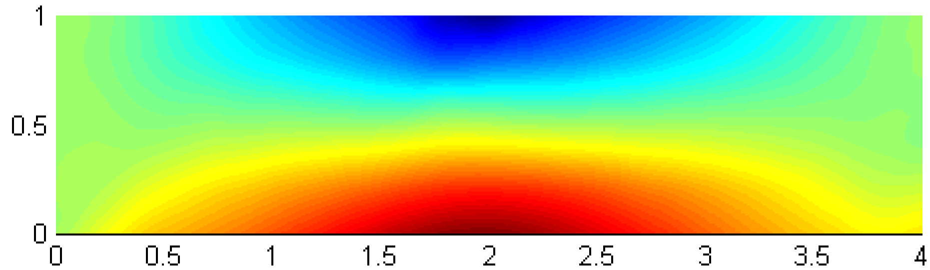
CST Mesh Size 2x8



CST Mesh Size 3x12

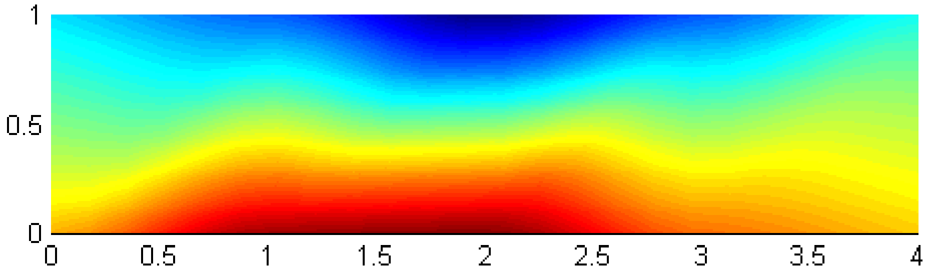


CST Mesh Size 4x16

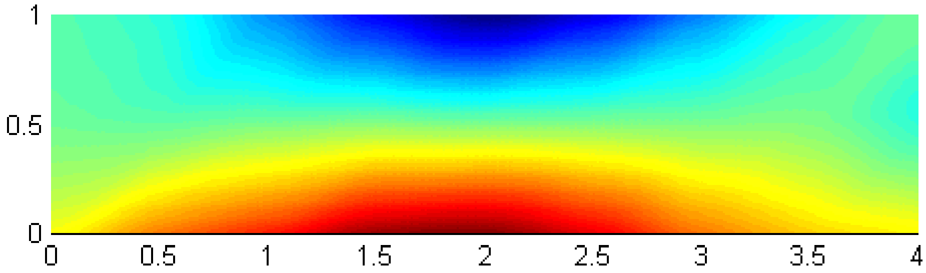


CST Mesh Size 5x20

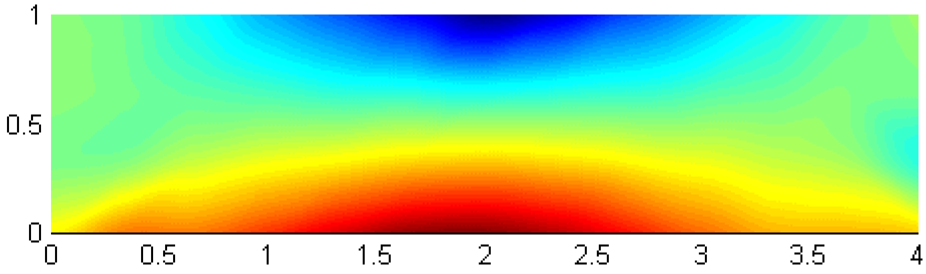
Figure 4-12: Bending Stress Pattern in the Simply Supported Beam Using CST



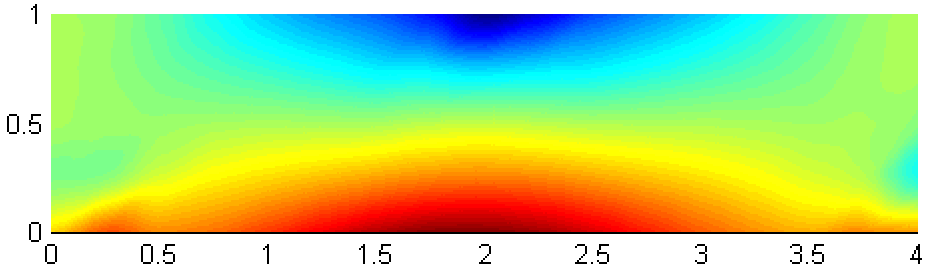
SBTREIR Mesh Size 1x4



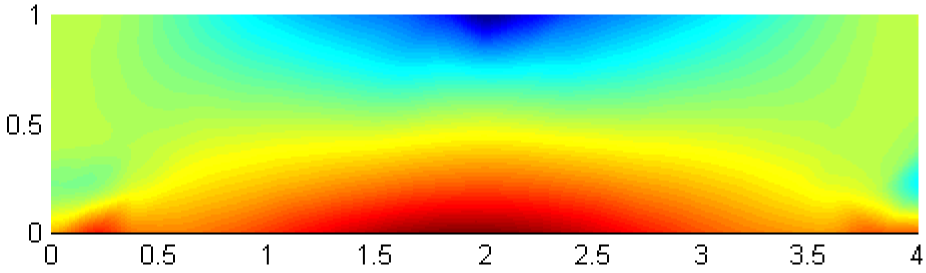
SBTREIR Mesh Size 2x8



SBTREIR Mesh Size 3x12



SBTREIR Mesh Size 4x16



SBTREIR Mesh Size 5x20

Figure 4-13: Bending Stress Pattern in the Simply Supported Beam Using SBTREIR

Table 4-4 shows a summary of the used meshes and results for vertical deflection, bending stress and shearing stress at the specified points within the simply supported beam as a percentage of the exact solutions for the CST and the new SBTREIR.

Table 4-4: Results of the Simply Supported Beam Problem Using Triangular Elements (CST and SBTREIR)

Mesh Size	No. of Elements	No. of Nodes	Vertical Deflection at A		Bending Stress at B		Shearing Stress at C	
			CST	SBTREIR	CST	SBTREIR	CST	SBTREIR
1 x 4	8	10	31.82%	50.43%	12.74%	78.37%	63.02%	65.16%
2 x 8	32	27	48.33%	62.62%	35.63%	87.38%	62.06%	81.76%
3 x 12*	72	52	57.99%	69.27%	53.49%	91.55%	69.52%	87.89%
4 x 16	128	85	64.08%	73.74%	65.04%	93.81%	86.35%	97.44%
5 x 20*	200	126	68.24%	77.02%	72.34%	95.28%	87.30%	98.54%
Analytical Solutions			**6.720**		**25.20**		**6.3**	

* Note: In the case that any of the required points does not lie on a node, (like the mesh sizes of 3x12 and 5x20 in this problem), results of bending stress and/or shearing stress are averaged from the nearest nodes to the location of the required point.

Again, for this problem, it is shown that the new triangular element gives higher accuracy results than the CST for the cases of vertical deflection, bending stress and shear stress.

The overall stress pattern converges to the analytical pattern in the solutions of the two triangular elements.

Figures 4-14 to 4-16 show graphical comparison between the results obtained by each of the SBTREIR and the constant strain element CST and the analytical solutions.

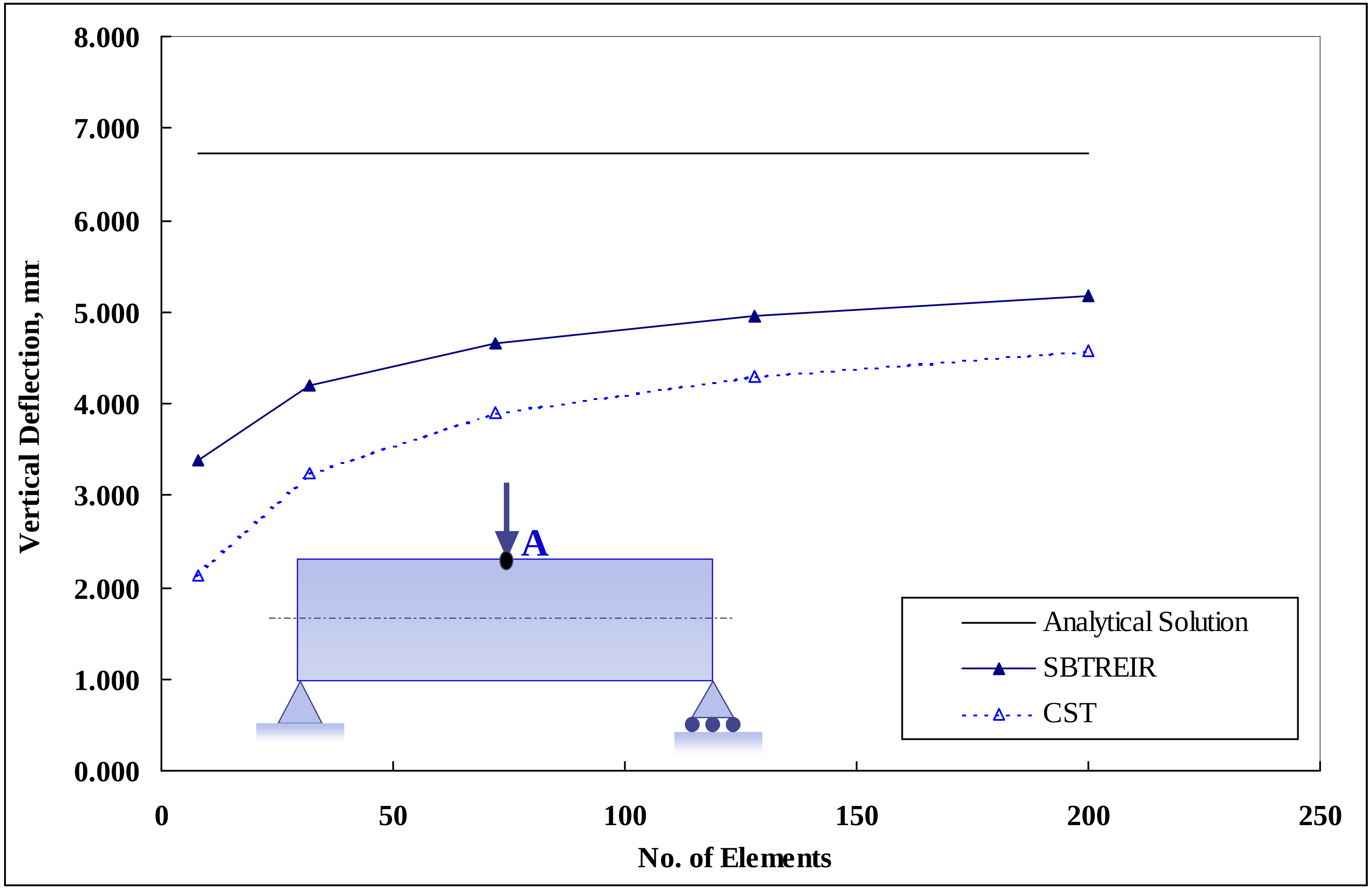


Figure 4-14: Vertical Deflection at "A", (mm) in the Simply Supported Beam Using Triangular Elements

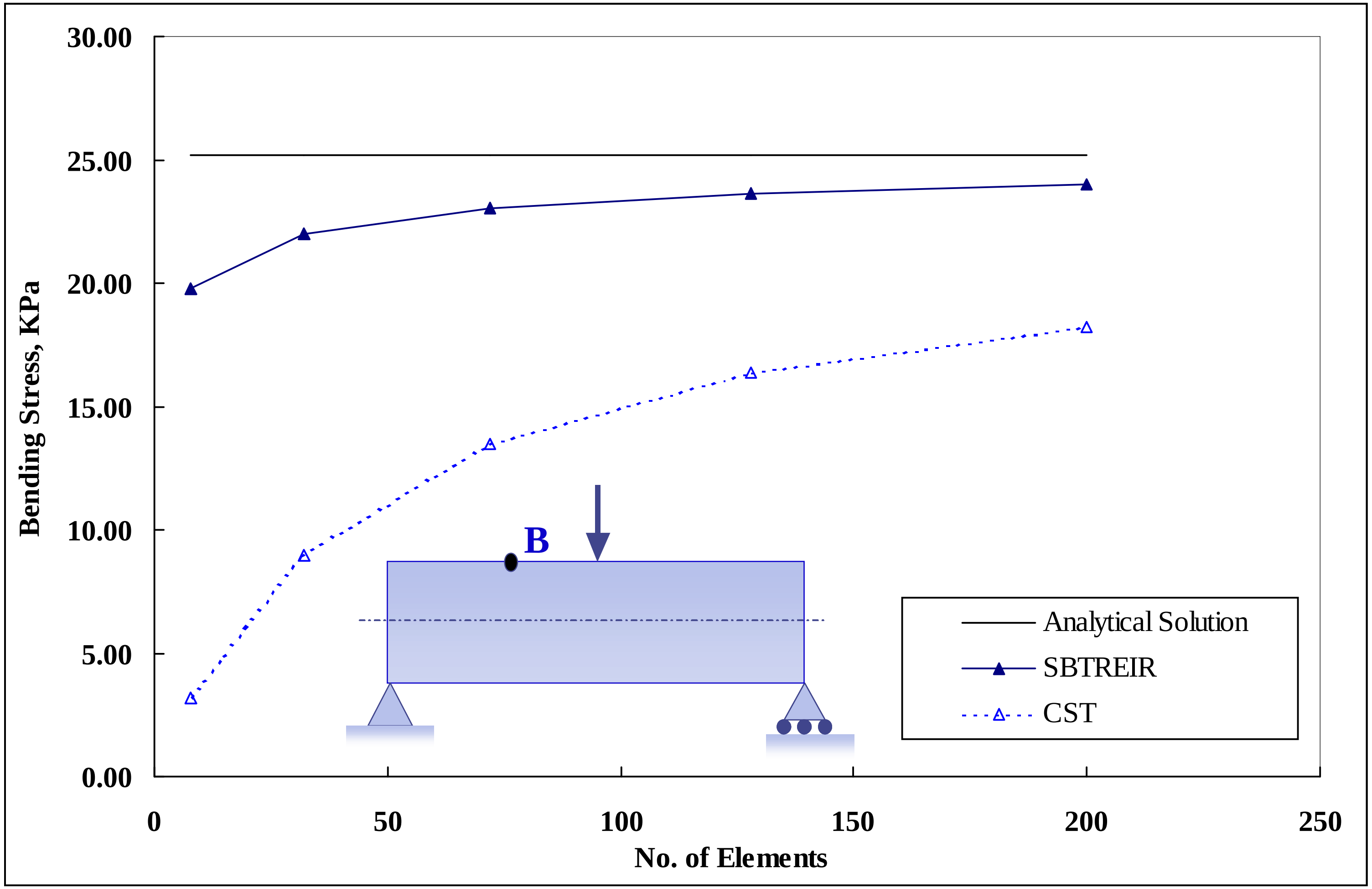


Figure 4-15: Bending Stress at "B", (KPa) in the Simply Supported Beam Using Triangular Elements

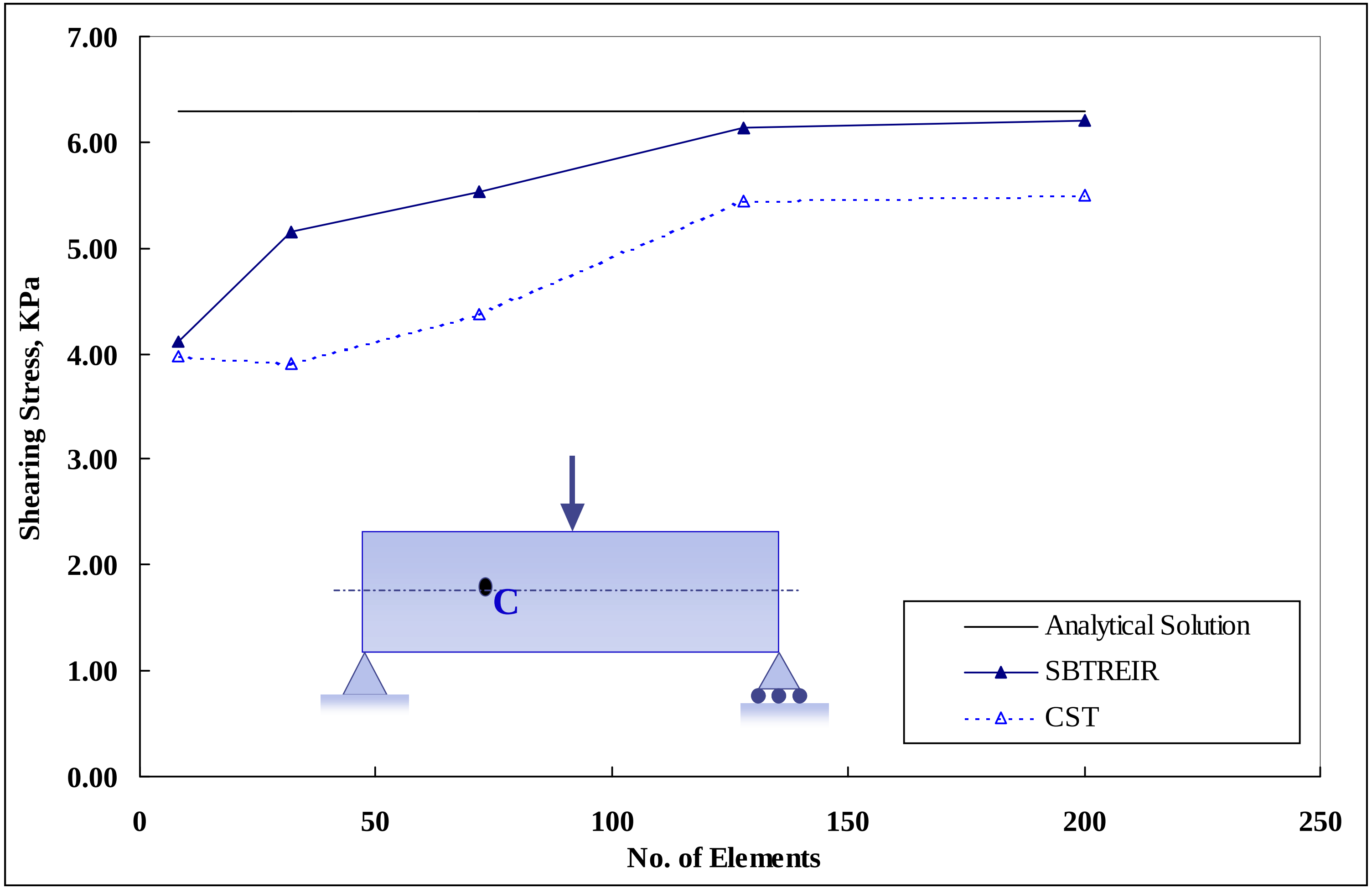


Figure 4-16: Shearing Stress at "C", (KPa) in the Simply Supported Beam Using Triangular Elements

5. CHAPTER FIVE: DEVELOPMENT OF NEW STRAIN-BASED RECTANGULAR ELEMENT

5.1 INTRODUCTION

In this chapter, the strain-based approach is used to investigate the ability of deriving displacement fields for a new strain-based rectangular element with the inclusion of the third degree of freedom at each node, which is the in-plane rotation, (also called drilling degree of freedom).

The performance of the new rectangular element is investigated by applying it to the solution of two of the common plane elasticity problems. These problems include: the problem of a plane deep cantilever beam fixed at one end and loaded by a point load at the free end and the problem of a simply supported beam loaded at the mid-span by a point load.

The results obtained by the developed rectangular strain based element are compared to those given by the well-known Bilinear Rectangular Element, (BRE) and the analytical values for deflection and stresses as detailed below.

5.2 DERIVATION OF DISPLACEMENT FIELDS FOR NEW RECTANGULAR ELEMENT WITH IN-PLANE ROTATION

The following outlines the assumptions and steps to derive the new strain-based rectangular element.

- The new rectangular element has four corner nodes with three degrees of freedom at each node as shown in the figure below.
- The displacement fields are required to satisfy the requirement of strain-free rigid body mode of displacement and straining of the element.

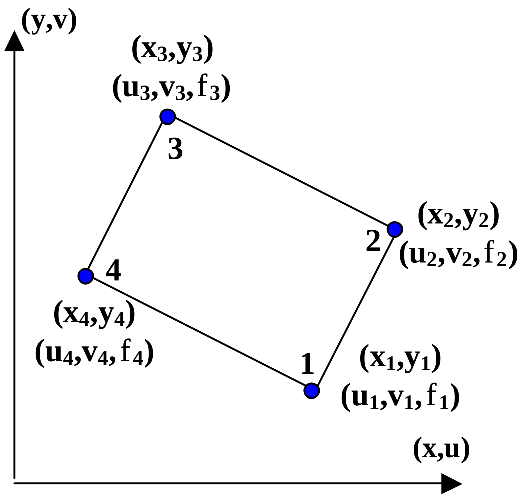


Figure 5-1: Coordinates and Node Numbering for the Rectangular Element with 3-DOF per node

§ To get the first part of the displacement fields for rigid-body motion (U_1 and V_1), we begin by writing the strain/displacement relationships for plane elasticity and make them equal to zero. These relationships are given by:

$$\varepsilon_x = \frac{\partial U}{\partial x} \qquad \varepsilon_y = \frac{\partial V}{\partial y} \qquad \gamma_{xy} = \frac{\partial U}{\partial y} + \frac{\partial V}{\partial x} \qquad \textbf{Eq. 5-1}$$

where

U and V : are the displacements in the x and y directions respectively.

ε_x and ε_y : are the direct axial strains in the x and y directions respectively.

γ_{xy} : is the shear strain.

Next, these stains are set equal to zero and then integrated:

$$\varepsilon_x = \frac{\partial U_1}{\partial x} = 0 \quad ===> U_1 = a_1 + f_1(y)$$

$$\varepsilon_y = \frac{\partial V_1}{\partial y} = 0 \quad ===> V_1 = a_2 + f_2(x)$$

$$\gamma_{xy} = \frac{\partial U_1}{\partial y} + \frac{\partial V_1}{\partial x} = 0 \quad ===> f_1'(y) + f_2'(x) = 0$$

where $f_1'(y)$ and $f_2'(x)$ are constants that can be taken as

$f_1'(y) = -a$ and $f_2'(x) = a_3$

hence,

$f_1(y) = -a_3\, y$ and $f_2(x) = a_3\, x$

thus,

$$U_1 = a_1 - a_3 y$$
$$V_1 = a_2 + a_3 x \qquad \textbf{Eq. 5-2}$$

where

U_1 and V_1 : are the displacements in the x and y directions respectively corresponding to strain-free rigid body mode of displacement.

a_1 and a_2 : represent the translations of the element in the x and y directions respectively.

a_3 : represent the rigid-body in plane rotation of the element.

- § Depending on the number of nodes and the number of degrees of freedom per node in the considered element, it is generally essential that the total number of degrees of freedom in the element (and hence the number of constants used in defining the displacement fields within the element) equals the number of nodes times the number of degrees of freedom per node.
- § The displacements within the element have to be defined by twelve constants (a_1 through a_{12}). Three constants have already been defined while the remaining nine have to be used to describe the deformation straining of the element.
- § As a first trial, these constants are arranged in the following manner:

$$\varepsilon_x = a_4 + a_5 x + a_6 y$$
$$\varepsilon_y = a_7 + a_8 x + a_9 y \qquad \textbf{Eq. 5-3}$$
$$\gamma_{xy} = a_{10} + a_{11} x + a_{12} y$$

- § This arrangement of strains leads to a singular displacement transformation matrix. Several other arrangements were tried to avoid this. A good arrangement that gives non-singular transformation matrix is found to be as follows:

$$\varepsilon_x = a_4 + a_5 y + (a_8 y^2 + a_9 x y^3)$$
$$\varepsilon_y = a_6 + a_7 x + (-a_8 x^2 - a_9 x^3 y) \qquad \textbf{Eq. 5-4}$$
$$\gamma_{xy} = a_{10} + a_{11} x + a_{12} y + (-a_5 \frac{x^2}{2} - a_7 \frac{y^2}{2})$$

§ We observe that, if the terms of this equation are twice differentiated, they satisfy the general compatibility equation of strains, namely:

$$\frac{\partial^2 \varepsilon_x}{\partial y^2} + \frac{\partial^2 \varepsilon_y}{\partial x^2} = \frac{\partial^2 \gamma_{xy}}{\partial x \, \partial y} \qquad \textbf{Eq. 5-5}$$

The constants a_4, a_6 and a_{10} are the terms corresponding to state of constant strain that ensures the convergence of the solution with mesh refinement. The constants a_5, a_7 and a_{11} are the terms corresponding to linear strain behavior within the element. The higher order bracketed terms are included to satisfy the compatibility equation.

The second part of the displacement fields (U_2, V_2) is obtained by following the same procedure as before; this gives:

$$U_2 = a_4 x + a_5 \, x\,y + a_8 \, x\,y^2 + a_9 \frac{x^2 \, y^3}{2} + f(y)$$
$$V_2 = a_6 y + a_7 \, x\,y - a_8 \, x^2 \, y - a_9 \frac{y^2 \, x^3}{2} + f(x) \qquad \textbf{Eq. 5-6}$$

$$\gamma_{xy} = \frac{\partial U_2}{\partial y} + \frac{\partial V_2}{\partial x}$$
$$a_{10} + a_{11} x + a_{12} y + (-a_5 \frac{x^2}{2} - a_7 \frac{y^2}{2}) = a_5 x + f'(y) + a_7 y + f'(x) \qquad \textbf{Eq. 5-7}$$

$$f(x) = \int [\frac{a_{10}}{2} + a_5(-x - \frac{x^2}{2}) + a_{11} x] \, dx \; = a_{10} \frac{x}{2} + a_5 (-\frac{x^2}{2} - \frac{x^3}{6}) + a_{11} \frac{x^2}{2}$$
$$f(y) = \int [\frac{a_{10}}{2} + a_7(-y - \frac{y^2}{2}) + a_{12} y] \, dx \; = a_{10} \frac{y}{2} + a_7 (-\frac{y^2}{2} - \frac{y^3}{6}) + a_{12} \frac{y^2}{2} \qquad \textbf{Eq. 5-8}$$

Now, f(x) and f(y) are substituted in U_2, V_2.

$$U_2 = a_4 x + a_5 xy + a_7 (-\frac{y^2}{2} - \frac{y^3}{6}) + a_8 xy^2 + a_9 \frac{x^2 y^3}{2} + a_{10} \frac{y}{2} + a_{12} \frac{y^2}{2}$$
$$V_2 = a_5 (-\frac{x^2}{2} - \frac{x^3}{6}) + a_6 y + a_7 xy - a_8 x^2 y - a_9 \frac{x^3 y^2}{2} + a_{10} \frac{x}{2} + a_{11} \frac{x^2}{2} \qquad \textbf{Eq. 5-9}$$

By adding the expressions for U_1 & V_1 and U_2 & V_2 then calculating the in-plane rotation, the complete expressions for the displacement fields are obtained as:

$$U = a_1 - a_3 y + a_4 x + a_5 xy + a_7(-\frac{y^2}{2} - \frac{y^3}{6}) + a_8 xy^2 + a_9 \frac{x^2 y^3}{2} + a_{10}\frac{y}{2} + a_{12}\frac{y^2}{2}$$
$$V = a_2 + a_3 x + a_5(-\frac{x^2}{2} - \frac{x^3}{6}) + a_6 y + a_7 xy - a_8 x^2 y - a_9 \frac{x^3 y^2}{2} + a_{10}\frac{x}{2} + a_{11}\frac{x^2}{2}$$
$$\Phi = a_3 + a_5(-x - \frac{x^2}{4}) + a_7(y + \frac{y^2}{4}) - 2a_8 xy - \frac{3}{2} a_9 x^2 y^2 + a_{11}\frac{x}{2} - a_{12}\frac{y}{2}$$

Eq. 5-10

- It is noted that we obtained quadratic and cubic terms (x^2, x^3, y^2 & y^3) without increasing the number of nodes beyond the four corner nodes. This is not achieved in the well-known bilinear rectangular element. It is expected that this increase in the degree of the polynomials will result in more accurate solutions using this element as will be shown in the subsequent sections.
- Having obtained the displacement fields, the stiffness matrix of the triangular element can be evaluated using the general expression

$$[K^e] = [C^{-1}]^T . \int [B]^T . [D] . [B] . d(vol) . [C^{-1}]$$ **Eq. 5-11**

where, the transformation matrix, [C] is calculated as

$$[C] = \begin{bmatrix} U_1 @ (x_1, y_1) \\ V_1 @ (x_1, y_1) \\ \Phi_1 @ (x_1, y_1) \\ --- \\ --- \\ --- \\ --- \\ U_4 @ (x_4, y_4) \\ V_4 @ (x_4, y_4) \\ \Phi_4 @ (x_4, y_4) \end{bmatrix}$$

the strain matrix [B] for this element is

$$[B] = \begin{bmatrix} 0 & 0 & 0 & 1 & y & 0 & 0 & y^2 & xy^3 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 & x & -x^2 & -x^3 y & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & -\frac{x^2}{2} & 0 & -\frac{y^2}{2} & 0 & 0 & 1 & x & y \end{bmatrix}$$

and [D] is the rigidity matrix given by

$$[D]=\frac{E}{(1-\nu^2)}\begin{bmatrix}1 & \nu & 0\\ \nu & 1 & 0\\ 0 & 0 & \frac{1-\nu}{2}\end{bmatrix}\text{ for the state of plane stress}$$

and

$$[D]=\frac{E}{(1+\nu)(1-2\nu)}\begin{bmatrix}1-\nu & \nu & 0\\ \nu & 1-\nu & 0\\ 0 & 0 & \frac{1-2\nu}{2}\end{bmatrix}\text{ for the state of plane strain.}$$

In the subsequent sections, this element will be called: Strain Based Rectangular Element with In-Plane Rotation, **(SBREIR)**.

5.3 <u>PROBLEMS CONSIDERD</u>

The performance of the new strain-based rectangular element derived in the previous section is applied to solve the same deep cantilever and simply supported beam problems as detailed below.

5.4 <u>DEEP CANTILEVER BEAM PROBLEM</u>

The same deep cantilever problem that was described in Section 4.4 was solved again using the new rectangular element as well as the existing bilinear rectangular element.

5.4.1 Used Mesh Size

Several mesh sizes were used in the solution of the problem with increasing the total number of rectangular elements. The adopted aspect ratio is 1:1 in almost all cases. The following table shows the number of elements and number of nodes and aspect ratio at each mesh size.

Table 5-1: Mesh Size and Aspect Ratio of the Deep Cantilever Beam Using Rectangular Elements

Elements in Short Side (L=4m)		Elements in Long Side (L=10m)		Aspect Ratio	Mesh Size	Total no. of Rectangular Elements	Total no. of Nodes
No.	Dimension (m)	No.	Dimension (m)				
2	2.000	5	2.000	1 : 1	2x5	10	18
4	1.000	10	1.000	1 : 1	4x10	40	55
5	0.800	12	0.833	1: 1.042	5x12	60	78
6	0.667	15	0.667	1 : 1	6x15	90	112
8	0.500	20	0.500	1 : 1	8x20	160	189
10	0.400	25	0.400	1 : 1	10x25	250	286

A sample mesh size is also illustrated in the figure below.

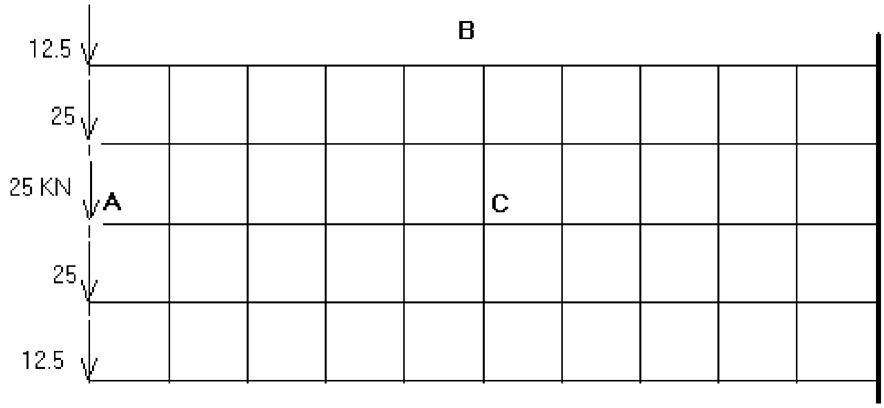
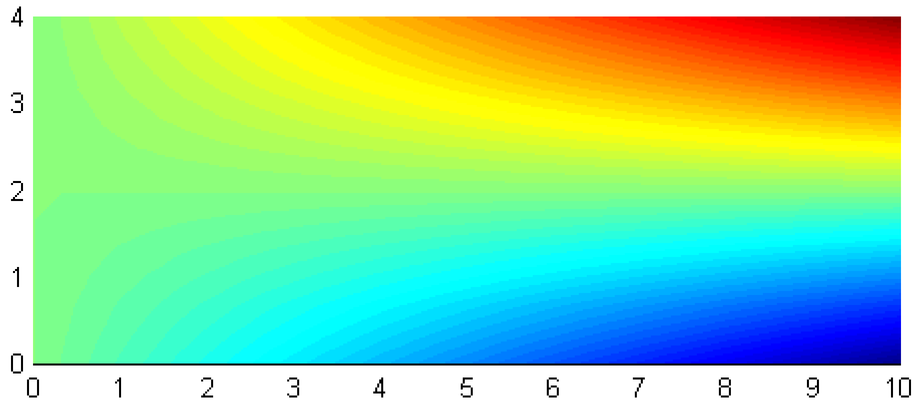


Figure 5-2: Sample Rectangular Mesh of the Deep Cantilever Beam Problem

5.4.2 Convergence Results

The problem was solved using the developed computer program (described in Chapter 3 and Annex A) for each of the mesh sizes listed in Table 5-1.

Convergence of the overall pattern of bending stress in the cantilever beam is shown below for the analytical solution as well as the rectangular elements (BRE and SBREIR) using each mesh size (Red: tension, Blue: compression).



Bending Stress Pattern – Analytical Solution (Repeated)

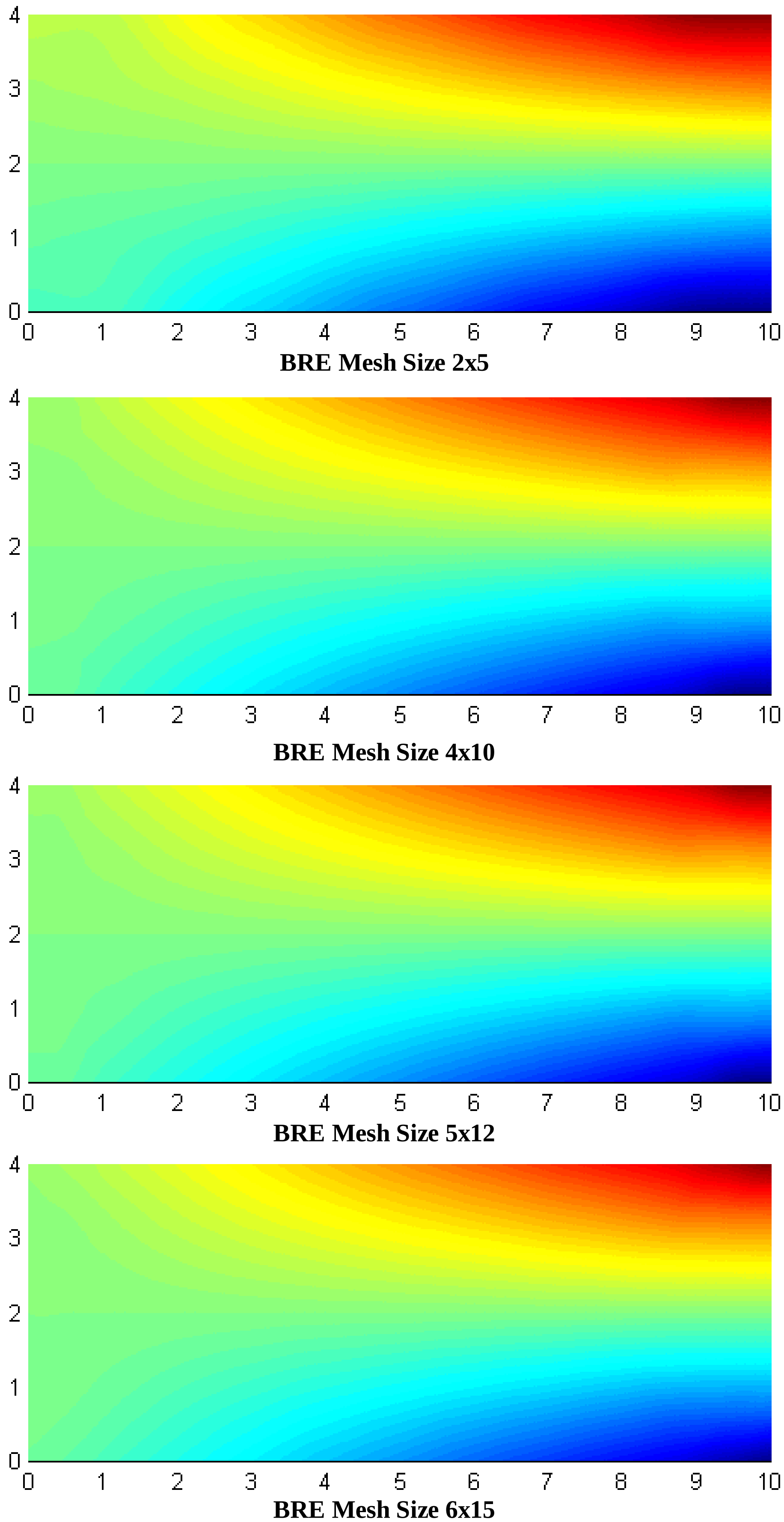
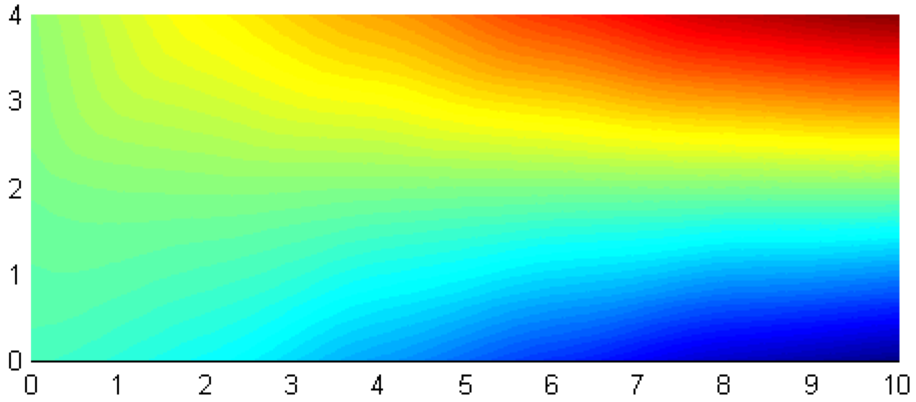
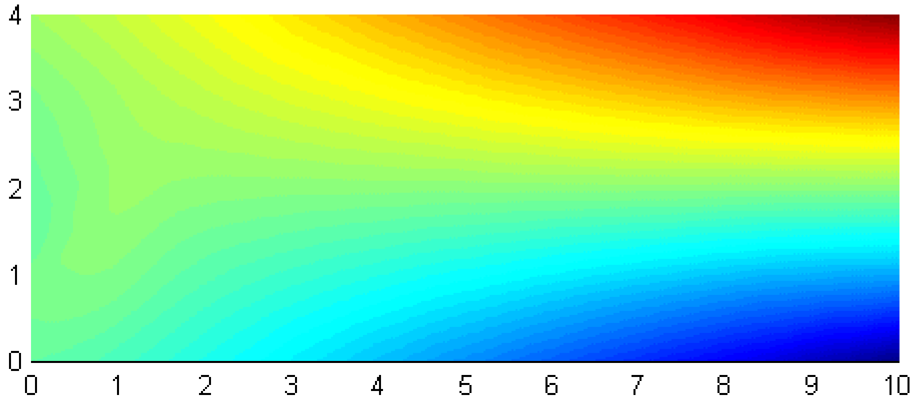


Figure 5-3: Bending Stress Pattern in the Deep Cantilever Beam Using BRE

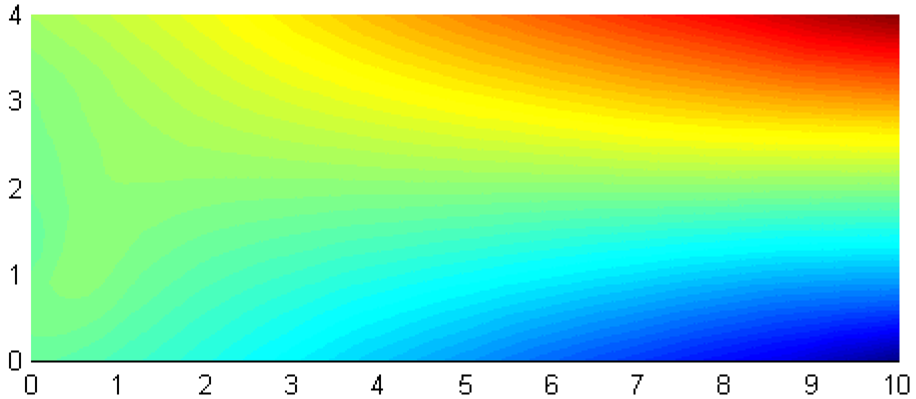


SBREIR Mesh Size 2x5

SBREIR Mesh Size 4x10



SBREIR Mesh Size 5x12



SBREIR Mesh Size 6x15

Figure 5-4: Bending Stress Pattern in the Deep Cantilever Beam Using SBREIR

Table 5-2 shows a summary of the used meshes and numerical results for vertical deflection, bending stress and shearing stress at the specified points within the deep cantilever beam as a percentage of the exact solutions for both the BRE and the new SBREIR elements.

Table 5-2: Results of the Deep Cantilever Beam Problem Using Rectangular Elements (BRE and SBREIR)

Mesh Size	No. of Elements	No. of Nodes	Vertical Deflection at A		Bending Stress at B		Shearing Stress at C	
			BRE	SBREIR	BRE	SBREIR	BRE	SBREIR
2 x 5	10	18	88.60%	96.47%	91.57%	100.07%	69.63%	112.31%
4 x 10	40	55	96.92%	99.00%	98.23%	100.19%	86.92%	96.66%
5 x 12*	60	78	97.92%	99.37%	98.88%	100.22%	94.49%	92.45%
6 x 15*	90	112	98.73%	99.55%	99.43%	100.07%	96.38%	95.81%
8 x 20	160	189	99.37%	99.82%	99.80%	100.10%	96.66%	98.35%
10 x 25*	250	286	99.64%	99.91%	99.96%	100.01%	98.69%	101.05%
Analytical Solutions			**1.105**		**3000.00**		**600.00**	

* Note: In the case that any of the required points does not lie on a node, (like the mesh sizes of 5x12, 6x15 and 10x25 in this problem), results of bending stress and/or shearing stress are averaged from the nearest nodes to the location of the required point.

For the deep cantilever problem, we notice that the new rectangular element gives higher accuracy results than the BRE for the case of vertical deflection and bending stress. For the bending stress, its results are very close but slightly more than the exact solution. For the shear stress, the results are almost the same as those obtained by the BRE. Both elements converge to the same values as the number of elements increases.

The overall stress pattern converges to the analytical pattern in the solutions of the two rectangular elements.

Figures 5-5 to 5-8 show graphical comparison between the results obtained by each of the SBREIR, the BRE and the analytical solutions.

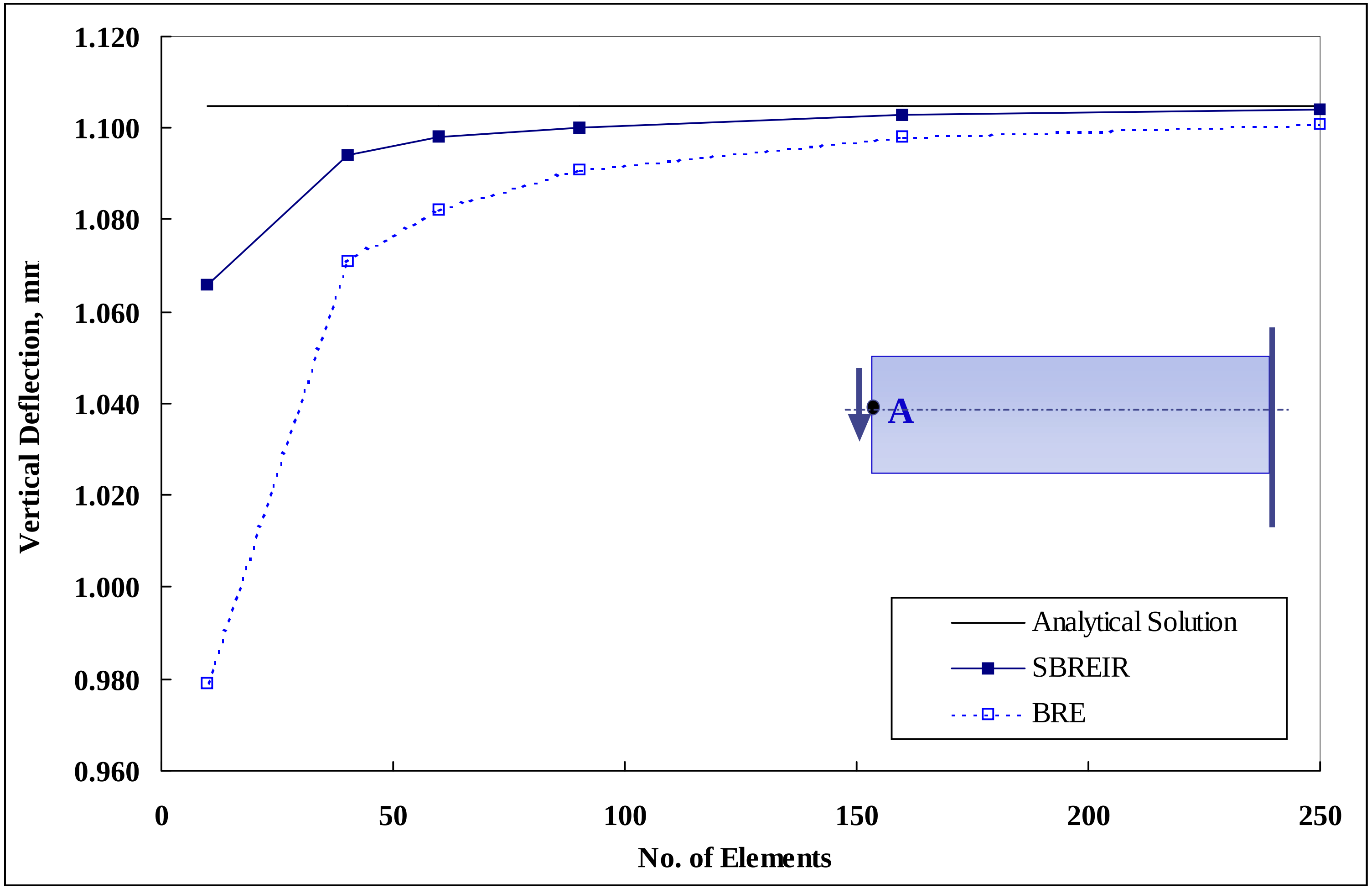


Figure 5-5: Vertical Deflection at “A”, (mm) in the Deep Cantilever Beam Using Rectangular Elements

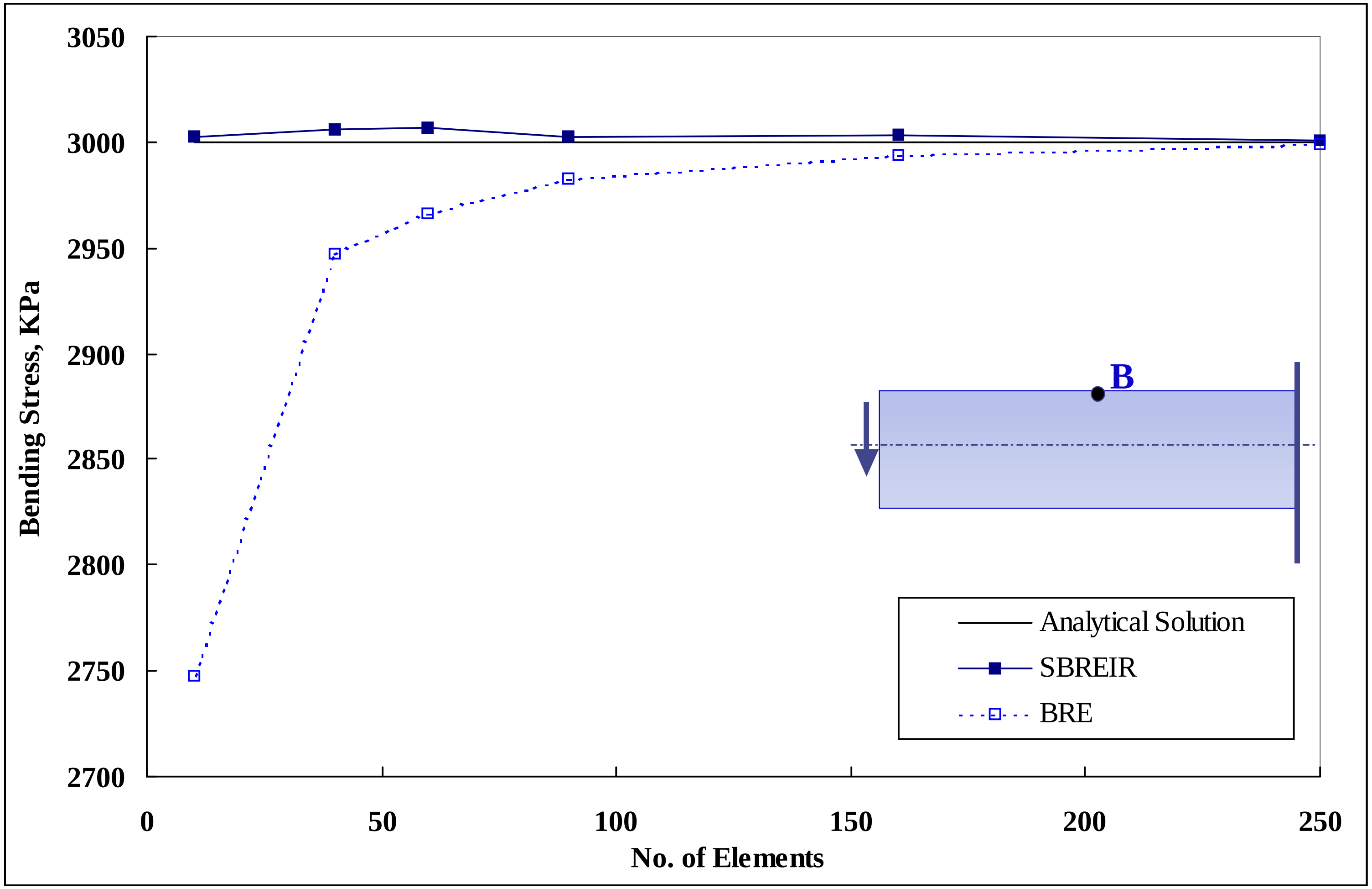


Figure 5-6: Bending Stress at "B", (KPa) in the Deep Cantilever Beam Using Rectangular Elements

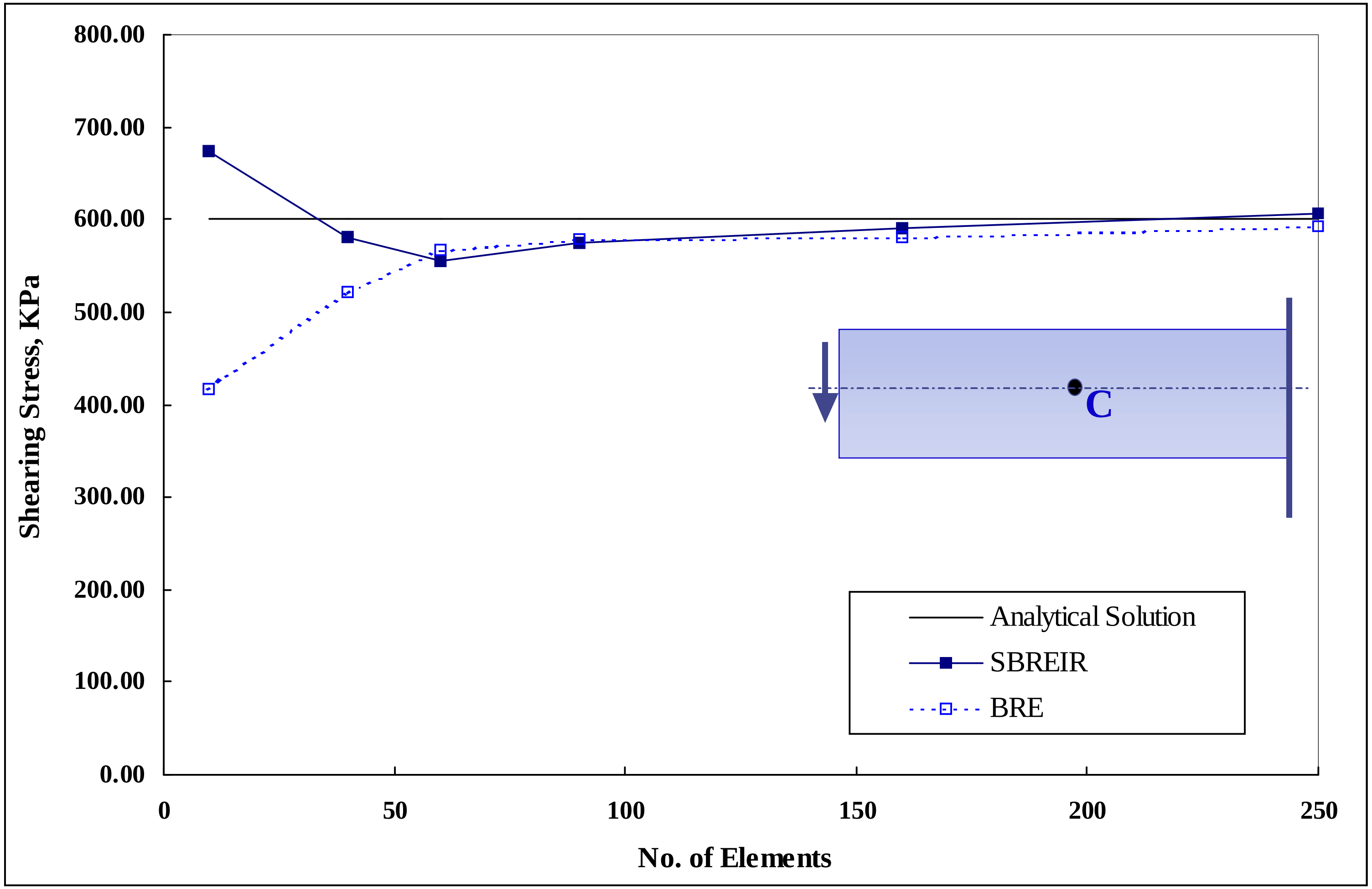


Figure 5-7: Shearing Stress at "C", (KPa) in the Deep Cantilever Beam Using Rectangular Elements

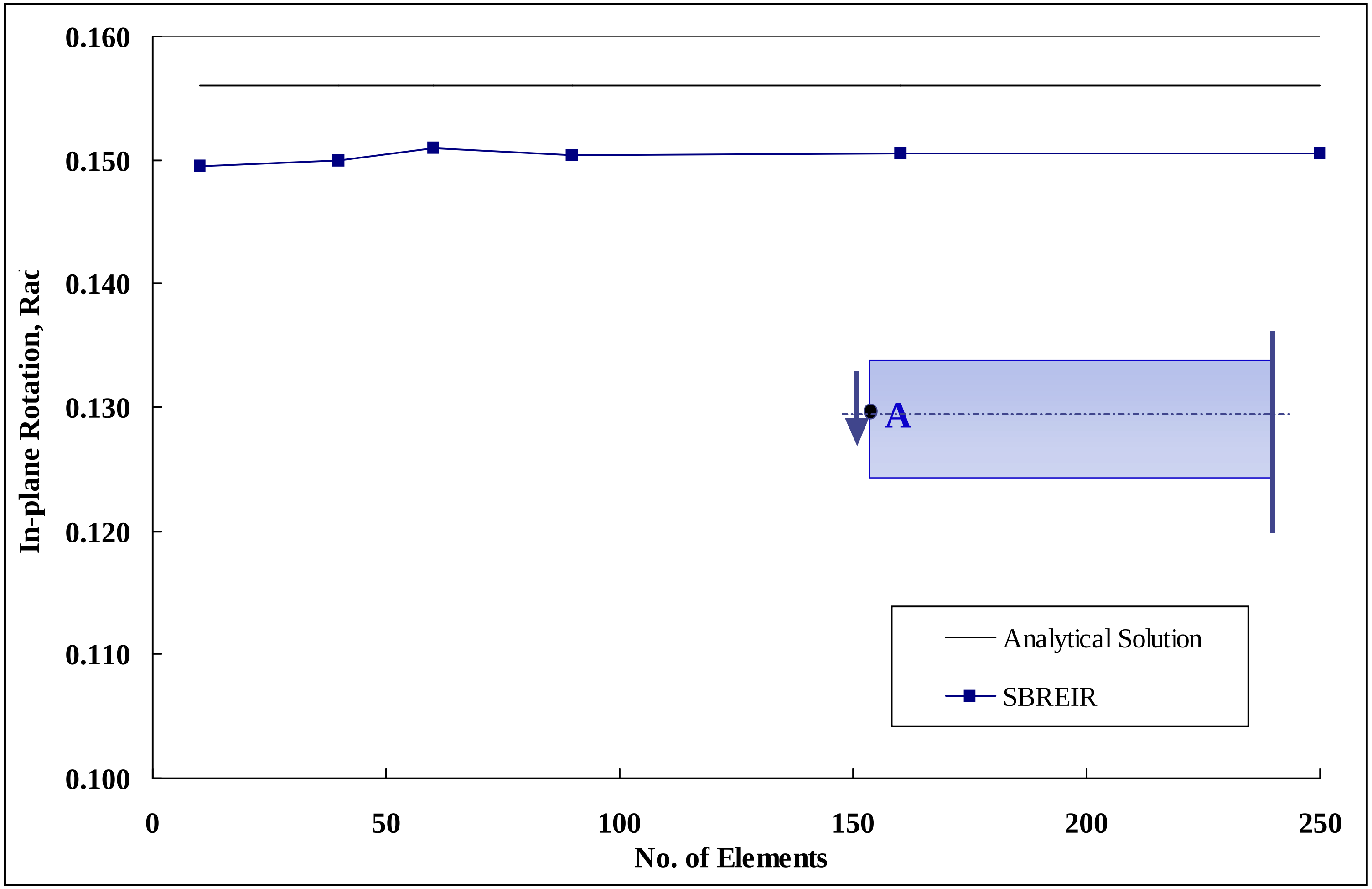


Figure 5-8: In-Plane Rotation at "A", (mm) in the Deep Cantilever Beam Using Rectangular Elements

5.5 SIMPLY SUPPORTED BEAM PROBLEM

The same simply supported problem that was described in Section 4.5 was solved again using the new rectangular element.

5.5.1 Used Mesh Size

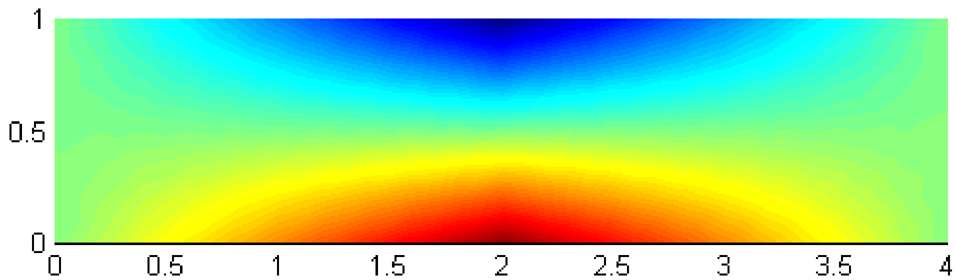
The following table shows the number of elements, number of nodes and aspect ratio for each mesh size.

Table 5-3: Mesh Size and Aspect Ratio of the Simply Supported Beam Using Rectangular Elements

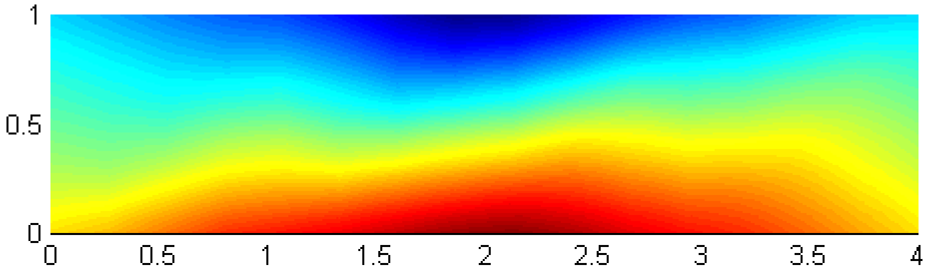
Elements in Short Side (L=1m)		Elements in Long Side (L=4m)		Aspect Ratio	Mesh Size	Total no. of Rectangular Elements	Total no. of Nodes
No.	Dimension (m)	No.	Dimension (m)				
1	1.000	4	1.000	1 : 1	1 x 4	8	10
2	0.500	8	0.500	1 : 1	2 x 8	32	27
3	0.333	12	0.333	1 : 1	3 x 12	72	52
4	0.250	16	0.250	1 : 1	4 x 16	128	85
5	0.200	20	0.200	1 : 1	5 x 20	200	126

5.5.2 Convergence Results

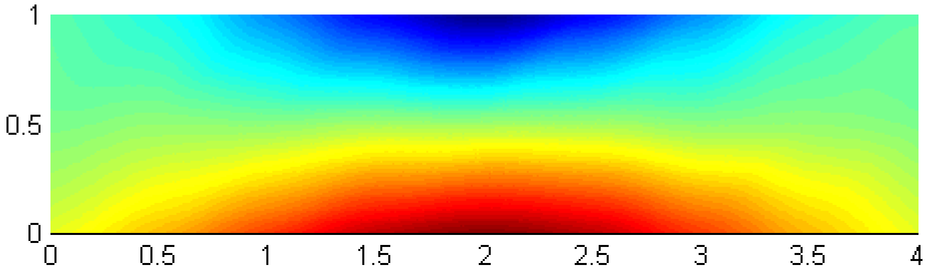
The problem was solved using the developed computer program (described in Chapter 3 and Annex A) using each of the mesh sizes listed in Table 5-3 above. Convergence of the overall pattern of bending stress in the simple beam is shown below for the analytical solution as well as the rectangular elements (BRE and SBREIR) using each mesh size (Red: tension, Blue: compression).



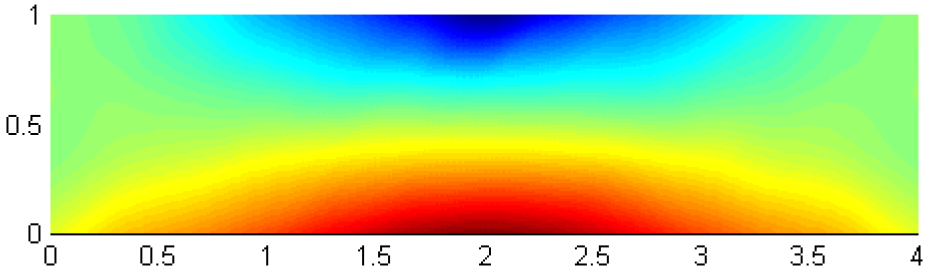
Bending Stress Pattern – Analytical Solution (repeated)



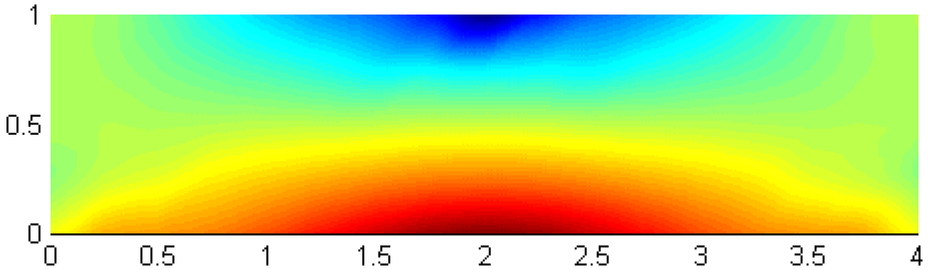
BRE Mesh Size 1x4



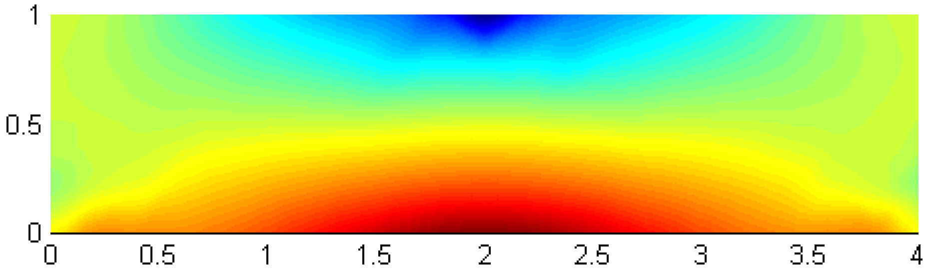
BRE Mesh Size 2x8



BRE Mesh Size 3x12

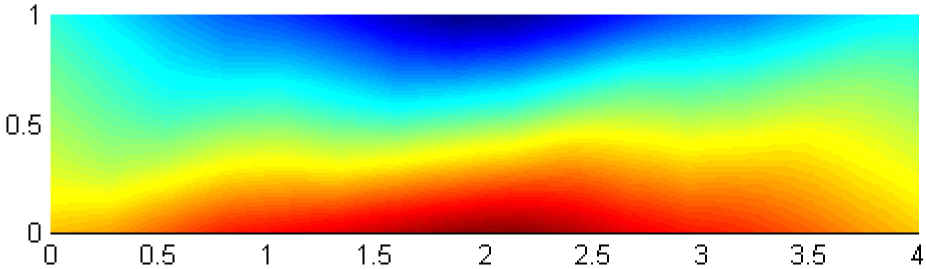


BRE Mesh Size 4x16

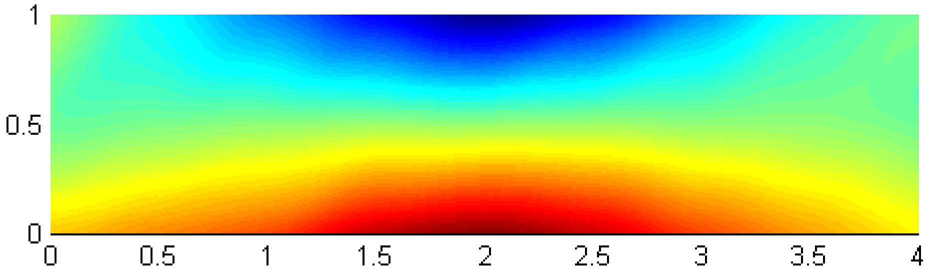


BRE Mesh Size 5x20

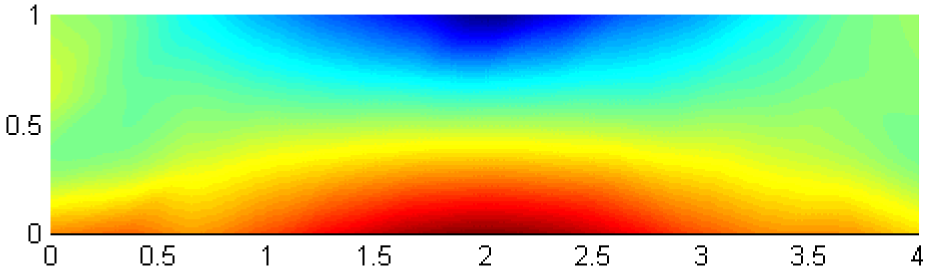
Figure 5-9: Bending Stress Pattern in the Simply Supported Beam Using BRE



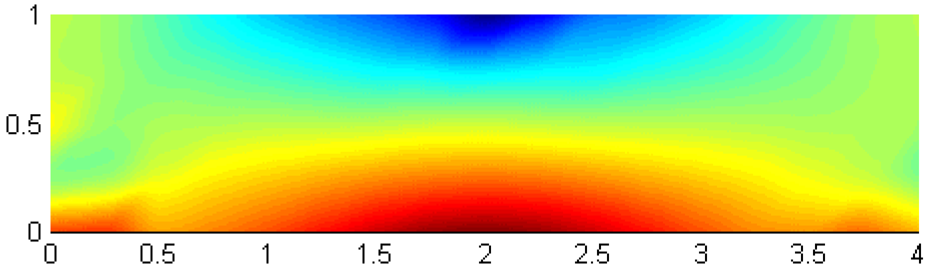
SBREIR Mesh Size 1x4



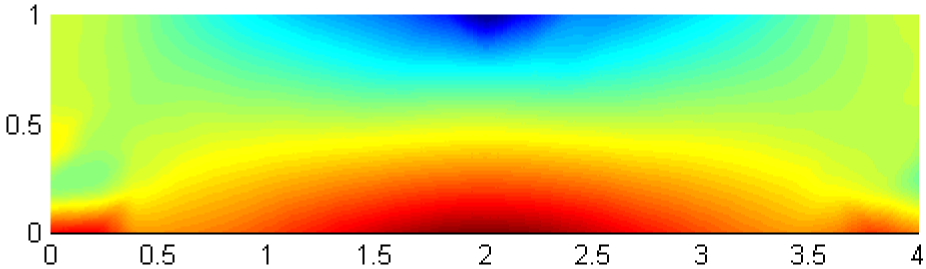
SBREIR Mesh Size 2x8



SBREIR Mesh Size 3x12



SBREIR Mesh Size 4x16



SBREIR Mesh Size 5x20

Figure 5-10: Bending Stress Pattern in the Simply Supported Beam Using SBREIR

Table 5-4 shows a summary of the used meshes and numerical results for vertical deflection, bending stress and shearing stress at the specified points within the simply supported beam as a percentage of the exact solutions for both the BRE and the new SBREIR elements.

Table 5-4: Results of the Simply Supported Beam Problem Using Rectangular Elements (BRE and SBREIR)

Mesh Size	No. of Elements	No. of Nodes	Vertical Deflection at A (mm)		Bending Stress at B (KPa)		Shearing Stress at C (KPa)	
			BRE	SBREIR	BRE	SBREIR	BRE	SBREIR
1 x 4	4	10	47.77%	52.23%	67.86%	97.78%	9.52%	38.25%
2 x 8	16	27	62.05%	62.80%	86.35%	96.55%	54.92%	83.33%
3 x 12*	36	52	69.35%	68.45%	92.42%	97.38%	66.51%	90.63%
4 x 16	64	85	73.66%	72.17%	95.16%	97.94%	86.83%	100.48%
5 x 20*	100	126	76.64%	75.00%	96.39%	97.94%	94.92%	97.94%
Exact Solutions			**6.720**		**25.20**		**6.30**	

* Note: In the case that any of the required points does not lie on a node, (like the mesh sizes of 3x12 and 5x20 in this problem), results of bending stress and/or shearing stress are averaged from the nearest nodes to the location of the required point.

Again for this problem, it is shown that the new rectangular element gives higher accuracy results than the BRE for the cases of bending stress and shearing stress. For vertical deflection the results are almost the same as those obtained by the BRE. Both elements converge to the same values as the number of elements increases.

Figures 5-11 to 5-13 show graphical comparison between the results obtained by each of the SBREIR, the BRE and the exact analytical solution.

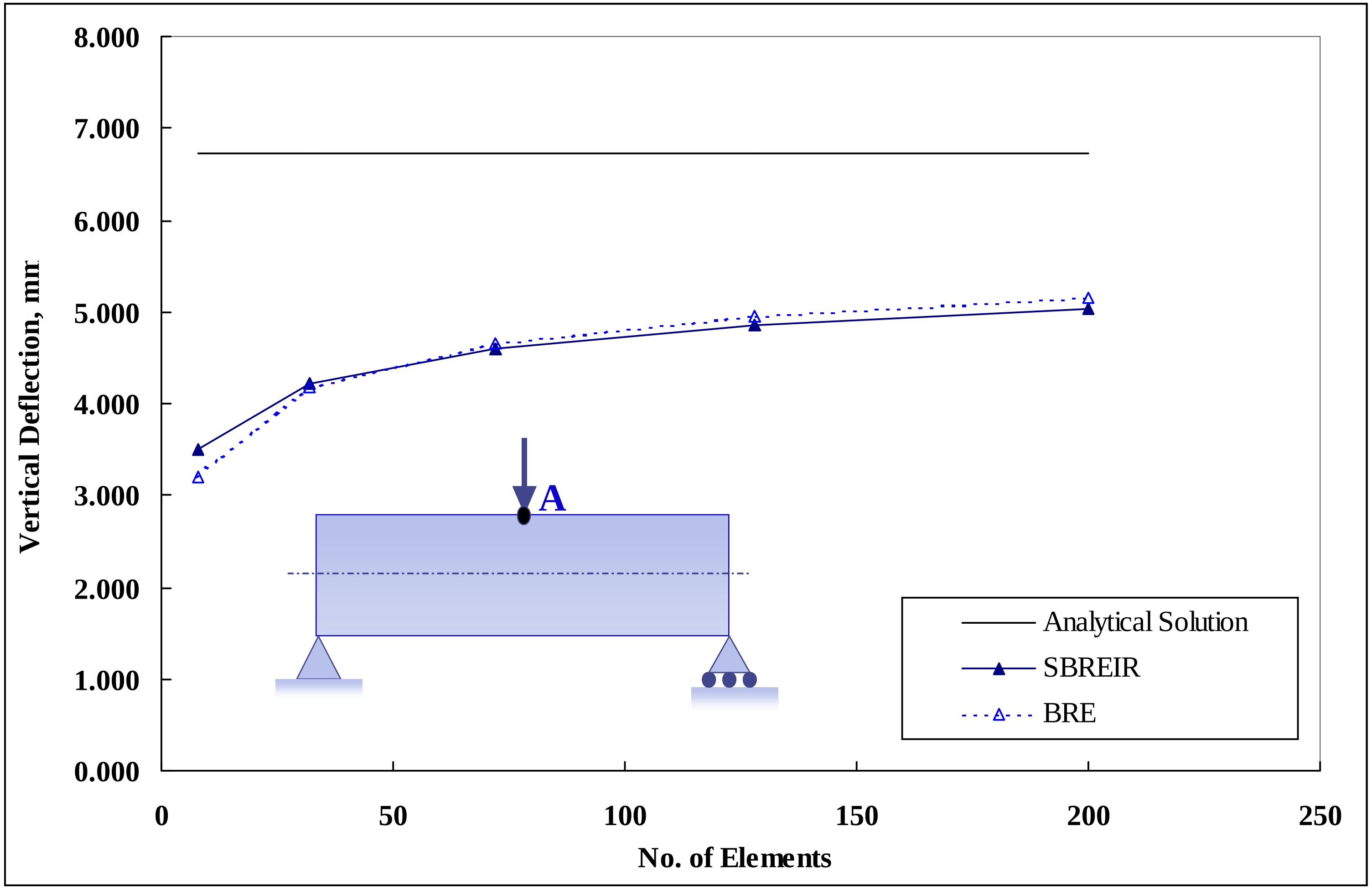


Figure 5-11: Vertical Deflection at “A”, (mm) in the Simply Supported Beam Using Rectangular Elements

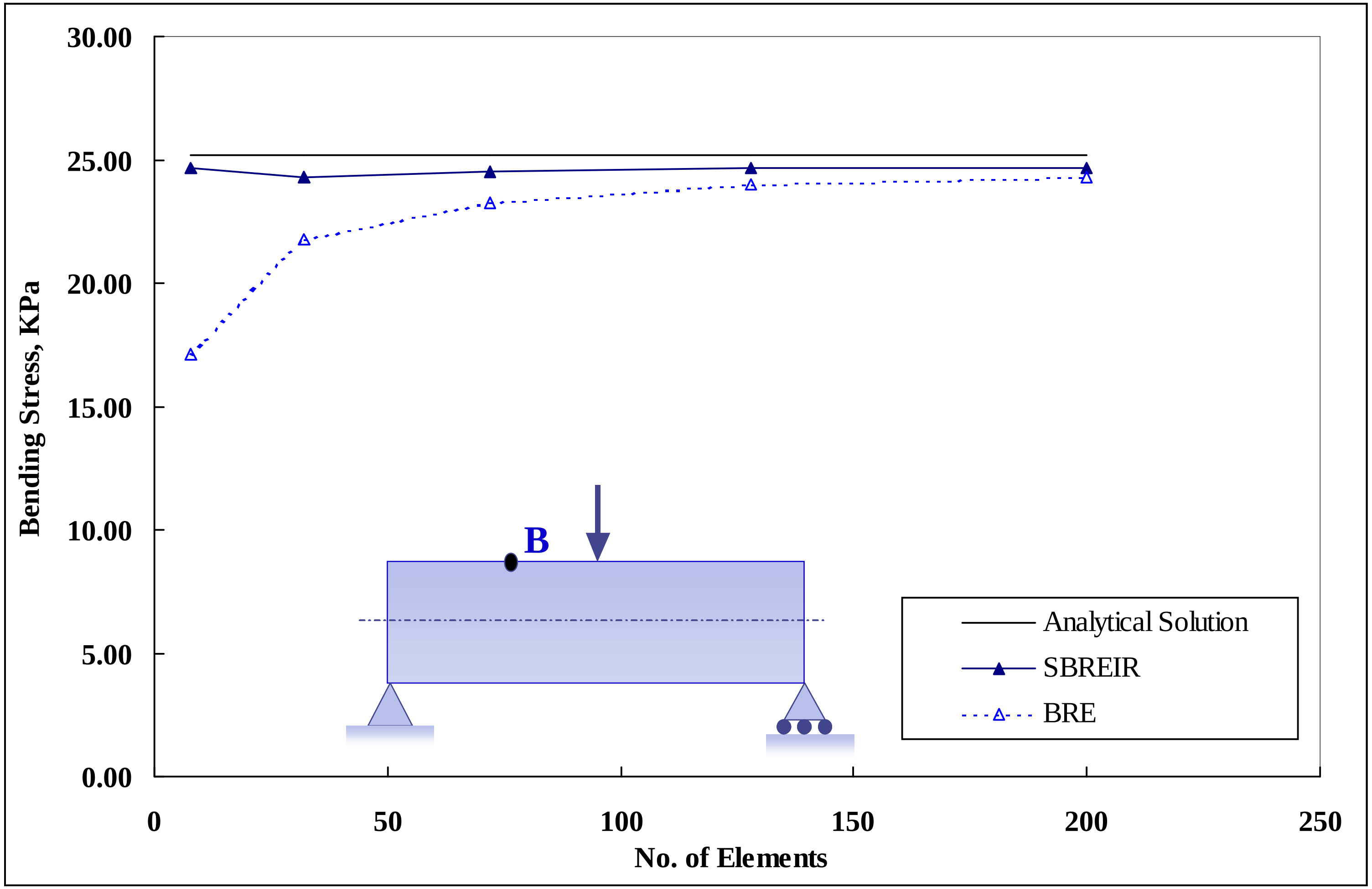


Figure 5-12: Bending Stress at "B", (KPa) in the Simply Supported Beam Using Rectangular Elements

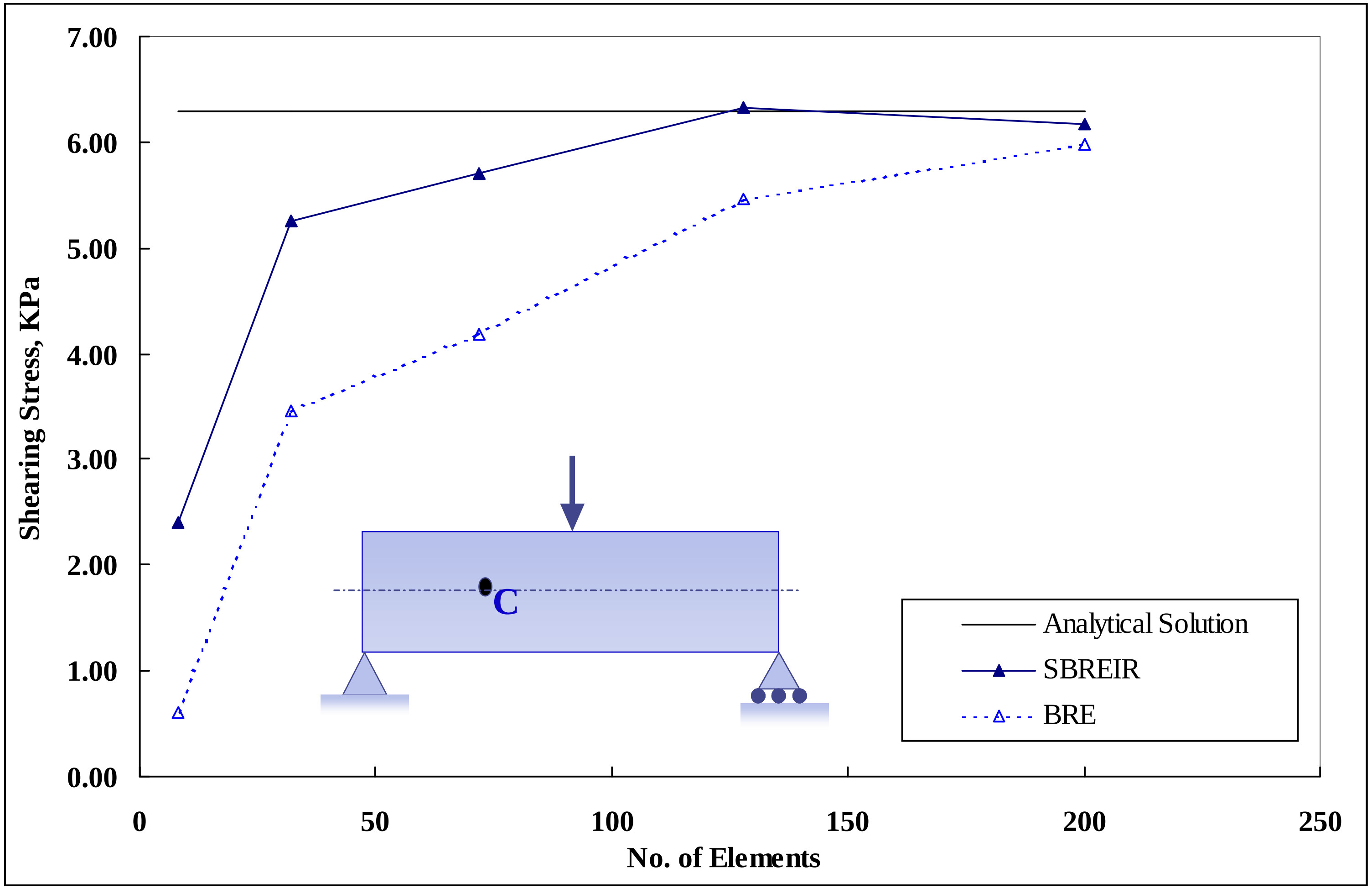


Figure 5-13: Shearing Stress at "C", (KPa) in the Simply Supported Beam Using Rectangular Elements

6. CHAPTER SIX: COMPARISON OF THE NEW TRIANGULAR AND RECTANGULAR ELEMENTS

The performance of the new triangular and rectangular elements is now studied by comparing their results of deflection and stresses for the problems of deep cantilever beam and simply supported beam that were solved previously. The results are compared based on the number of nodes for each mesh size.

6.1 SUMMARY AND DISCUSSION OF RESULTS FOR THE DEEP CANTILEVER BEAM

The following tables show a summary of results obtained for the deep cantilever beam problem using various triangular and rectangular elements. Both results of available elements (CST & BRE) and the new elements (SBTREIR & SBREIR) are included.

Tables 6-1 to 6-4 summarize the results for vertical deflection at point "A", bending stress at "B", shearing stress at point "C" and in-plane rotation at point "A" respectively. Graphical representations of these results are shown in Figures 6-1 to 6-4.

Table 6-1: Vertical Deflection at "A" (mm) in the Deep Cantilever Beam from Various Elements

Mesh Size	No. of Nodes	Triangular Elements		Rectangular Elements	
		CST	Tri-SBTREIR	BRE	Rect-SBREIR
2x5	18	57.19%	71.13%	88.60%	96.47%
4x10	55	83.44%	89.41%	96.92%	99.00%
5x12	78	88.51%	92.67%	97.92%	99.37%
6x15	112	91.86%	95.57%	98.73%	99.55%
8x20	189	95.29%	98.28%	99.37%	99.82%
10x25	286	---	---	99.64%	99.91%
Analytical Solution = 1.105					

Table 6-2: Bending Stress at "B" (KPa) in the Deep Cantilever Beam from Various Elements

Mesh Size	No. of Nodes	Triangular Elements		Rectangular Elements	
		CST	SBTREIR	BRE	SBREIR
2x5	18	36.53%	70.18%	91.57%	100.07%
4x10	55	67.78%	87.48%	98.23%	100.19%
5x12	78	75.00%	90.66%	98.88%	100.22%
6x15	112	80.17%	93.18%	99.43%	100.07%
8x20	189	86.11%	96.00%	99.80%	100.10%
10 x 25	286	---	---	99.96%	100.01%
Analytical Solution = 3000.00					

Table 6-3: Shearing Stress at "C", (KPa) in the Deep Cantilever Beam from Various Elements

Mesh Size	No. of Nodes	Triangular Elements		Rectangular Elements	
		CST	SBTREIR	BRE	SBREIR
2x5	18	63.17%	63.11%	69.63%	112.31%
4x10	55	85.47%	97.79%	86.92%	96.66%
5x12	78	86.83%	94.73%	94.49%	92.45%
6x15	112	93.28%	99.18%	96.38%	95.81%
8x20	189	96.08%	99.59%	96.66%	98.35%
10 x 25	286	---	---	98.69%	101.05%
Analytical Solution = 600.00					

Table 6-4: In-Plane Rotation at "A", (KPa) in the Deep Cantilever Beam from New Elements

Mesh Size	No. of Nodes	Triangular Elements		Rectangular Elements	
		CST	SBTREIR	BRE	SBREIR
2x5	18	---*	72.1%	---*	95.9%
4x10	55	---	88.1%	---	96.1%
5x12	78	---	91.4%	---	96.8%
6x15	112	---	93.4%	---	96.4%
8x20	189	---	95.7%	---	96.5%
10 x 25	286	---	---	---	96.5%
Analytical Solution = 0.156					

* Note: CST and BRE do not give the value of in-plane rotation.

- The CST has the least accurate results for deflection and stresses. The new triangular element (SBTREIR) gives higher accuracy results than the CST for the cases of vertical deflection, bending stress and shear stress. This is attributed to the higher number of degrees of freedom per node and the quadratic variation of displacement fields in the SBTREIR.

- The new rectangular element (SBREIR) gives higher accuracy results than the BRE for the case of vertical deflection and bending stress. This is attributed to the higher number of degrees of freedom per node and the quadratic variation of displacement fields in the SBREIR. For the bending stress, the results are very close but slightly more than the exact solution. For the shear stress, the results are almost the same as those obtained by the BRE.
- Results of all elements are not good when the structure is divided into a small number of elements (i.e. at course mesh size) as anticipated, especially for the shearing stress.
- For all elements, the solution converges to the analytical value as more elements and nodes are used (mesh refinement). Also, the rate of convergence slows down as more refinement is made. These are well known features of the finite element solutions
- The new strain based elements give the value of in-plane rotation while the CST and BRE don't. This is considered as an advantage of the new elements over the CST and BRE.

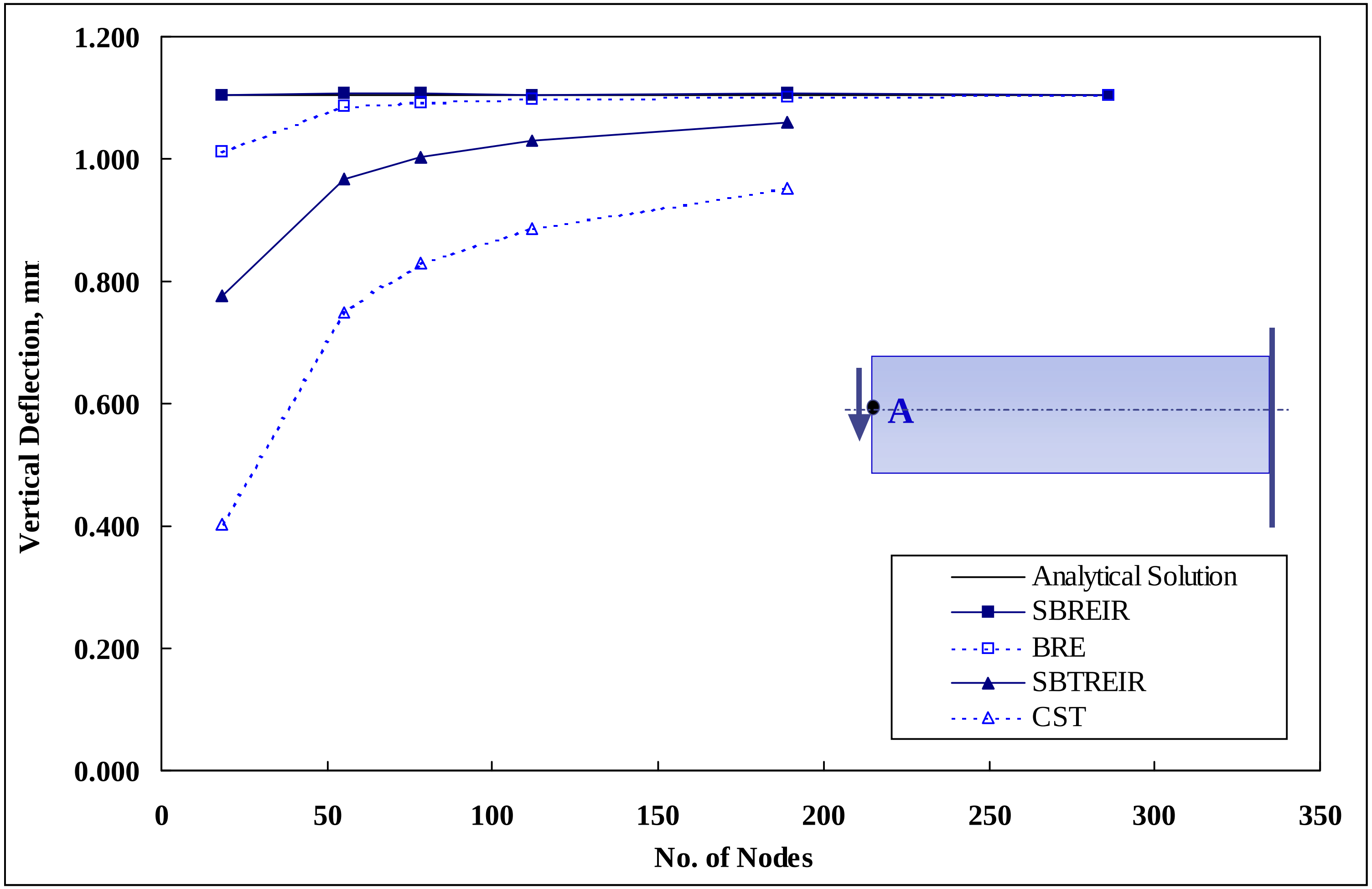


Figure 6-1: Vertical Deflection at "A" (mm) in the Deep Cantilever Beam from Various Elements

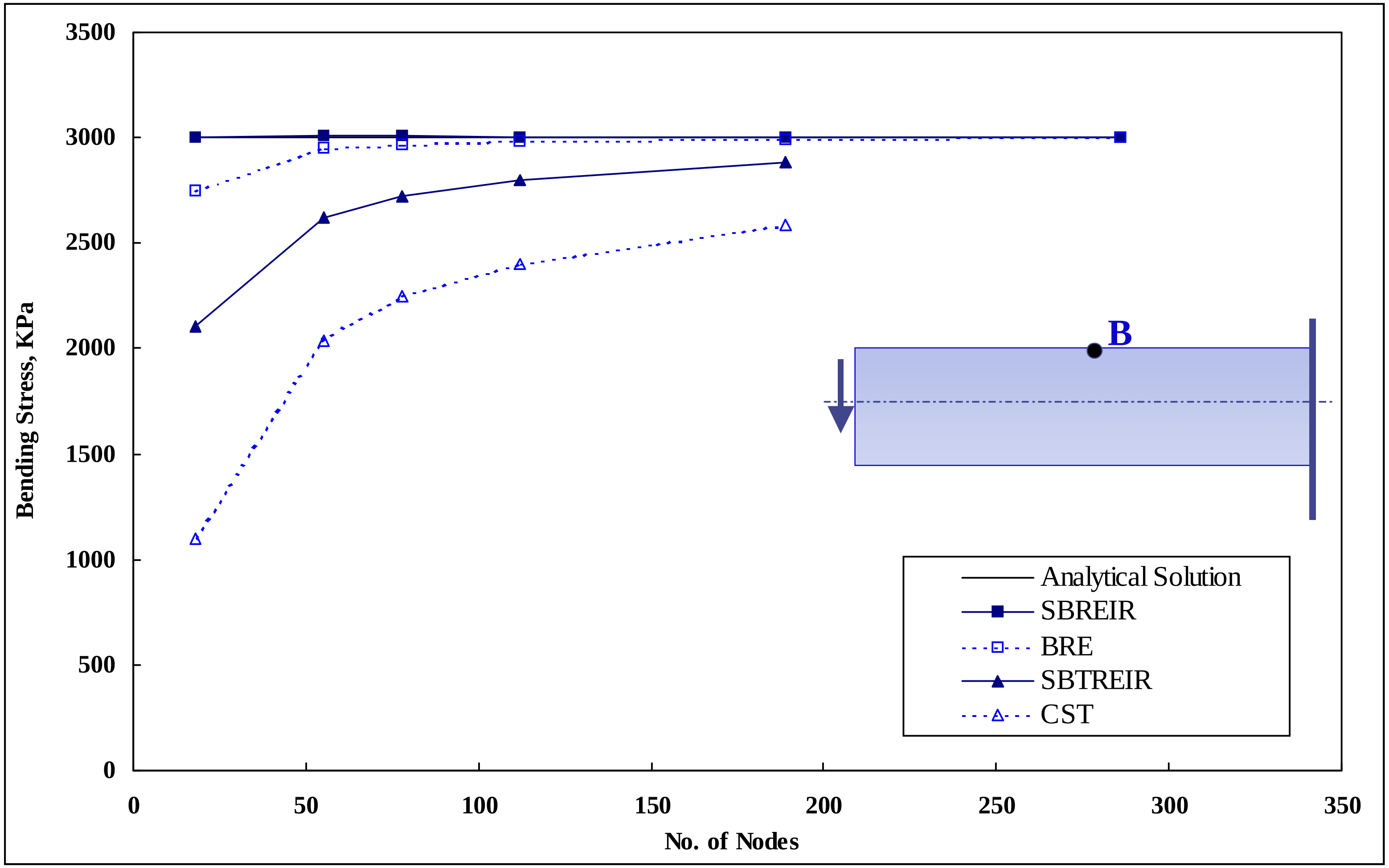


Figure 6-2: Bending Stress at "B" (KPa) in the Deep Cantilever Beam from Various Elements

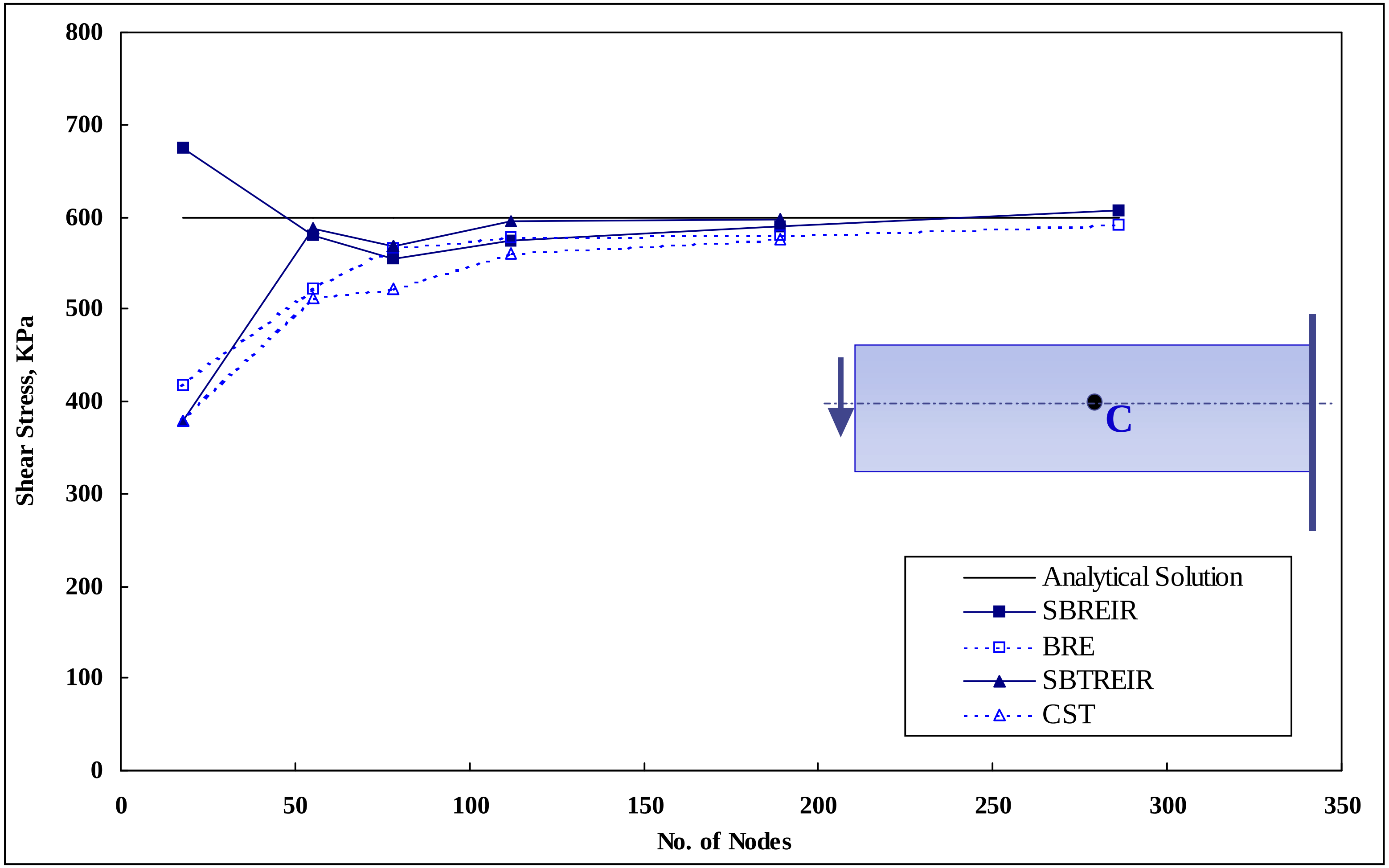


Figure 6-3: Shearing Stress at "C", (KPa) in the Deep Cantilever Beam from Various Elements

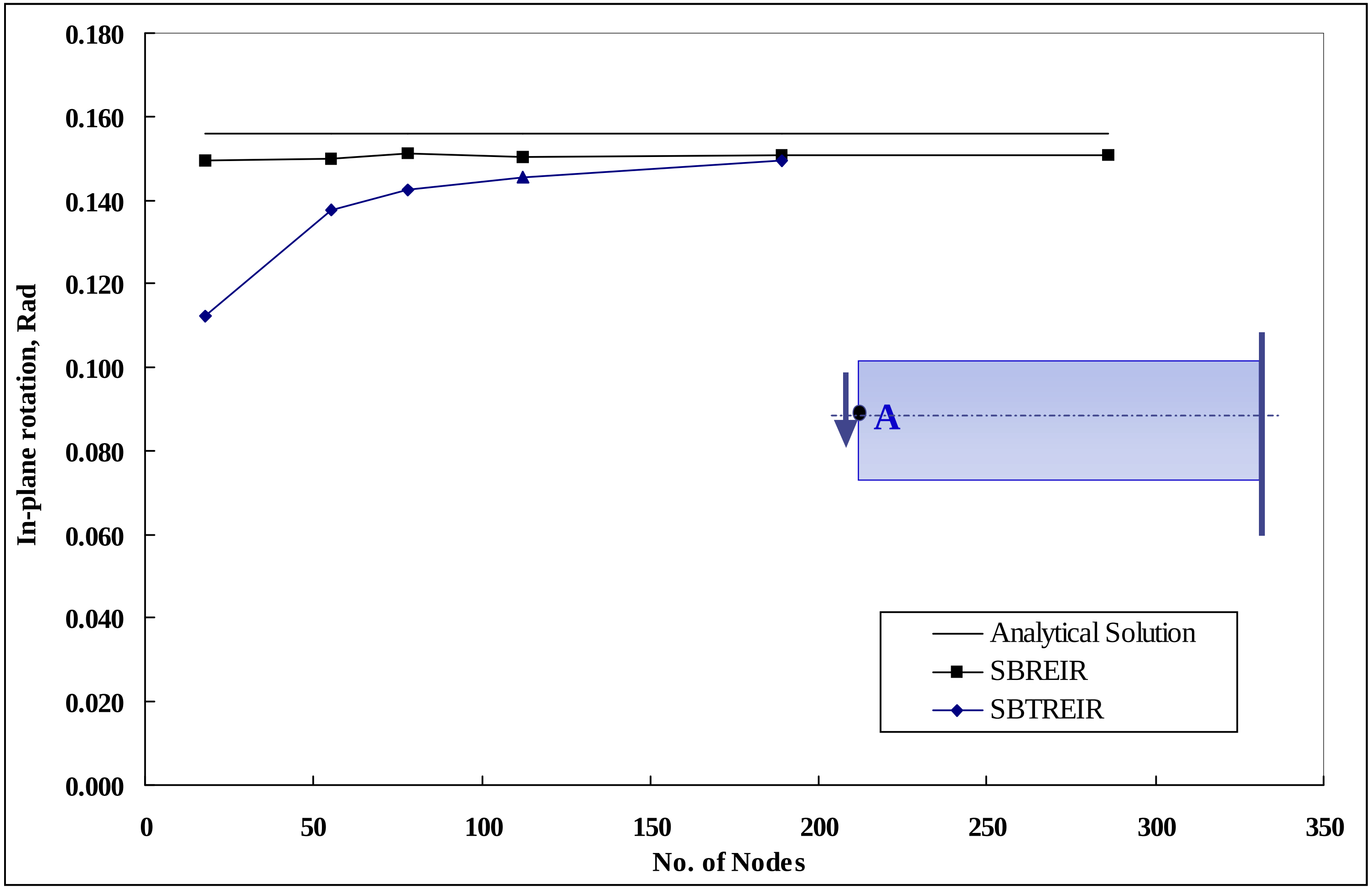


Figure 6-4: In-Plane Rotation at "A", (Rad) in the Deep Cantilever Beam from the new Elements

6.2 SUMMARY AND DISCUSSION OF RESULTS FOR THE SIMPLY SUPPORTED BEAM

The following tables show a summary of results obtained for the simply supported beam problem using various triangular and rectangular elements. Both results of available elements (CST & BRE) and the new elements (SBTREIR & SBREIR) are included.

Tables 6-5 to 6-7 summarize the results for vertical deflection at point "A", bending stress at "B" and shearing stress at point "C" respectively. Graphical representations of these results are shown in Figures 6-5 to 6-7.

Table 6-5: Vertical Deflection at "A" (mm) in the Simply Supported Beam from Various Elements

Mesh Size	No. of Nodes	Triangular Elements		Rectangular Elements	
		CST	SBTREIR	BRE	SBREIR
1 x 4	10	2.14	3.39	3.21	3.51
2 x 8	27	3.25	4.21	4.17	4.22
3 x 12	52	3.90	4.66	4.66	4.60
4 x 16	85	4.31	4.95	4.95	4.85
5 x 20	126	4.59	5.18	5.15	5.04
Analytical Solution = 6.72					

Table 6-6: Bending Stress at "B" (KPa) in the Simply Supported Beam from Various Elements

Mesh Size	No. of Nodes	Triangular Elements		Rectangular Elements	
		CST	SBTREIR	BRE	SBREIR
1 x 4	10	3.21	19.75	17.10	24.64
2 x 8	27	8.98	22.02	21.76	24.32
3 x 12	52	13.48	23.07	23.29	24.54
4 x 16	85	16.39	23.64	23.98	24.68
5 x 20	126	18.23	24.01	24.29	24.68
Analytical Solution = 25.20					

Table 6-7: Shearing Stress at "C", (KPa) in the Simply Supported Beam from Various Elements

Mesh Size	No. of Nodes	Triangular Elements		Rectangular Elements	
		CST	Tri-SBTREIR	BRE	Rect-SBREIR
1 x 4	10	3.97	4.10	0.60	2.41
2 x 8	27	3.91	5.15	3.46	5.25
3 x 12	52	4.38	5.54	4.19	5.71
4 x 16	85	5.44	6.14	5.47	6.33
5 x 20	126	5.50	6.21	5.98	6.17
Analytical Solution = 6.30					

- The CST has the least accurate results for deflection and stresses. The new triangular element (SBTREIR) gives higher accuracy for results than the CST for the cases of vertical deflection, bending stress and shear stress. This is attributed to the higher number of degrees of freedom per node and the quadratic variation of displacement fields in the SBTREIR.
- The new triangular element (SBTREIR), the BRE, and the new rectangular element (SBREIR) give almost the same results for deflection and bending stress.
- The new rectangular element (SBREIR) gives higher accuracy for results than the BRE for the case of vertical deflection and bending stress. For the bending stress, the results are very close to the exact solution. For vertical deflection the results are almost the same as those obtained by the BRE.
- Results of all elements are not good when the structure is divided into a small number of elements (i.e. at course mesh size) as anticipated, especially for the shearing stress.
- For all elements, the solution converges to the analytical value as more elements and nodes are used (mesh refinement). Also, the rate of convergence slows down as more refinement is made. These are well known features of the finite element solutions

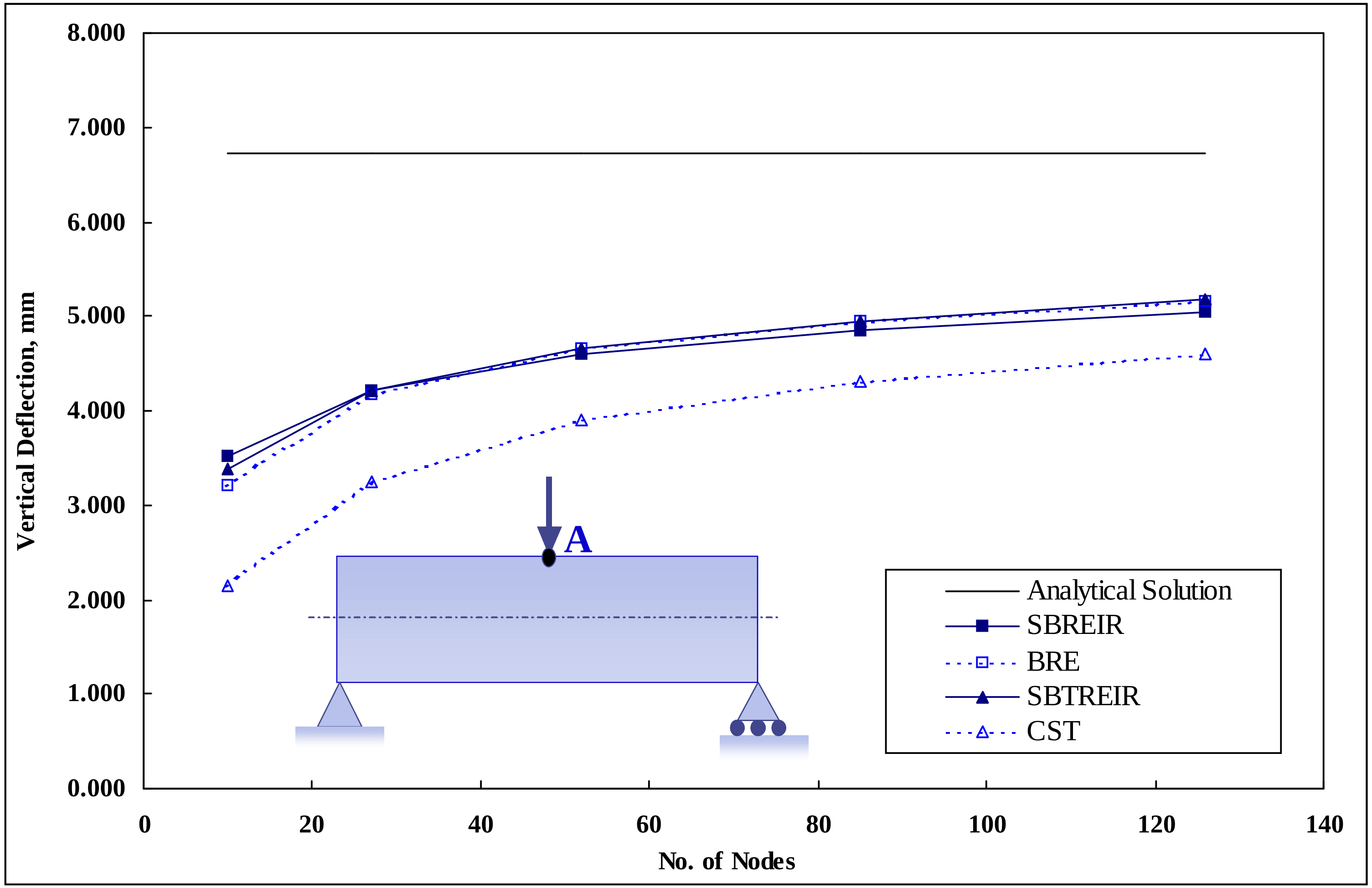


Figure 6-5: Vertical Deflection at Point "A" (mm) in the Simply Supported Beam from Various Elements

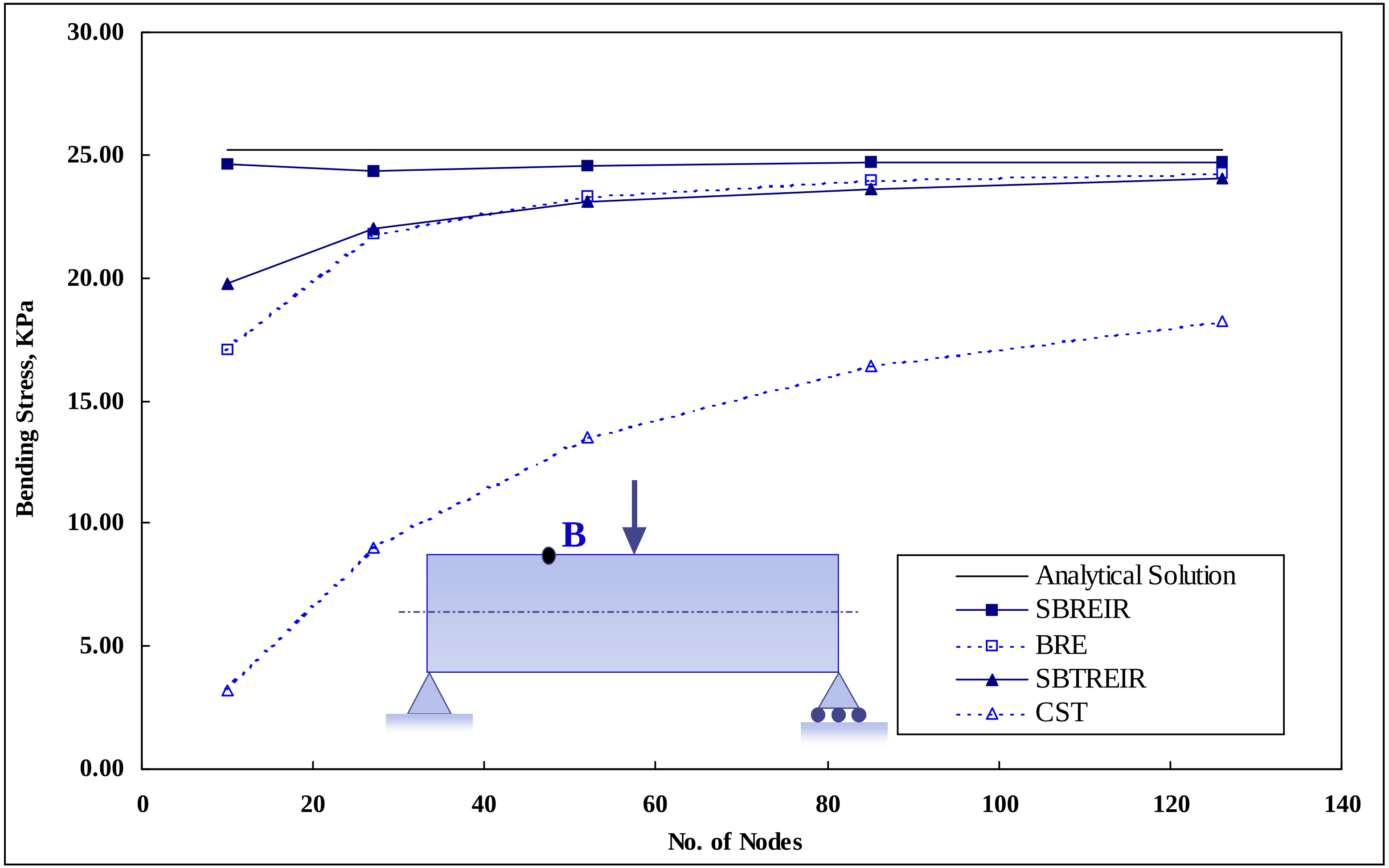


Figure 6-6: Bending Stress at "B" (KPa) in the Simply Supported Beam From Various Elements

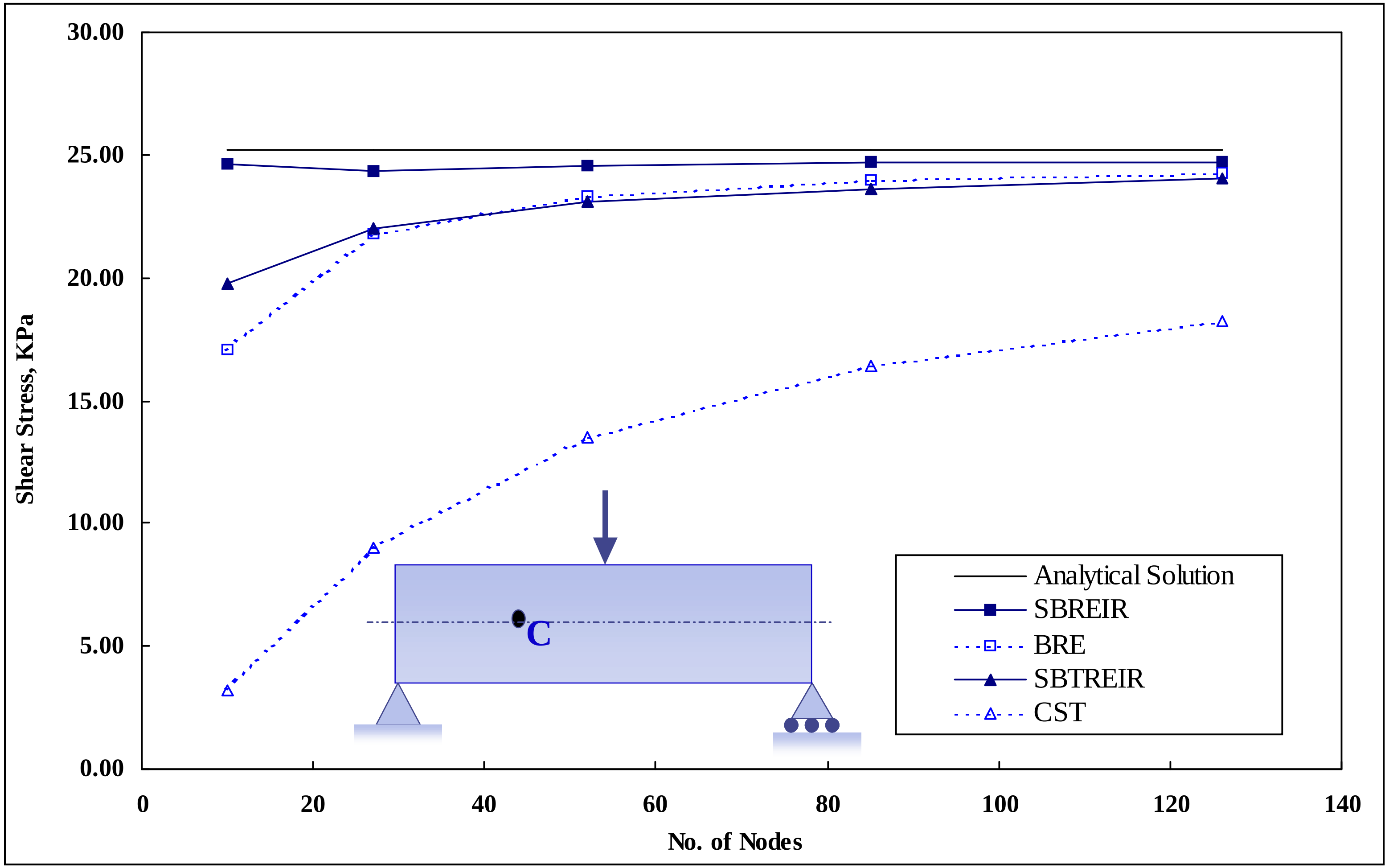


Figure 6-7: Shearing Stress at "C" (KPa) in the Simply Supported Beam From Various Elements

7. CHAPTER SEVEN: CONCLUSIONS AND RECOMMENDATIONS

7.1 CONCLUSIONS

Two new strain-based triangular and rectangular finite elements in cartesian coordinates system, for two dimensional elasticity problems have been developed in this thesis. To test the performance of these elements, they have been applied to solve two common plane elasticity problems: a deep cantilever beam problem and a simply supported beam problem. Results obtained using the new elements were compared to those of the well-known constant strain triangular element (CST) and the bilinear rectangular element (BRE). All results are then compared with the exact elasticity solutions.

It is concluded that

§ The new triangular element (SBTREIR) gives higher accuracy results than the (CST) for the cases of vertical deflection, bending stress and shear stress for both of the considered problems.

§ The new rectangular element (SBREIR) gives higher accuracy results than those of the (CST), (BRE) and (SBTREIR) for vertical deflection and bending stress in the case of the deep cantilever beam problem. Results for bending stress are very close to the exact solution while the results for shearing stress are almost the same as those obtained by the (BRE).

§ Results obtained by (SBREIR, SBTREIR and BRE) are almost the same for deflection and bending stress in the case of the simply supported beam problem. Their results are of higher accuracy than the results of the (CST).

7.2 RECOMMENDATIONS

Even though the new strain-based triangular and rectangular elements give accurate results and compare well to the CST and BRE respectively, it is recommended that another future research be conducted to compare our proposed elements to higher order versions of displacement-based elements such as the linear strain triangle (LST) and the quadratic isoparametric rectangular element.

REFERENCES

[1] Ashwell, D. G. and Sabir A. B., "A New Cylindrical Shell Element Based on Simple Independent Strain Functions", Int. J. Mech. Sci., Vol. 14, pp. 171-183, 1972.

[2] Ashwell, D. G., Sabir A. B. and Roberts, T. M., "Further studies in the application of curved finite elements to circular arches", Int. J. Mech. Sci., Vol. 13, 1971.

[3] Ashwell, D. G., Sabir A. B. and Roberts, T. M., "Limitations of certain finite elements when applied to arches", Int. J. Mech. Sci., Vol. 13, 1971.

[4] Boresi, A. P. and Chong, K. P., "Elasticity in Engineering Mechanics", Second Edition, Elsevier, London, 1974.

[5] Courant R., "Variational Methods for the Solution of Problems of Equilibrium and Vibration", Bulletin of the American Mathematical Society, Vol. 49, pp. 1-23, 1943.

[6] Djoudi, M. S., "A Sector Finite Element for Plane Elasticity with In-Plane Rotation", M. Sc. Thesis, University of Wales, U.K.,1985.

[7] Djoudi, M. S., "Strain Based Shallow Shell Finite Element" Ph.D. Thesis, University of Wales, College of Cardiff, 1990

[8] El-Erris Hazim, "Finite Element Analysis of Shell Structures", Ph. D. Thesis, University of Wales, College of Cardiff, 1989.

[9] Hayrettin Kardestuncer, "Finite Element Handbook", McGrow-Hill, 1987.

[10] Hrennikoff, A., "Solution of Problems in Elasticity by The Frame Work Method", Journal of Applied Mechanics, Vol. 8, pp. 169-175, Dec. 1941.

[11] Irons, B. M., "Engineering Applications of Numerical Integration in stiffness Methods", Journal of the American Institute of Aeronautics and Astronautics, Vol. 4, No. 11, pp. 2035-2037, 1966.

[12] J. H. Argyris and S. Kelsey, "Energy Theorems and Structural Analysis", Aircraft Engineering, Butterworths, London, 1954-1955.

[13] Levy, S., "Structural Analysis and Influence Coefficients for Aircraft Structures with Discontinuities and Sweepback", Journal of Aeronautical Sciences, Vol. 14, No. 10, pp 547-560, Oct. 1947.

[14] Levy, S., "Structural Analysis and Influence Coefficients for Delta Wings", Journal of Aeronautical Sciences, Vol. 20, No. 7, pp 449-454, July. 1953.

[15] Logan D. L., "A First Course in the Finite Element Method", Second Edition, 1992.

[16] McHenry, D., "A Lattice Analogy of the Solution of Plane Stress Problems", Journal of Civil Engineers, Vol. 21, pp. 59-82, Dec. 1943.

[17] Mousa, A. I. and Sabir A. B., "Finite Element Analysis of Fluted Conical Shell Roof Structures", Civil-Comp-Ltd, Edinburgh, Scotland, Computational Structural Engineering for Practice, 1994, pp. 173-181.

[18] Mousa, A. I. and Sabir A. B., "Finite Element Analysis of Hyper Umbrella Shell Roof", Integrated Design Process Technology, IDPT-Vol. 1, Printed in USA, December 1995.

[19] Mousa, A. I. and Sabir A. B., "The Analysis of Storage Tanks by The Finite Element Method", Civil-Comp-Ltd, Edinburgh, Scotland, Computational Structural Engineering for Practice, 1994, pp. 189-196.

[20] Mousa, A. I., "Application of A Triangular and Rectangular Strain Based Finite Element Analysis to the Analysis of Doubly Curved Dam", Proceedings of Jordan 1st Civil Engineering Conference, Structural Engineering, Amman, Jordan, October 1997.

[21] Mousa, A. I., "Finite Element Analysis of Shell Structures". Ph. D. Dissertation, University of Wales, College of Cardiff, U.K., 1991.

[22] Mousa, A. I., "Finite Element Analysis of Rectangular-Plan Cylindrical Croined Vault", Al Azhar University Engineering Journal, AUEJ, Vol. 2, Egypt, 2001.

[23] Sabir A. B., "A new class of finite elements for plane elasticity problems", CAFEM7, 7th International Conference on Structural Mechanics in Reactor Technology, Chicago, August, 1983

[24] Sabir, A. B. and Ashwell, D. G., "Diffusion of Concentrated Loads into Thin Cylindrical Shells", The Math of Finite Element and Applications II, Academic Press, 33, pp. 379-389, 1979.

[25] Sabir, A. B. and Charchafchi T. A., "Curved Rectangular and General Quadrilateral Shell Element for Cylindrical Shells", The Math. of Finite Element and Applications II, Academic Press, pp. 213-239, 1982.

[26] Sabir, A. B. and Ramadanhi F., "A Shallow Shell Finite Element for General Shell Analysis", Proc. of Sec. Int. Conf. On Variational Methods in Eng., Univ. of Southampton, pp. 5-3 to –13, 1985.

[27] Sabir, A. B., "Stiffness Matrices for General Deformation (Out of Plane and In Plane) of Curved Beam Members Based on Independent Strain Functions", The Math. of Finite Element and Applications II, Academic Press, 3, pp. 411-421, 1975.

[28] Sabir, A. B., "Strain Based Finite Elements for the Analysis of Cylinders with Holes and Normally Intersecting Cylinders", Nuclear Eng. and Design, 76, pp. 111-1210, 1983.

[29] Sabir, A. B., "Strain Based Shallow Spherical Shell Element", Proc. Of Int. Conf. on the Mathematics of Finite Elements and Applications, Bunel University, 1987.

[30] Sabir, A. B., "THE NODAL solution routine for large number of linear simultaneous equations in finite element analysis of plates and shells", Finite elements for thin shells and curved members, Ashwell and Gallagher, Willey, 1976.

[31] Sfendji, A., "Finite Elements for Plane Elasticity Problems". M. Sc. Thesis, University of Wales, U.K., 1988.

[32] Timoshenko S. and Goodier J. N., "Theory of Elasticity", Third Edition, McGraw Hill, New York, 1985.

[33] Tocher, J. L., and Hartz, B, J., “Higher Order Finite Element for Plane Stress”, Journal of the Engineering Mechanics Division, Proceedings of the American Society of Civil Engineers, Vol. 93, No. EM4 pp. 149-174, Aug. 1967.

[34] Turner, M. J., Clough, R. W., Martin, H. C. and Topp, L. J., “Stiffness and Deflection Analysis of Complex Structures”, Journal of Aeronautical Sciences, Vol. 23, No. 9, pp. 805-824, Sept. 1956.

[35] Urugal, A. C. and Fenster, S. K., “Advanced strength and Applied Elasticity”, Second Edition, Prentice-Hall Inc., New Jersy, USA, 1987.

[36] Zienkiewicz O. C., “The Finite Elements Method”, Third Expanded and Revised Edition, McGrow-Hill, UK. 1977.

APPENDIX A: COMPUTER CODES

A.1 GENERATION OF TRIANGULAR MESH IN RECTANGULAR DOMAIN (USING MATLAB)

```
function [] = Trimesh
% Generate Simple Triangular Mesh
% By Salah
% Input Data: dimensions & No. of elements in x & y direction
fprintf('%s \n\n', '==========Generation of Triangular Mesh========
');
lx = input('Dimension in x-direction: ' );
ly = input('Dimension in y-direction: ' );
nelex = input('Enter No. of Elements in x-direction: ' );
neley = input('Enter No. of Elements in y-direction: ' );

%Calculations
%No. of elements and increment in x & y direction
nnodex = nelex + 1;
nnodey = neley + 1;
dx = lx / nelex;
dy = ly / neley;

% Generate Coordinates
nelexy = nelex * neley*2;    % total no. of Triangular elements
nnodexy = nnodex * nnodey;        % total no. of nodes
        % zero matrices and vectors
        xc=zeros(nnodexy,1);
        yc=zeros(nnodexy,1);
        nodedata=zeros(nnodexy,3);

for row = 1: nnodey
    for col = 1: nnodex
        nt = (row-1)* nnodex + col;
        xc(nt)= (col-1)*dx;
        yc(nt)= (row-1)*dy;
    end
end
% combining node data into a matrix nodedata=[no., xc, yc]
% and elementdata=[no., 3 nodes]
format short
for i= 1: nnodexy
    nodedata(i,1)= i;
    nodedata(i,2)= xc(i);
    nodedata(i,3)= yc(i);
 end
 for elerow = 1: neley
    for elecol = 1: nelex
        eleno = (elerow-1)* (nnodex-1) + elecol;
        node1 = eleno + (elerow-1);
        node2 = eleno + (elerow);
        node3 = node2 + nnodex;
        node4 = node1 + nnodex;
        elementdata(eleno,1)=eleno;
        elementdata(eleno,2)=node1;
        elementdata(eleno,3)=node2;
        elementdata(eleno,4)=node3;
```

```
        elementdata(eleno+nelex * neley,1)=eleno+nelex * neley;
        elementdata(eleno+nelex * neley,2)=node1;
        elementdata(eleno+nelex * neley,3)=node3;
        elementdata(eleno+nelex * neley,4)=node4;
     end
  end

% Plot of the structure
figure (1)
for n=1:nelexy
      x(1)= xc(elementdata(n,2));
      x(2)= xc(elementdata(n,3));
      x(3)= xc(elementdata(n,4));
      x(4)= x(1);
      y(1)= yc(elementdata(n,2));
      y(2)= yc(elementdata(n,3));
   y(3)= yc(elementdata(n,4));
   y(4)= y(1);
   plot (x, y, '-o', 'linewidth', 0.05, 'markersize', 1.5,
         'markerfacecolor', 'b')
   axis equal
  hold on
end
leng=length(xc);
for i=1:leng
   istr=num2str(i);
text(xc(i)+0.1*dx, yc(i)+0.1*dy, istr )
end

for i=1:length (elementdata)
   elementno=elementdata(i,1);
   nd1=elementdata((i),2);
   x1=xc(nd1);
   y1=yc(nd1);
   nd2=elementdata((i),3);
   x2=xc(nd2);
   y2=yc(nd2);
   nd3=elementdata((i),4);
   x3=xc(nd3);
   y3=yc(nd3);
   xx= [x1; x2; x3];
   yy= [y1; y2; y3];
   xaverage=mean(xx);
   yaverage=mean(yy);
   elnStr = num2str(elementno);
   text(xaverage, yaverage, elnStr )
end

nodedata
elementdata
NoOfElements = nelexy
NoOfNodes= nnodexy
```

A1.1 Sample Triangular Mesh

Input

```
lx = input('Dimension in x-direction: ' ); 10
ly = input('Dimension in y-direction: ' ); 4
nelex = input('Enter No. of Elements in x-direction: ' ); 5
neley = input('Enter No. of Elements in y-direction: ' ); 2
```

Results

```
nodedata =
     ١     ٠     ٠
     ٢     ٢     ٠
     ٣     ٤     ٠
     ٤     ٦     ٠
     ٥     ٨     ٠
     ٦    ١٠     ٠
     ٧     ٠     ٢
     ٨     ٢     ٢
     ٩     ٤     ٢
    ١٠     ٦     ٢
    ١١     ٨     ٢
    ١٢    ١٠     ٢
    ١٣     ٠     ٤
    ١٤     ٢     ٤
    ١٥     ٤     ٤
    ١٦     ٦     ٤
    ١٧     ٨     ٤
    ١٨    ١٠     ٤
elementdata =
     ١     ١     ٢     ٨
     ٢     ٢     ٣     ٩
     ٣     ٣     ٤    ١٠
     ٤     ٤     ٥    ١١
     ٥     ٥     ٦    ١٢
     ٦     ٧     ٨    ١٤
     ٧     ٨     ٩    ١٥
     ٨     ٩    ١٠    ١٦
     ٩    ١٠    ١١    ١٧
    ١٠    ١١    ١٢    ١٨
    ١١     ١     ٨     ٧
    ١٢     ٢     ٩     ٨
    ١٣     ٣    ١٠     ٩
    ١٤     ٤    ١١    ١٠
    ١٥     ٥    ١٢    ١١
    ١٦     ٧    ١٤    ١٣
    ١٧     ٨    ١٥    ١٤
    ١٨     ٩    ١٦    ١٥
    ١٩    ١٠    ١٧    ١٦
    ٢٠    ١١    ١٨    ١٧
NoOfElements =     ٢٠
NoOfNodes =        ١٨
```

A.2 GENERATION OF RECTANGULAR MESH IN RECTANGULAR DOMAIN (USING MATLAB)

```
function [] = recmesh
% Generate Simple Rectangular Mesh
% By Salah
% Input Data: dimensions & No. of elements in x & y direction
fprintf('%s \n\n', '=======Generation of Rectanglar Mesh========= ');
lx = input('Dimension in x-direction: ' );
ly = input('Dimension in y-direction: ' );
nelex = input('Enter No. of Elements in x-direction: ' );
neley = input('Enter No. of Elements in y-direction: ' );

%Calculations
%No. of elements and increment in x & y direction
nnodex = nelex + 1;
nnodey = neley + 1;
dx = lx / nelex;
dy = ly / neley;

% Generate Coordinates
nelexy = nelex * neley;    % total no. of elements
nnodexy = nnodex * nnodey;        % total no. of nodes
        % zero matrices and vectors
        xc=zeros(nnodexy,1);
        yc=zeros(nnodexy,1);
        nodedata=zeros(nnodexy,3);

for row = 1: nnodey
    for col = 1: nnodex
        nt = (row-1)* nnodex + col;
        xc(nt)= (col-1)*dx;
        yc(nt)= (row-1)*dy;
    end
end
% combining node data into a matrix nodedata=[no., xc, yc]
% and elementdata=[no., 4 nodes]
for i= 1: nnodexy
    nodedata(i,1)= i;
    nodedata(i,2)= xc(i);
    nodedata(i,3)= yc(i);
 end
 for elerow = 1: neley
    for elecol = 1: nelex
        eleno = (elerow-1)* (nnodex-1) + elecol;
        node1 = eleno + (elerow-1);
        node2 = eleno + (elerow);
        node3 = node2 + nnodex;
        node4 = node1 + nnodex;
        elementdata(eleno,1)=eleno;
        elementdata(eleno,2)=node1;
        elementdata(eleno,3)=node2;
        elementdata(eleno,4)=node3;
        elementdata(eleno,5)=node4;
     end
  end
```

```
% Plot of the structure
figure (1)
plot (xc,yc, '-o','linewidth', 0.05,'markersize', 1.5,
'markerfacecolor', 'b')
grid on
axis equal
elementdata
nodedata
NoOfElements = nelexy
NoOfNodes= nnodexy
```

A2.1 Sample Rectangular Mesh

Input

```
lx = input('Dimension in x-direction: ' ); 10
ly = input('Dimension in y-direction: ' ); 4
nelex = input('Enter No. of Elements in x-direction: ' ); 5
neley = input('Enter No. of Elements in y-direction: ' ); 2
```

Results

```
elementdata =
     ١     ١     ٢     ٨     ٧
     ٢     ٢     ٣     ٩     ٨
     ٣     ٣     ٤    ١٠     ٩
     ٤     ٤     ٥    ١١    ١٠
     ٥     ٥     ٦    ١٢    ١١
     ٦     ٧     ٨    ١٤    ١٣
     ٧     ٨     ٩    ١٥    ١٤
     ٨     ٩    ١٠    ١٦    ١٥
     ٩    ١٠    ١١    ١٧    ١٦
    ١٠    ١١    ١٢    ١٨    ١٧

nodedata =
     ١     ٠     ٠
     ٢     ٢     ٠
     ٣     ٤     ٠
     ٤     ٦     ٠
     ٥     ٨     ٠
     ٦    ١٠     ٠
     ٧     ٠     ٢
     ٨     ٢     ٢
     ٩     ٤     ٢
    ١٠     ٦     ٢
    ١١     ٨     ٢
    ١٢    ١٠     ٢
    ١٣     ٠     ٤
    ١٤     ٢     ٤
    ١٥     ٤     ٤
    ١٦     ٦     ٤
    ١٧     ٨     ٤
    ١٨    ١٠     ٤


NoOfElements =    ١٠
NoOfNodes =    ١٨
```

A.3 MATLAB CODE FOR THE STRAIN BASED TRIANGULAR ELEMENT WITH IN-PLANE ROTATION (SBTREIR)

```
function [] = q3b
% New Strain Based Triangular Element with In-Plane Rotation
(SBTRIER)
% No. of Nodes = 3
% No. of DOF per Node = 3
% Total No. of DOF Node = 3x3=9
% By Salah
% 1) control parameters
InFileName=input('Name of Input File (without extension):','s');
OutFileName=input('Name of Output File (with .m extension):','s');
%Data input
eval (InFileName);
nnpel = 3;
ndofpn=3;
edof=nnpel*ndofpn;
sdof=nnode*ndofpn;
elk=zeros(edof,edof);
iopt=1;  % 1 for plane stress and 2 for plane strain
ngptXiEt=3; % # of Gauss Integration points = 3,4 or 7 for triangles
%2) zero matrices and vectors
GF=zeros(sdof,1);
GK=zeros(sdof,sdof);
GD=zeros(sdof,1);
eld=zeros(edof,1);
stress=zeros(ngptXiEt,3);
strain=zeros(ngptXiEt,3);
Bmatrix=zeros(3,edof);
Dmatrix=zeros(3,3);
nonnect=zeros(nele,nnpel);
LinG=zeros(edof,1);
fix=zeros(sdof,2);
% Reading node data [node No., XC, YC ]
for n=1:nnode
   node(n)=n;
   xc(n)=nodedata(n,2);
   yc(n)=nodedata(n,3);
end
% Reading Element Data
for i=1:nele
   element(i)=elementdata(i,1);
   nconnect(i,1)=elementdata(i,2);
   nconnect(i,2)=elementdata(i,3);
   nconnect(i,3)=elementdata(i,4);
end
% Reading Nodal Forces [node No., Fx, Fy, MZ ]
nforce=size(forcedata,1);
for i=1:nforce
   nno(i)=forcedata(i,1);     % nno = node number with force
   GF(3*nno(i)-2)=forcedata(i,2);
   GF(3*nno(i)-1)=forcedata(i,3);
   GF(3*nno(i))=forcedata(i,4);
end
```

```
% Reading fixation data
nfix=size(fixdata,1);
for i=1:nfix
   nnofix(i)=fixdata(i,1); % nnofix=# of node with specified fixation
   fix(3*nnofix(i)-2)=fixdata(i,2);
   fix(3*nnofix(i)-1)=fixdata(i,3);
   fix(3*nnofix(i))=fixdata(i,4);
end
fixeddisp2=find(fixdata(:,2)==1);
lng2=length(fixeddisp2);
for i=1:lng2
   bcdof(i)=fixdata(fixeddisp2(i),1)*3-2;
   bcval(i)=0;
end
fixeddisp3=find(fixdata(:,3)==1);
lng3=length(fixeddisp3);
for i=1 : lng3
   bcdof(i+lng2)=fixdata(fixeddisp3(i),1)*3-1;
   bcval(i+lng2)=0;
end
fixeddisp4=find(fixdata(:,4)==1);
lng4=length(fixeddisp4);
for i=1 : lng4
   bcdof(i+lng2+lng3)=fixdata(fixeddisp3(i),1)*3;
   bcval(i+lng2+lng3)=0;
end
bcdof=sort(bcdof);
% Plot of the structure
figure (1)
for n=1:nele
   x(1)= xc(nconnect(n,1));
   x(2)= xc(nconnect(n,2));
   x(3)= xc(nconnect(n,3));
   x(4)= x(1);
   y(1)= yc(nconnect(n,1));
   y(2)= yc(nconnect(n,2));
   y(3)= yc(nconnect(n,3));
   y(4)= y(1);
   plot (x,y, '-o','markersize',1.5)
   axis equal
   title('Graph of the Q3B Problem')
   hold on
end
% writing Heading and input to output file
fid1 = fopen(OutFileName,'w');
fprintf(fid1,'%s \n','--------- <<<<< Node Data >>>>> -----------');
fprintf(fid1,'%s \t %s \t\t %s \n', 'Node','[x y-Fixation, Z
rotation]','[x & y Coordinates]');
for i=1:nnode
fprintf(fid1,'%d \t %d \t %d \t %d \t %8.4f \t %8.4f \n',[i;fix(3*i-
2);fix(3*i-1);fix(3*i);xc(i);yc(i)]);
end
fprintf(fid1, '\n');
fprintf(fid1,'%s \n','------ <<<<< Element Data >>>>> ---------');
fprintf(fid1,'%s \t\t\t %s \t\t\t \n', 'Element','Nodes i-j-m
Counterclockwise');
fprintf(fid1, '\n');
for i=1:nele
   fprintf(fid1,'%d \t %d \t %d \t %d  \n', [i; nconnect(i,1);
nconnect(i,2); nconnect(i,3)]);
end
```

```
fprintf(fid1, '\n');
% sampling Points and weights
format long
[points,weights]=Gquad(ngptXiEt);
[Dmatrix]=elmatmtx(emod,noo);
for n=1:nele
   for i=1:nnpel
      nd(i)=nconnect(n,i);
      xcoord(i)=xc(nd(i));
      ycoord(i)=yc(nd(i));
   end
   C=trans(xcoord, ycoord);
   elk=zeros(edof,edof);
% numerical Intergrtation
   for i=1:ngptXiEt
      Xival=points(i,1);
      Etval=points(i,2);
      wtxy=weights(i);
      [Ns3,pNpXi3,pNpEt3]=ShapeDer3(Xival,Etval);
      [detJac2,Bmatrix]= JacobBmatrix3 (nnpel,edof, pNpXi3,pNpEt3,
      Xival, Etval, xcoord,ycoord);
      elk=elk+Bmatrix'*Dmatrix*Bmatrix*detJac2*th*wtxy/2;
   end
   elk= transpose(inv(C))*elk*inv(C);
   format bank
   LinG=feeldof(nd,nnpel,ndofpn);
   GK=assemble(GK,elk,LinG);
end      % end of loop for elk and GK
%-----------------------------------------------------------
%apply boundary conditions
[GK,GF]=applybc(GK,GF,bcdof,bcval);
%-----------------------------------------------------------
%Solve for Global Displacements, GD
GD=inv(GK)*GF;
%-----------------------------------------------------------
fprintf(fid1,'%s \n','------ <<<<< Nodal Displacements >>>>> ---');
fprintf(fid1,'%s  \t %s \t %s \t %s  \t %s  \t %s \n', 'Node','--u--
','--v--','--RZ--', '--x--','--y--');
for i=1:nnode
   fprintf(fid1,'%d \t %8.5f \t %8.5f \t %8.5f \t %8.5f \t %8.5f \t
\n',[i;GD(3*i-2);GD(3*i-1);GD(3*i); xc(i); yc(i)]);
end
fprintf(fid1, '\n');
fprintf(fid1,'%s \n','-------- <<<<< Element Stresses>>>>> -------');
fprintf(fid1,'%s \t %s \t %s \t %s \t %s \t %s \n','Element',...
   'Sigma-x','Sigma-y','Shear-xy','xlocation','ylcoation');
%-----------------------------------------------------------
% Element stress
kk = 1;
for n=1:nele
   for i=1:nnpel
      nd(i)=nconnect(n,i);
      xcoord(i)=xc(nd(i));
      ycoord(i)=yc(nd(i));
   end
   C=trans(xcoord, ycoord);
   LinG=feeldof(nd,nnpel,ndofpn);
   intp=0;
   for i=1:edof
      eld(i)=GD(LinG(i));
   end
```

```
   Xivalues = [0 1 0];
   Etavalues =[0 0 1];
   % Values of Shape Functions at NODES
   npoints = length(Xivalues);
   for intp=1:npoints
      Xival=Xivalues (intp);
      Etval=Etavalues (intp);
      intp=intp+1;
      [Ns3,pNpXi3,pNpEt3]=ShapeDer3(Xival,Etval);
      [detJac2,Bmatrix]= JacobBmatrix3 (nnpel,edof, pNpXi3,pNpEt3,
      Xival,Etval, xcoord,ycoord);
      % Compute and store stress and strain
      elstrain=Bmatrix* inv(C)* eld;
      elstress=Dmatrix * Bmatrix* inv(C)* eld;
      for i=1:3
         strain(intp,i)=elstrain(i);
         stress(intp,i)=elstress(i);
         stressx(kk,1) = stress(intp,1);
      end
      xlocation=Ns3* xcoord';
      ylocation=Ns3* ycoord';
      xcoor(kk)  = xlocation;
      ycoor(kk)  = ylocation;
      kk=kk+1;
      fprintf(fid1,'%d \t %10.2f \t %10.2f \t %10.2f \t %10.4f \t
      %10.4f \t\t \n', [n;stress(intp,1); stress(intp,2);
      stress(intp,3); xlocation; ylocation]);
   end
end
fclose(fid1);
%------------------------------------------------------------
%plot of the structure showing bending stress
figure (2)
nl=length (xcoor);
xlin = linspace(min(xcoor),max(xcoor),nl);
ylin = linspace(min(ycoor),max(ycoor),nl);
[X,Y] = meshgrid(xlin,ylin);
Z = griddata(xcoor,ycoor,stressx,X,Y,'cubic');
surf(X,Y,Z);
axis equal
shading interp;
colormap;
%colorbar;
view(0,90)
%------------------------------------------------------------
function [Ns3,pNpXi3,pNpEt3]=ShapeDer3(Xival,Etval);
%Shape functions
Ns3(1)=1-Xival-Etval;
Ns3(2)=Xival;
Ns3(3)=Etval;
%Derivatives
pNpXi3(1)=-1;
pNpXi3(2)=1;
pNpXi3(3)=0;
pNpEt3(1)=-1;
pNpEt3(2)=0;
pNpEt3(3)=1;
%------------------------------------------------------------
function
[detJac2,Bmatrix]=JacobBmatrix3(nnpel,edof,pNpXi,pNpEt,Xival,Etval,
xcoord,ycoord);
```

```
Jac2=zeros(2,2);
for i=1:nnpel
   Jac2(1,1)=Jac2(1,1)+pNpXi(i)*xcoord(i);
   Jac2(1,2)=Jac2(1,2)+pNpXi(i)*ycoord(i);
   Jac2(2,1)=Jac2(2,1)+pNpEt(i)*xcoord(i);
   Jac2(2,2)=Jac2(2,2)+pNpEt(i)*ycoord(i);
end
detJac2=det(Jac2);
Ns3(1)=1-Xival-Etval;
Ns3(2)=Xival;
Ns3(3)=Etval;
x = Ns3*xcoord';      % x = N1 x1 + N2 x2 + N3 x3
y = Ns3*ycoord';      % y = N1 y1 + N2 y2 + N3 y3
% Bmatrix BY SALAH- (SBTE)
Bmatrix =[
   0,0,0,1,y,0,0,0,y^2/4;
   0,0,0,0,0,1,x,0,-x^2/4;
   0,0,0,0,-x^2/4,0,y^2/4,1,(x+y)];        %[3x9]
%------------------------------------------------

% Function Gquad
function [points,weights]=Gquad(ngptXiEt)
% ngpt      = number of Gauss Sampling / Integration Points
% points = vector containing locations of integration points 1-D
% weights   = vector containing locations weighting factors 1-D
% initalization
points=zeros(ngptXiEt,2);
weights=zeros(ngptXiEt,1);
% use long format to capture maximum significant figures Important
format long
% find corresponding integration points
% Intialization
% find corresponding integration points
format long
if ngptXiEt ==3
   points(1,1)=0.1666666666667;
   points(1,2)=0.1666666666667;
   points(2,1)=0.6666666666667;
   points(2,2)=0.1666666666667;
   points(3,1)=0.1666666666667;
   points(3,2)=0.6666666666667;
   weights(1)=1/3;
   weights(2)=1/3;
   weights(3)=1/3;
elseif ngptXiEt ==4
   points(1,1)=1/3;
   points(1,2)=1/3;
   points(2,1)=0.6;
   points(2,2)=0.2;
   points(3,1)=0.2;
   points(3,2)=0.6;
   points(4,1)=0.2;
   points(4,2)=0.2;
   weights(1)=-0.5625;
   weights(2)=0.520833;
   weights(3)=0.520833;
   weights(4)=0.520833;
end
```

```
%-------------------------------------------------------
function[Dmatrix]=elmatmtx(emod,noo)
Dmatrix=[1,noo,0;...
         noo,1,0;...
         0,0,0.5*(1-noo)]*emod/(1-noo^2);

%-------------------------------------------------------
function [LinG]=feeldof(nd,nnpel,ndofpn)
k=0;
for i =1:nnpel
   start=(nd(i)-1)*ndofpn;
   for j= 1:ndofpn
      k=k+1;
      LinG(k)=start+j;
   end
end
%-------------------------------------------------------
function [GK]=assemble(GK,elk,LinG)
edof=length(LinG);
for i=1:edof
   ii=LinG(i);
   for j=1:edof
      jj=LinG(j);
      GK(ii,jj)=GK(ii,jj)+elk(i,j);
   end
end
%-------------------------------------------------------
function [GK,GF]=applybc(GK,GF,bcdof,bcval)
n=length(bcdof);
sdof=size(GK);
for i=1:n
   c=bcdof(i);
   for j=1:sdof
      GK(c,j)=0;
      GK(j,c)=0;
   end
   GK(c,c)=1;
   GF(c)=bcval(i);
end
%-------------------------------------------------------
function [C]= trans(xcoord, ycoord);
C=zeros(9,9);
for j =1:3
   i1 = 3*j-2;
   i2 = 3*j-1;
   i3 = 3*j;
   x=xcoord(j);
   y=ycoord(j);
   % Coordinate Transformation matrix : BY SALAH-TR4C
   C(i1,:)=[1,0,-y,x,x*y,0,(y^3/12-y^2/2),y/2,(x*y^2/4+y^2/2)];
   C(i2,:)=[0,1,x,0,(-x^2/2-x^3/12),y,x*y,x/2,(-y*x^2/4+x^2/2)];
   C(i3,:)=[0,0,1,0,(-x-x^2/8),0,(y-y^2/8),0,(x-y-x*y)/2];
end
```

A.4 MATLAB CODE FOR THE STRAIN BASED RECTANGULAR ELEMENT WITH IN-PLANE ROTATION (SBREIR)

```
function [] = q4b
% Strain Based Rectangular Element with In-Plane Rotation (SBREIR)
% No. of Nodes = 4
% No. of DOF per Node = 3
% Total No. of DOF Node = 4x3=12
% By Salah
% 1) control parameters
InFileName = input('Name of Input File (without extension): ','s' );
OutFileName = input('Name of Output File (with .m extension): ','s');
%Data input
eval (InFileName);
nnpel = 4;
ndofpn=3;
edof=nnpel*ndofpn;
sdof=nnode*ndofpn;
elk=zeros(edof,edof);
C=zeros(edof,edof);
iopt=1;  % 1 for plane stress and 2 for plane strain
ngptXi=4;   %  integ. points give accurate numerical integration
ngptEt=4;   %  integ. points give accurate numerical integration
ngptXiEt=ngptXi*ngptEt;
%2) zero matrices and vectors
GF=zeros(sdof,1);
GK=zeros(sdof,sdof);
GD=zeros(sdof,1);
eld=zeros(edof,1);
stress=zeros(ngptXiEt,3);
strain=zeros(ngptXiEt,3);
Bmatrix=zeros(3,edof);
Dmatrix=zeros(3,3);
nonnect=zeros(nele,nnpel);
LinG=zeros(edof,1);
fix=zeros(sdof);
% Reading node data [node No., XC, YC ]
for n=1:nnode
   node(n)=n;
   xc(n)=nodedata(n,2);
   yc(n)=nodedata(n,3);
end
% Reading Element Data
for i=1:nele
   element(i)=elementdata(i,1);
   nconnect(i,1)=elementdata(i,2);
   nconnect(i,2)=elementdata(i,3);
   nconnect(i,3)=elementdata(i,4);
   nconnect(i,4)=elementdata(i,5);
end
% Reading Nodal Forces [node No., Fx, Fy ]
nforce=size(forcedata,1);
for i=1:nforce
   nno(i)=forcedata(i,1);     % nno = node number with force
   GF(3*nno(i)-2)=forcedata(i,2);
   GF(3*nno(i)-1)=forcedata(i,3);
```

```
   GF(3*nno(i))=forcedata(i,4);
end
% Reading fixation data
nfix=size(fixdata,1);
for i=1:nfix
   nnofix(i)=fixdata(i,1);    % nnofix =  number of node with
specified fixation
   fix(3*nnofix(i)-2)=fixdata(i,2);
   fix(3*nnofix(i)-1)=fixdata(i,3);
   fix(3*nnofix(i))=fixdata(i,4);
end
fixeddisp2=find(fixdata(:,2)==1);
lng2=length(fixeddisp2);
for i=1:lng2
   bcdof(i)=fixdata(fixeddisp2(i),1)*3-2;
   bcval(i)=0;
end
fixeddisp3=find(fixdata(:,3)==1);
lng3=length(fixeddisp3);
for i=1 : lng3
   bcdof(i+lng2)=fixdata(fixeddisp3(i),1)*3-1;
   bcval(i+lng2)=0;
end
fixeddisp4=find(fixdata(:,4)==1);
lng4=length(fixeddisp4);
for i=1 : lng4
   bcdof(i+lng2+lng3)=fixdata(fixeddisp3(i),1)*3;
   bcval(i+lng2+lng3)=0;
end
bcdof=sort(bcdof);
% Plot of the structure
figure (1)
for n=1:nele
   x(1)= xc(nconnect(n,1));
   x(2)= xc(nconnect(n,2));
   x(3)= xc(nconnect(n,3));
   x(4)= xc(nconnect(n,4));
   x(5)= x(1);
   y(1)= yc(nconnect(n,1));
   y(2)= yc(nconnect(n,2));
   y(3)= yc(nconnect(n,3));
   y(4)= yc(nconnect(n,4));
   y(5)= y(1);
   plot (x,y, '-o','markersize',1.5)
   axis equal
   title('Graph of the Q4b Problem')
   hold on
end
% writing results to output file
fid1 = fopen(OutFileName,'w');
fprintf(fid1,'%s \n','------- <<<<< Node Data >>>>> ------------');
fprintf(fid1,'%s \t %s \t\t %s \n', 'Node','[x y-Fixation, Z
rotation]','[x & y Coordinates]');
for i=1:nnode
   fprintf(fid1,'%d \t %d \t %d \t %d \t %8.4f \t %8.4f
\n',[i;fix(3*i-2);fix(3*i-1);fix(3*i);xc(i);yc(i)]);
end
fprintf(fid1, '\n');
fprintf(fid1,'%s \n','------- <<<<< Element Data >>>>> ---------');
fprintf(fid1,'%s \t\t\t %s \t\t\t \n', 'Element','Nodes i-j-m-n
Counterclockwise');
```

```
fprintf(fid1, '\n');
for i=1:nele
   fprintf(fid1,'%d \t %d \t %d \t %d \t %d
\n',[i;nconnect(i,1);nconnect(i,2);nconnect(i,3);nconnect(i,4)]);
 end
fprintf(fid1, '\n');

% sampling Points and weights
[point2,weight2]=Gquad2(ngptXi,ngptEt);
Dmatrix=elmatmtx(emod,noo);
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
for n=1:nele
   for i=1:nnpel
      nd(i)=nconnect(n,i);
      xcoord(i)=xc(nd(i));
      ycoord(i)=yc(nd(i));
   end
   C=trans(xcoord, ycoord);
   elk=zeros(edof,edof);
   format long
   for ix=1:ngptXi
      Xival=point2(ix,1);
      wtx=weight2(ix,1);
      for iy=1:ngptEt
         Etval=point2(iy,2);
         wty=weight2(iy,2);
         [Ns,pNpXi,pNpEt]=ShapeDerQ4(Xival,Etval);
         [detJac2,Bmatrix]= JacobBmatrixQ4(nnpel,edof, pNpXi,pNpEt,
         Xival, Etval, xcoord, ycoord);
         elk=elk+Bmatrix'*Dmatrix*Bmatrix*detJac2*th*wtx*wty;
      end
   end
   elk= transpose(inv(C))*elk*inv(C);
   format bank
   LinG=feeldof(nd,nnpel,ndofpn);
   GK=assemble(GK,elk,LinG);
end      % end of loop for elk and GK
%------------------------------------------------------------
%apply boundary conditons
[GK,GF]=applybc(GK,GF,bcdof,bcval);
%------------------------------------------------------------
%Solve for Global Displacements, GD
GD=inv(GK)*GF;
%------------------------------------------------------------
fprintf(fid1,'%s \n','----- <<<<< Nodal Displacements >>>>> ----');
fprintf(fid1,'%s  \t %s \t %s \t %s  \t %s  \t %s \n', 'Node','--u--
', '--v--','--RZ--', '--x--','--y--');
for i=1:nnode
   fprintf(fid1,'%d \t %8.5f \t %8.5f \t %8.5f \t %8.5f \t %8.5f \t
   \n', [i;GD(3*i-2); GD(3*i-1); GD(3*i); xc(i); yc(i)]);
end
fprintf(fid1, '\n');
fprintf(fid1,'%s \n','--------- <<<<< Element Stresses>>>>> ----');
fprintf(fid1,'%s \t %s \t %s \t %s \t %s \t %s \n','Element',...
   'Sigma-x','Sigma-y','Shear-xy','xlocation','ylcoation');
%------------------------------------------------------------
% Element stress
kk = 1;
for n=1:nele
   for i=1:nnpel
      nd(i)=nconnect(n,i);
```

```
      xcoord(i)=xc(nd(i));
      ycoord(i)=yc(nd(i));
   end
   C=trans(xcoord, ycoord);
   LinG=feeldof(nd,nnpel,ndofpn);
   for i=1:edof
      eld(i)=GD(LinG(i));
   end
   Xivalues = [-1  1 1 -1 ];
   Etavalues =[-1 -1 1  1 ];
   % Values of Shape Functions at NODES
   npoints = length(Xivalues);
   for intp=1:npoints
      Xival=Xivalues (intp);
      Etval=Etavalues (intp);
      [Ns,pNpXi,pNpEt]=ShapeDerQ4(Xival,Etval);
      [detJac2,Bmatrix]= JacobBmatrixQ4(nnpel, edof,pNpXi, pNpEt,
      Xival, Etval, xcoord,ycoord);
      % Compute and store stress and strain
      elstrain=Bmatrix* inv(C)* eld;
      elstress=Dmatrix * Bmatrix* inv(C)* eld;
      for i=1:3
         strain(intp,i)=elstrain(i);
         stress(intp,i)=elstress(i);
         stressx(kk,1) = stress(intp,1);
      end
      xlocation=Ns* xcoord';
      ylocation=Ns* ycoord';
      xcoor(kk)  = xlocation;
      ycoor(kk)  = ylocation;
      kk=kk+1;
      format short
      fprintf(fid1,'%d \t %10.2f \t %10.2f \t %10.2f \t %10.4f \t
      %10.4f \t\t \n',...
      [n;stress(intp,1); stress(intp,2); stress(intp,3); xlocation;
      ylocation]);
   end
end
fclose(fid1);
%------------------------------------------------------------
%plot of the structure showing bending stress
figure (2)
nl=length (xcoor);
xlin = linspace(min(xcoor),max(xcoor),nl);
ylin = linspace(min(ycoor),max(ycoor),nl);
[X,Y] = meshgrid(xlin,ylin);
Z = griddata(xcoor,ycoor,stressx,X,Y,'cubic');
surf(X,Y,Z);
axis equal
shading interp;
colormap;
%colorbar;
view(0,90)
%-----------------------------------------------------------
function [NsQ4,pNpXiQ4,pNpEtQ4]=ShapeDerQ4(Xival,Etval);
%Shape functions
NsQ4(1)=0.25*(1-Xival)*(1-Etval);
NsQ4(2)=0.25*(1+Xival)*(1-Etval);
NsQ4(3)=0.25*(1+Xival)*(1+Etval);
NsQ4(4)=0.25*(1-Xival)*(1+Etval);
```

```
%Derivatives
pNpXiQ4(1)=-0.25*(1-Etval);
pNpXiQ4(2)=0.25*(1-Etval);
pNpXiQ4(3)=0.25*(1+Etval);
pNpXiQ4(4)=-0.25*(1+Etval);

pNpEtQ4(1)=-0.25*(1-Xival);
pNpEtQ4(2)=-0.25*(1+Xival);
pNpEtQ4(3)=0.25*(1+Xival);
pNpEtQ4(4)=0.25*(1-Xival);

%-----------------------------------------------------------
function
[detJac2,Bmatrix]=JacobBmatrixQ4(nnpel,edof,pNpXi,pNpEt,Xival,Etval,
xcoord,ycoord);
Jac2=zeros(2,2);
for i=1:nnpel
   Jac2(1,1)=Jac2(1,1)+pNpXi(i)*xcoord(i);
   Jac2(1,2)=Jac2(1,2)+pNpXi(i)*ycoord(i);
   Jac2(2,1)=Jac2(2,1)+pNpEt(i)*xcoord(i);
   Jac2(2,2)=Jac2(2,2)+pNpEt(i)*ycoord(i);
end
detJac2=det(Jac2);
NsQ4(1)=0.25*(1-Xival)*(1-Etval);
NsQ4(2)=0.25*(1+Xival)*(1-Etval);
NsQ4(3)=0.25*(1+Xival)*(1+Etval);
NsQ4(4)=0.25*(1-Xival)*(1+Etval);
x = NsQ4*xcoord';      % x = N1 x1 + N2 x2 + N3 x3 + N4 x4
y = NsQ4*ycoord';      % y = N1 y1 + N2 y2 + N3 y3 + N4 y4
%Bmatrix:  (SBREIR,  BY SALAH)
Bmatrix =[  0,0,0,1,y,0,0,y^2,x*y^3,0,0,0;
            0,0,0,0,0,1,x,-x^2,-y*x^3,0,0,0;
            0,0,0,0,-x^2/2,0,-y^2/2,0,0,1,x,y]    ; 3x12
%-------------------------------------------------
% Function Gquad1
function [point1,weight1]=Gquad1(ngpt)
% ngpt      = number of Gauss Sampling / Integration Points
% point1 = vector containing locations of integration points 1-D
% weight1   = vector containing locations weighting factors 1-D

% initalization
point1=zeros(ngpt,1);
weight1=zeros(ngpt,1);
% use long format to capture maximum significant figures Important
format long
% find corresponding integration points
if ngpt==1
   point1(1)=0;
   weight1(1)=0;
elseif ngpt==2
   point1(1)=-0.577350269189626;
   point1(2)=-point1(1);
   weight1(1)=1;
   weight1(2)=1;
elseif ngpt==3
   point1(1)=-0.774596669241483;
   point1(2)=0.0;
   point1(3)=-point1(1);
   weight1(1)=0.555555555555556;
   weight1(2)=0.888888888888889;
   weight1(3)=weight1(1);
```

```
elseif ngpt==4
   point1(1)=-0.861136311594053;
   point1(2)=-0.339981043584856;
   point1(3)=-point1(2);
   point1(4)=-point1(1);
   weight1(1)=0.347854845137454;
   weight1(2)=0.652145154862546;
   weight1(3)=weight1(2);
   weight1(4)=weight1(1);
elseif ngpt==5
   point1(1)=-0.906179845938664;
   point1(2)=-0.538469310105683;
   point1(3)=0;
   point1(4)=-point1(2);
   point1(5)=-point1(1);
   weight1(1)=0.236926885056189;
   weight1(2)=0.478628670499366;
   weight1(3)=0.568888888888889;
   weight1(4)=weight1(2);
   weight1(5)=weight1(1);
end
%-----------------------------------------------------------
%function Gquad 2
function [point2,weight2]=Gquad2(ngptx,ngpty)
% ngptx= # of Gauss Sampling / Integration Points in the x-direction
% ngpty= # of Gauss Sampling / Integration Points in the y-direction
% point2 = vector containing locations of integration points 2-D
% weight2   = vector containing locations weighting factors 2-D
if ngptx > ngpty
   ngpt=ngptx;
else
   ngpt=ngpty;
end
% Intialization
point2=zeros(ngpt,2);
weight2=zeros(ngpt,2);
% find corresponding integration points
[pointx,weightx]=Gquad1(ngptx);
[pointy,weighty]=Gquad1(ngpty);
% store the obtained vectors in a 2-D vector
for ix=1:ngptx
   point2(ix,1)=pointx(ix);
   weight2(ix,1)=weightx(ix);
end
for iy=1:ngpty
   point2(iy,2)=pointy(iy);
   weight2(iy,2)=weighty(iy);
end
%-------------------------------------------------------
function[Dmatrix]=elmatmtx(emod,noo)
Dmatrix=[1,noo,0;...
      noo,1,0;...
      0,0,0.5*(1-noo)]*emod/(1-noo^2);
%-------------------------------------------------------
function [LinG]=feeldof(nd,nnpel,ndofpn)
k=0;
for i =1:nnpel
   start=(nd(i)-1)*ndofpn;
   for j= 1:ndofpn
      k=k+1;
      LinG(k)=start+j;
```

```
   end
end
%-------------------------------------------------------
function [GK]=assemble(GK,elk,LinG)
edof=length(LinG);
for i=1:edof
   ii=LinG(i);
   for j=1:edof
      jj=LinG(j);
      GK(ii,jj)=GK(ii,jj)+elk(i,j);
   end
end
%-------------------------------------------------------
function [GK,GF]=applybc(GK,GF,bcdof,bcval)
n=length(bcdof);
sdof=size(GK);
for i=1:n
   c=bcdof(i);
   for j=1:sdof
      GK(c,j)=0;
      GK(j,c)=0;
   end
   GK(c,c)=1;
   GF(c)=bcval(i);
end
%-------------------------------------------------------
function [C]= trans(xcoord, ycoord);
C=zeros(12,12);
for j =1:4
   i1 = 3*j-2;
   i2 = 3*j-1;
   i3 = 3*j;
   x=xcoord(j);
   y=ycoord(j);

   C(i1,:)= [1,0,-y,x,x*y,0,(-y^3/6-y^2/2),x*y^2,(x^2*y^3)/2,y/2, 0,
            (y^2)/2];
   C(i2,:)= [0,1,x,0,(-x^3/6-x^2/2),y,x*y,-y*x^2,-(x^3*y^2)/2, x/2,
            (x^2)/2,0];
   C(i3,:)=[0,0,1,0,(-x-x^2/4),0,(y+y^2/4),-2*x*y,-(3*x^2*y^2)/2,0,
            x/2, -y/2];
end
```

A4.1 Sample Input File for New Strain Based Rectangular Element

```
% Input for Rectangular Strain Based Element
% Problem: Deep Cantilever Beam Loaded at the free end
nele = 40;          %Nele = Total number of elements
nnode= 55;          %Nnode = Total number of nodes in the structure
% Node Data:        % Enter data for each node here: Node number, X&Y
coordinates
nodedata =[
     1     0     0;
     2     1     0;
     3     2     0;
     4     3     0;
     5     4     0;
     6     5     0;
     7     6     0;
     8     7     0;
     9     8     0;
    10     9     0;
    11    10     0;
    12     0     1;
    13     1     1;
    14     2     1;
    15     3     1;
    16     4     1;
    17     5     1;
    18     6     1;
    19     7     1;
    20     8     1;
    21     9     1;
    22    10     1;
    23     0     2;
    24     1     2;
    25     2     2;
    26     3     2;
    27     4     2;
    28     5     2;
    29     6     2;
    30     7     2;
    31     8     2;
    32     9     2;
    33    10     2;
    34     0     3;
    35     1     3;
    36     2     3;
    37     3     3;
    38     4     3;
    39     5     3;
    40     6     3;
    41     7     3;
    42     8     3;
    43     9     3;
    44    10     3;
    45     0     4;
    46     1     4;
    47     2     4;
    48     3     4;
    49     4     4;
    50     5     4;
```

```
    51      6      4;
    52      7      4;
    53      8      4;
    54      9      4;
    55     10      4];

% Element Data: Enter data for each Element:
% Element Number and four nodes (counterclockwise)
elementdata =[
     1      1      2     13     12;
     2      2      3     14     13;
     3      3      4     15     14;
     4      4      5     16     15;
     5      5      6     17     16;
     6      6      7     18     17;
     7      7      8     19     18;
     8      8      9     20     19;
     9      9     10     21     20;
    10     10     11     22     21;
    11     12     13     24     23;
    12     13     14     25     24;
    13     14     15     26     25;
    14     15     16     27     26;
    15     16     17     28     27;
    16     17     18     29     28;
    17     18     19     30     29;
    18     19     20     31     30;
    19     20     21     32     31;
    20     21     22     33     32;
    21     23     24     35     34;
    22     24     25     36     35;
    23     25     26     37     36;
    24     26     27     38     37;
    25     27     28     39     38;
    26     28     29     40     39;
    27     29     30     41     40;
    28     30     31     42     41;
    29     31     32     43     42;
    30     32     33     44     43;
    31     34     35     46     45;
    32     35     36     47     46;
    33     36     37     48     47;
    34     37     38     49     48;
    35     38     39     50     49;
    36     39     40     51     50;
    37     40     41     52     51;
    38     41     42     53     52;
    39     42     43     54     53;
    40     43     44     55     54];
% Element E-modulus and Poisson’s ratio noo and thickness
emod= 100000; %KN/m2 NOTE: the displacement will be in millimeters
noo= 0.2;
th =0.0625;

% Applied Forces Data:  Node Number, Fx, Fy, Mz
forcedata= [
   1, 0,-100/8,0;
   12,0,-100/4,0;
   23,0,-100/4,0;
   34,0,-100/4,0;
   45,0,-100/8,0 ];
```

```
% Nodal Fixation Data: % Node Number, X-fixation, Y-fixation and Z
rotation
% (1 fixed & 0 not fixed)

fixdata= [
   11,1,1,1;
   22,1,1,1;
   33,1,1,1;
   44,1,1,1;
   55,1,1,1 ];
```

A4.2 Sample Output File for New Strain Based Rectangular Element

```
% Input for Rectangular Strain Based Element with In-Plane Rotation
% Problem: Deep Cantilever Beam Loaded at the free end
```

---------------- <<<<< Nodal Displacements >>>>> -------

Node	--u--	--v--	--RZ--	--x--	--y--
1	0.3039	-1.1007	0.1661	0.0000	0.0000
2	0.2985	-0.9365	0.1569	1.0000	0.0000
3	0.2887	-0.7813	0.1517	2.0000	0.0000
---	---	---	---	---	---
---	---	---	---	---	---
---	---	---	---	---	---
22	0.0000	0.0000	0.0000	10.0000	1.0000
23	0.0001	-1.0937	0.1499	0.0000	2.0000
24	0.0001	-0.9357	0.1502	1.0000	2.0000
---	---	---	---	---	---
---	---	---	---	---	---
---	---	---	---	---	---
54	-0.0572	-0.0269	0.0340	9.0000	4.0000
55	0.0000	0.0000	0.0000	10.0000	4.0000

---------------- <<<<< Element Stresses>>>>> ----------

Element	Sigma-x	Sigma-y	Shear-xy	xlocation	ylcoation
1	-468.53	367.59	129.24	0.00	0.00
1	-525.42	83.14	207.25	0.50	0.00
1	-499.03	215.05	182.79	1.00	0.00
---	---	---	---	---	---
---	---	---	---	---	---
3	-725.44	115	431.1	2.50	1.00
3	-738.07	41.63	408.61	2.00	1.00
---	---	---	---	---	---
---	---	---	---	---	---
35	2753.26	-31.96	62.38	5.00	4.00
36	3258.37	106.95	63.44	5.00	4.00