Geometrically Non-linear Finite Element Analysis of Plane Stress Structures Using MATLAB Programme

A Thesis submitted to Department of Structural Engineering College of Engineering in partial fulfillment of the requirements of the degree of Master of Science in Civil Engineering (Structures)

Prepared by:

Abdullatif Yousif Mohammed Adam
(B.Sc (Hon), Nyala University, 2011)

Supervisor:

Prof/Abd Elrahman Elzubeir Mohamed

March 2017
بسم الله الرحمن الرحيم

(وَإِذْ يَرْفَعُ إِبْرَاهِيمُ الْقَوَاعِدَ مِنَ الْبَيْتِ وَإِسْمَاعِيلُ رَبِّنَا تَقَبَّلَ مِنَّا ۖ إِنَّكَ أَنتَ السَّمِيعُ العَلِيمُ (127))

صدق الله العظيم...

(سورة البقرة)
Dedication

To the meaning of love and compassion …
To the mystery of existence and the meaning of life …

To my Mather
My sisters….
I dedicate this work
ACKNOWLEDGEMENT

First all praise and thanks are to Allah, who helped me to complete this work.

I would like to express sincere appreciation and gratitude to my Supervisor professor. Dr. A/Elrahman Elzubeir Mohamed, for his teaching, guidance, patience, and above all as supervisor.

I also want to thanks and my gratitude to teachers, colleagues, friend, and all who contributed to the completion of this research.

My infinite gratitude to my small family, my mother, my sisters

Finally, I would like to thank Nyala University foundation.
Abstract

In this research Total Lagrangian formulation for geometric nonlinear plane stress problems was developed. The formulation is based on Green strains and 2nd Piola-Kirchoff stresses. The formulation was applied on two- dimensional elasticity using 4-node plane finite element. The formulation was implemented as a finite element program using MATLAB (2010b). The program was developed for linear and nonlinear analysis of plane stress structure subjected to different types of loading.

The solution of nonlinear equilibrium equation was obtained by incremental method with Newton-Raphson approach. The program was applied to obtain nodal displacements, direct stresses and shear stresses at integration points of element based on Green strains and 2nd Piola-Kirchoff stresses.

The accuracy of results was demonstrated by using three numerical examples for linear analysis. The results were in very good agreement when compared with results from references and known exact solutions. It was, also observed from Graphs and Tables that the results show high accuracy with mesh refinement and increment applied load.

For the nonlinear analysis, the resulting displacements show good agreement when compared with known results or published papers results.
تجريد

في هذا البحث تم تطوير الصيغ الهندسية اللاخطية لمنشآت إجهاد المستوى الواحد باستخدام تقنين لاجرانج الكلي (Total Lagrangian) بناءً على انفعالات قرين واجهادات بيولا كريتشوف الثانية. هذه الصيغ طبقت على المرونة ثنائية الابعاد باستخدام عنصر الأربع عقد المحدد.

تم تطبيق الصيغ بتطوير برنامج عنصر محدد باستخدام لغة البرمجة ماتلاب (2010). تم طور البرنامج للتحليل الخطي واللاخطي لمنشآت الإجهاد المستوى المعرضة لانواع مختلفة من الاعمال.

حل معادلات الانزكان اللاخطية بناءً على طريقة الالامال التزايدية وطريقة نيوتن- رابسون. طبق البرنامج للحصول على الازاحات عند العقد واجهادات المباشرة واجهادات القص عند نقاط التكامل (integration point). بناءً على انفعالات قرين واجهادات بيولا كريتشوف الثانية.

وقد نلاحظ أيضاً من الجداول والمخططات، أن هناك تزايد في دقة النتائج مع زيادة عدد العناصر والمحدود الاعمال المطبقة.

تم التحقق من دقة النتائج باستخدام ثلاثة أمثلة عددة، التحليل الخطي اعطت النتائج المتصلة عليها توافقاً جيداً جداً مقارنة بالحلول الدقيقة المعروفة والنشرة. أما في حالة التحليل اللاخطي فقد أعطت النتائج توافقاً جيداً مع النتائج المنشورة في عدة أوراق علمية.
Contents

A. Verse I
B. Dedication II
C. Acknowledgements III
D. Abstract in Arabic IV
E. Abstract V
F. Table of contents VI
G. List of tables VII
H. List of figures VIII
I. List of symbols IX

Chapter one: General introduction

1.1 General introduction 1
1.2 Problems statement 3
1.3 Objective of research 4
1.4 Methodology 5
1.5 Outline of thesis 5

Chapter two: Literature review

2.1 Introduction to finite element analysis method 6
2.2 Using MATLAB finite element programming 7
2.3 Sources of nonlinearity in finite element analysis 8
    2.3.1 Geometric nonlinearities 8
    2.3.2 Material nonlinearities 8
    2.3.3 Force nonlinearities 8
    2.3.4 Kinematic nonlinearities 8
2.4 Total Lagrangian and Updated Lagrangian formulation 9
    2.4.1 Comparison of T.L and U.L formulation 9
2.5 Stress – Strain Measures 9
2.6 Brief description of the finite element method 11
2.7 Previous studies 12
2.8 Non-linear solution techniques 14
  2.8.1 Introduction 14
  2.8.2 Newton -Raphson method 14
  2.8.3 Modified Newton- Raphson method 15
  2.8.4 Incremental Force Method 15

Chapter three: Geometrically non-linear finite element of plain stress/strain

3.1 Introduction 17
3.2 Finite Element formulation of 4-node Isoperimetric plane stress/strain element 18
  3.2.1 Introduction 18
  3.2.2 Stress-strain relation 19
  3.2.3 Incremental formulations for geometrically nonlinear plane stress/strain Element 22
3.3 Theory method of formulation for plane stress/strain element 25
  3.3.1 Stress-strain relation 25
  3.3.2 Strain-displacement relation 25
  3.3.3 Green's Lagrangian strain 26
  3.3.4 Stiffness matrix due to Green strain 26
  3.4 Incremental equilibrium equations 30

Chapter four: Implementation and Application program and Verification

4.1 Computer program for linear and non-linear finite element for plane stress/strain using MATLAB 32
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.1.1 Introduction</td>
<td>32</td>
</tr>
<tr>
<td>4.1.2 Main program procedures</td>
<td>32</td>
</tr>
<tr>
<td>4.1.3 Program layout</td>
<td>34</td>
</tr>
<tr>
<td>4.1.4 Function explanations</td>
<td>36</td>
</tr>
<tr>
<td>4.2 Verification of formulation</td>
<td>37</td>
</tr>
<tr>
<td>4.2.1 Introduction</td>
<td>37</td>
</tr>
<tr>
<td>4.2.2 Linear analysis</td>
<td>37</td>
</tr>
<tr>
<td>4.2.2.1 Cantilever plate beam under concentrated load at the end</td>
<td>37</td>
</tr>
<tr>
<td>4.2.2.2 Cantilever plate beam under pure bending</td>
<td>44</td>
</tr>
<tr>
<td>4.2.2.3 Simple supported plate beam under udl</td>
<td>50</td>
</tr>
<tr>
<td>4.2.3 Nonlinear analysis</td>
<td>55</td>
</tr>
<tr>
<td>4.2.3.1 Clamped plate beam under concentrated load</td>
<td>55</td>
</tr>
<tr>
<td>4.2.3.2 Cantilever plate beam under pure bending</td>
<td>57</td>
</tr>
<tr>
<td>4.2.3.3 Discussion of result</td>
<td>59</td>
</tr>
<tr>
<td><strong>Chapter five: Conclusions and Recommendations</strong></td>
<td></td>
</tr>
<tr>
<td>5.1 Summary of work</td>
<td>62</td>
</tr>
<tr>
<td>5.2 Conclusions</td>
<td>62</td>
</tr>
<tr>
<td>5.3 Recommendations</td>
<td>63</td>
</tr>
<tr>
<td><strong>References</strong></td>
<td>64</td>
</tr>
<tr>
<td><strong>Appendix A</strong></td>
<td>66</td>
</tr>
<tr>
<td><strong>Appendix B</strong></td>
<td>72</td>
</tr>
</tbody>
</table>
# List of Tables

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Table (4.1)</td>
<td>Vertical displacement at free end and mesh effect</td>
<td>42</td>
</tr>
<tr>
<td>Table (4.2)</td>
<td>Vertical displacement at the free end</td>
<td>43</td>
</tr>
<tr>
<td>Table (4.3)</td>
<td>Vertical displacement at point A (200EL, 5INC)</td>
<td>44</td>
</tr>
<tr>
<td>Table (4.4)</td>
<td>Vertical displacement at point A (200EL, 10INC)</td>
<td>44</td>
</tr>
<tr>
<td>Table (4.5)</td>
<td>Vertical displacement at point A (200EL, 15INC)</td>
<td>44</td>
</tr>
<tr>
<td>Table (4.6)</td>
<td>Deflected shape along center line (100EL)</td>
<td>45</td>
</tr>
<tr>
<td>Table (4.7)</td>
<td>Stress in x-direction at integration point (INT1)</td>
<td>46</td>
</tr>
<tr>
<td>Table (4.8)</td>
<td>Stress in y-direction at integration point (INT1)</td>
<td>46</td>
</tr>
<tr>
<td>Table (4.9)</td>
<td>Shear stress at integration point (INT1)</td>
<td>47</td>
</tr>
<tr>
<td>Table (4.10)</td>
<td>Vertical Displacement at free end and with aspect ratio</td>
<td>49</td>
</tr>
<tr>
<td>Table (4.11)</td>
<td>Vertical Displacement at free end</td>
<td>49</td>
</tr>
<tr>
<td>Table (4.12)</td>
<td>Vertical displacement at free end (200EL, 4INC)</td>
<td>50</td>
</tr>
<tr>
<td>Table (4.13)</td>
<td>Vertical displacement at free end (200EL, 8INC)</td>
<td>50</td>
</tr>
<tr>
<td>Table (4.14)</td>
<td>Vertical displacement at free end (200EL, 12INC)</td>
<td>51</td>
</tr>
<tr>
<td>Table (4.15)</td>
<td>Deflected shape along center line (60EL)</td>
<td>51</td>
</tr>
<tr>
<td>Table (4.16)</td>
<td>Stress in x-direction at integration point (EL (1), INT 1)</td>
<td>52</td>
</tr>
<tr>
<td>Table (4.17)</td>
<td>Stress in y-direction at integration point (EL (1), INT 1)</td>
<td>53</td>
</tr>
<tr>
<td>Table (4.18)</td>
<td>Shear stress at integration point (EL (1), INT 1)</td>
<td>53</td>
</tr>
<tr>
<td>Table (4.19)</td>
<td>Vertical displacement at mid-span</td>
<td>55</td>
</tr>
<tr>
<td>Table (4.20)</td>
<td>Vertical displacement at mid-span different element</td>
<td>55</td>
</tr>
<tr>
<td>Table (4.21)</td>
<td>Deflected shape along center line (220EL)</td>
<td>56</td>
</tr>
<tr>
<td>Table (4.22)</td>
<td>Stress in x-direction at integration point at mid-span (INT 1)</td>
<td>57</td>
</tr>
</tbody>
</table>
Table (4.23) Shear Stress at integration point at mid-span (INT 1) 58
Table (4.24) Vertical displacement at mid-span (Clamped) 59
Table (4.25) Deflected shape along center line (Clamped) 60
Table (4.26) Vertical displacement at free end (pure bending) 62
Table (4.27) Deflected shape along center line (pure bending) 62
# List of Figures

<table>
<thead>
<tr>
<th>Figures</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fig (2.1)</td>
<td>Configuration change during deformation</td>
<td>9</td>
</tr>
<tr>
<td>Fig (2.2)</td>
<td>Newton-Raphson Method</td>
<td>14</td>
</tr>
<tr>
<td>Fig (2.3)</td>
<td>Modified Newton-Raphson Method</td>
<td>15</td>
</tr>
<tr>
<td>Fig (2.4)</td>
<td>Incremental Force method</td>
<td>16</td>
</tr>
<tr>
<td>Fig (3.1)</td>
<td>Total Lagrangian</td>
<td>17</td>
</tr>
<tr>
<td>Fig (3.2)</td>
<td>Updated Lagrangian</td>
<td>17</td>
</tr>
<tr>
<td>Fig (3.3)</td>
<td>4-node Isoperimetric element</td>
<td>18</td>
</tr>
<tr>
<td>Fig (3.4)</td>
<td>4-node Gauss Points</td>
<td>19</td>
</tr>
<tr>
<td>Fig (3.5)</td>
<td>Deformed and Undeformed Shape</td>
<td>26</td>
</tr>
<tr>
<td>Fig (4.1)</td>
<td>Main program flow chart</td>
<td>36</td>
</tr>
<tr>
<td>Fig (4.1a)</td>
<td>Flow chart of NONA2D function</td>
<td>38</td>
</tr>
<tr>
<td>Fig (4.1b)</td>
<td>Flow chart of LINA2D function</td>
<td>39</td>
</tr>
<tr>
<td>Fig (4.2)</td>
<td>Cantilever plate beam with vertical load at free end</td>
<td>42</td>
</tr>
<tr>
<td>Fig (4.3)</td>
<td>Effect of mesh and aspect ratio on accuracy of results</td>
<td>43</td>
</tr>
<tr>
<td>Fig (4.4)</td>
<td>Vertical displacement at point A</td>
<td>43</td>
</tr>
<tr>
<td>Fig (4.5)</td>
<td>Deformed shape along center line</td>
<td>45</td>
</tr>
<tr>
<td>Fig (4.6)</td>
<td>Stress in x-direction at integration point (El 1)</td>
<td>46</td>
</tr>
<tr>
<td>Fig (4.7)</td>
<td>Stress in y-direction at integration point (El 1)</td>
<td>47</td>
</tr>
<tr>
<td>Fig (4.8)</td>
<td>Shear stress at integration point (El 1)</td>
<td>47</td>
</tr>
<tr>
<td>Fig (4.9)</td>
<td>Cantilever plate under pure bending</td>
<td>48</td>
</tr>
<tr>
<td>Fig (4.10)</td>
<td>Effect of mesh and aspect ratio on accuracy of results</td>
<td>49</td>
</tr>
<tr>
<td>Fig (4.11)</td>
<td>Vertical displacement at free end at point A</td>
<td>50</td>
</tr>
<tr>
<td>Fig (4.12)</td>
<td>Deformed shape along center line</td>
<td>52</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
<td></td>
</tr>
<tr>
<td>--------</td>
<td>-------------</td>
<td></td>
</tr>
<tr>
<td>Fig (4.13)</td>
<td>Stress in x-direction at integration point (INT 1, El 1)</td>
<td></td>
</tr>
<tr>
<td>Fig (4.14)</td>
<td>Stress in y-direction at integration point (INT 1, El 1)</td>
<td></td>
</tr>
<tr>
<td>Fig (4.15)</td>
<td>Shear stress at integration point (INT 1, El 1)</td>
<td></td>
</tr>
<tr>
<td>Fig (4.16)</td>
<td>Simple supported plate beam under uniform load</td>
<td></td>
</tr>
<tr>
<td>Fig (4.17)</td>
<td>Vertical displacement at mid-span</td>
<td></td>
</tr>
<tr>
<td>Fig (4.18)</td>
<td>Deformed shape along center line</td>
<td></td>
</tr>
<tr>
<td>Fig (4.19)</td>
<td>Stress at integration point in x-direction at El (108,118)</td>
<td></td>
</tr>
<tr>
<td>Fig (4.20)</td>
<td>Shear stress at integration point at El (108,118)</td>
<td></td>
</tr>
<tr>
<td>Fig (4.21)</td>
<td>Clamped plate beam under concentrated load (Non-linear)</td>
<td></td>
</tr>
<tr>
<td>Fig (4.22)</td>
<td>Vertical displacement at mid-span (Non-linear)</td>
<td></td>
</tr>
<tr>
<td>Fig (4.23)</td>
<td>Deformed shape along center line (Non-linear)</td>
<td></td>
</tr>
<tr>
<td>Fig (4.24)</td>
<td>Cantilever plate under pure bending (Non-linear)</td>
<td></td>
</tr>
<tr>
<td>Fig (4.25)</td>
<td>Vertical displacement at free end point A (Non-linear)</td>
<td></td>
</tr>
<tr>
<td>Fig (4.26)</td>
<td>Deformed shape along center line</td>
<td></td>
</tr>
</tbody>
</table>
## List of Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$FEA$</td>
<td>Finite element analysis</td>
</tr>
<tr>
<td>$\sigma_E$</td>
<td>Engineering stress</td>
</tr>
<tr>
<td>$A_0$</td>
<td>Original area</td>
</tr>
<tr>
<td>$\sigma_c$</td>
<td>Cauchy stresses (true)</td>
</tr>
<tr>
<td>$L_0$</td>
<td>Original length</td>
</tr>
<tr>
<td>$[k]_e$</td>
<td>Element stiffness matrix</td>
</tr>
<tr>
<td>$[\delta]_e$</td>
<td>Element nodal displacement vector</td>
</tr>
<tr>
<td>$[F]_e$</td>
<td>Element Nodal force vector</td>
</tr>
<tr>
<td>$\varepsilon$</td>
<td>Green strain</td>
</tr>
<tr>
<td>$D$</td>
<td>Elastic matrix</td>
</tr>
<tr>
<td>$J$</td>
<td>Jacobian matrix</td>
</tr>
<tr>
<td>$r, s$</td>
<td>Natural coordinates</td>
</tr>
<tr>
<td>$\varepsilon_0$</td>
<td>Linear strain vector</td>
</tr>
<tr>
<td>$\varepsilon_L$</td>
<td>Nonlinear strain</td>
</tr>
<tr>
<td>$A$</td>
<td>Matrix containing displacement derivative</td>
</tr>
<tr>
<td>$\theta$</td>
<td>Vector containing displacement derivative</td>
</tr>
<tr>
<td>$a$</td>
<td>Vector of nodal displacement</td>
</tr>
<tr>
<td>$B_0$</td>
<td>Linear strain-displacement transformation matrix</td>
</tr>
</tbody>
</table>
Nonlinear strain-displacement transformation matrix

Component of deformation gradient vector

Displacement gradient vector

Residual force

Nonlinear strain-displacement transformation matrix

Displacement stiffness matrix

Large displacement stiffness matrix

Initial stress matrix

A symmetrical initial stress stiffness matrix

Total tangential stiffness matrix

Incremental displacement

Element

Integration point

Increment

uniform distribution load
Chapter one

General introduction

1.1 Introduction

The finite element method is a numerical technique for solution of differential equations. In this method all complexities of the problems like varying shape, boundary conditions and loads are maintained as they are, but the solutions obtained are approximate.

The finite element method originated as a method of stress analysis in the design of aircraft. It started as an extension of matrix method of structure analysis. Nowadays this method is used not only for the analysis of solid mechanics, but even in the analysis of fluid flow, heat transfer, electric and many other physical phenomena. In civil engineering is used this method extensively for the analysis of beams, space frames, plates, shells, foundations. Both static and dynamic problems can be handled by finite element method (Bhavikatti, 2005).

Finite element method is a powerful technique for obtaining approximate solution with good accuracy. The particular advantages of finite element method have made it the most accepted design tool and the greatest advantage of it is ability to handle truly arbitrary geometry. Probably its other most important features are the ability to deal with general boundary conditions and to include non-homogeneous and anisotropic materials, nonlinear stress-strain relations, nonlinear strain-displacement relations and complicated loading conditions (Akasha, 2009).

Nonlinear finite element analysis is an essential component of computer-aided design. Testing of prototypes is increasingly being replaced by simulation
with nonlinear finite element methods because this provides a more rapid and less expensive way to evaluate design concepts and design detail.

Both analysts and developers of nonlinear finite element programs should understand the fundamental concepts of nonlinear finite element analysis. Without an understanding of the fundamentals, a finite element program is a black box that provides simulations. However, nonlinear finite element analysis confronts the analyst with many choices and pitfalls. Without an understanding of the implication and meaning of these choices and difficulties, an analyst is at a severe disadvantage (Belytschison, et. el, 2013).

Two sources of nonlinearity exist in the analysis of solid continua, namely, material and geometric nonlinearity. The former occurs when, for whatever reason, the stress strain behavior given by the constitutive relation is nonlinear, whereas the latter is important when changes in geometry, whether large or small, have a significant effect on the load deformation behavior. the geometric nonlinearity includes deformation-dependent boundary conditions and loading.

The nonlinear strain and stress measures in definition of stress-strain relation are one of key concepts of several nonlinearities. Strain is defined as change of the shape or geometry produced by applied load. The loads are defined by the general term stress (Kim, 2014).

To practically assess the stress state on the structure, strain must be measured. There are alternative strain measures used to derive finite element equations, such as Green's strain which is associated with 2nd Piola-Kirchhoff stress, engineering strain which is associated with engineering stress and logarithmic strain which is associated with true (Cauchy) stress. The Green's strain is most common definition applied to materials used in mechanical and structural engineering problems, which are subjected to small deformations. The engineering strain is expressed as the ratio of total deformation to initial dimension of the material body in which the loads are being applied. The logarithmic strain also called natural strain or true strain is based on the final
deformed dimension and considering an incremental strain and obtained by integrating this incremental strain (Akasha and Mohamed, 2012).

The fast improvements in computer hardware technology and slashing of cost of computers have boosted this method, since the computer is the basic need for its application of this method. A number of popular brand of finite element analysis packages are now available commercially. Some of the popular packages are STAAD-PRO, GT-STRUDEL, NASTRAN and ANSYS. Using these packages one can analyze several complex structures.

MATLAB is commercial software and a trademark of The Math Works, Inc., It is an integrated programming system, and MATLAB is getting increasingly popular all fields of science and engineering.

MATLAB presently offers a nice combination of handy programming features with powerful built-in numerical capabilities. On the one hand, its m-file programming environment allows researchers to implement moderately complicated algorithms in a structured and coherent fashion. On the other hand, its built-in, numerical capabilities empower researchers to solve more difficult problems.

MATLAB is a high-level language specially designed for dealing with matrices. This makes it particularly suited for programming the finite element method. In addition, MATLAB will allow the user to focus on the finite element method by alleviating the programming burden (Khennan, 2013).

1.2 Problem statement

For many structures, the relation between the deflection and the applied load is linear when the deflection is small. This includes small strain, small displacement, and small rotation. But in case of the large deflection the relation becomes nonlinear.

If the structure system is solved using the linearity assumption, the results may end up physically inexact, because the linear analysis uses the undeformed geometry which lead to invalid result. Also many engineering applications cannot be modeled as linear system when the deformation is large.
For above reasons the study focuses to using geometrically nonlinear analysis.

The available commercial packages are only used as a closed box. That is, given data in a certain format to give results in a well organized manner. Also, difficulty in obtaining original copies of these commercial packages leads to the need for developing finite element programs that are original and that can be easily modified. This helps in having an efficient research tool (Khennan, 2013).

Minimizing the overall cost of analysis and obtaining a good result of analysis reduce risk of resulting design errors.

Since MATLAB is getting increasingly popular computer software in all fields of engineering, especially in finite element analysis and design of structure, it was used in this research.

1.3 Objectives

The objectives of the study are:

1- To use the MATLAB for programming of the plane stress/strain problems using geometrically nonlinear finite element analysis.

2- To formulate the geometric nonlinear Total Lagrangian elasticity problems using plane stress/strain isoparametric finite element based on Green's strain.

3- To develop a computer program based on the formulation.

4- To implement the program on computer using MATLAB language.

5- To verify and check the accuracy of the results obtained from application of the program by comparison with known published results.

1.4 Methodology of Research

The methodology of research as the following:

1- Finite element formulation using 4-node plane stress/strain element.

2- The formulation based on Green strain was considered. Linear and geometrically nonlinear analysis based on Green strain was adopted. The nonlinear equilibrium equations solved using the incremental with Newton-Raphson method.
3- The main computer program in MATLAB language was developed and the formulation was implemented in it. The program was designed to carry out linear analysis and geometric nonlinear analysis based on Green stress, for linear elastic isotropic material.

4- The program was applied and tested using numerical examples to obtain the displacement, direct stresses and shear stresses at integration points of the element. The results obtained were discussed, analyzed, and compared with known results.

1.5 Outlines of research

This study covers three main areas, the introduction of finite element method (FEM) and the literature review, geometrically nonlinear formulation of plane stress/strain, implementation of the formulation into MATLAB program. The contents can be summarized as:


Chapter two: presents the literature review

Chapter three: contains the general description of nonlinear method and development of the formulation of geometric nonlinear plane strain/stress finite element.

Chapter four: contains the description of the computer program

Chapter five: presents the results and discussion of results, conclusions and recommendations for further work.
Chapter two

Literature review

2.1 Introduction to Finite Element Analysis Method

The finite element method is one of the numerical methods for solving differential equations that describe many engineering problems. The finite element method (FEM), originated in the area of structural mechanics, has been extended to other areas of solid mechanics and later to other fields such as heat transfer, fluid dynamics, and electromagnetism. In fact, FEM has been recognized as a powerful tool for solving partial differential equations and integrodifferential equations, it has become the numerical method of choice in almost all engineering and applied science areas. One of the reasons for FEM’s popularity is that the method results in computer programs versatile in nature that can solve many practical problems with least amount of training (Kim, 2014).

In modern engineering design it is rare to find a project that does not require some type of finite element analysis. The greatest advantage of FEM is its ability to handle truly arbitrary geometry. Probably its next most important features are the ability to deal with general boundary conditions and to include non-homogeneous and anisotropic materials. These features mean that systems of arbitrary shape that are made up of numerous different material regions. Each material could have constant properties or the properties could vary with spatial location. To these very desirable features a large amount of freedom in prescribing the loading conditions and in the post processing of items such as the stresses and strains etc.

The classes of problems include stress analysis, heat conduction, electrical fields, magnetic fields, ideal fluid flow (Akin, 2000).
In comparison with classical methods, it's clearly seen that:

1. In classical methods exact equations are formed and exact solutions are obtained where as in finite element analysis exact equations are formed but approximate solutions are obtained.

2. Solution have been obtained for few standard cases by classical method, where as solutions can be obtained for all problems by FEM.

3. FEM can handle structures with anisotropic properties without any difficulty, in classical methods when material property is not isotropic, solution for problems become very difficulty.

4. If the structure consists of more than one material, it is difficult to use classical methods, but FEM can be used without any difficult.

5. Problems with material and geometric nonlinearities cannot be handle by classical methods. (Bhavikatti, 2005).

### 2.2 Using MATLAB in Finite Element Programming

Understanding basic program structure of finite element analysis is important for better comprehension of the FEM. MATLAB is especially convenient to write and understand FEM programs because a MATLAB program manipulates matrices and vectors with ease. MATLAB is interactive software which has been used in various areas of engineering and scientific applications, It is not a computer language in the normal sense but it does most of the work of a computer language, One attractive aspect of MATLAB is that it is relatively easy to learn, and it does not require in-depth knowledge on operational principles of computer programming like compiling and linking in most of other programming languages.

The power of MATLAB is represented by the length and simplicity of the code; Numerical calculation in MATLAB uses collections of well written scientific/mathematical subroutines as LINPACK and EIPACK. MATLAB Provides Graphical user interface (GUI) as well as three dimensional graphical animations (Young, 1997).
2.3 Sources of Nonlinearity in Structures

1- Geometric Nonlinearities

Geometric nonlinearities represent the cases when the relations among displacement quantities, whether translations and/or rotation are nonlinear. They can also occur when rotations are large, even with small deformation (Kim, 2014).

2- Material Nonlinearities

Material nonlinearity represents the case when the relation between stress and strain is not linear. This relation is often referred to as the constitutive relation. In the linear system the elastic modulus matrix [D] is constant, when the stress-strain relation cannot be represented by a constant matrix, [D].

3- Force Nonlinearities

Force nonlinearity occurs when the applied forces depend on deformations. Since force is a vector, its magnitude or direction can change according to the deformation of a structure. Force nonlinearity is often accompanied by geometric nonlinearity.

4- Kinematic Nonlinearity

Kinematic nonlinearity is also called boundary nonlinearity, as the displacement boundary conditions depend on the deformation of the structure. When the boundary conditions change as a function of displacements, both the displacements and boundary conditions are unknown.
2.4 Total Lagrangian and Updated Lagrangian Formulations

![Diagram showing configuration change during deformation]

**Fig (2.1) Configuration change during deformation**

In the total Lagrangian formulation (T.L.) refers all the static and kinematic variables to the initial configuration. The updated Lagrangian (U.L.) formulation is based on the same process as in the T.L. formulation, but it refers all static and kinematic variables to the last in equilibrium configuration.

- Comparison of T.L and U.L formulations is listed as follows:

1. In the T.L formulation, all derivatives are with respect to initial coordinates where as in U.L formulation, all derivatives are with respect to the current coordinates.

2. In the U.L formulation the calculated updated stresses represent the actual physical stresses (true stresses)

3. The same assumptions are made in the linearization and indeed almost the same finite element stiffness and force vectors are calculated

4. The linear strain displacement matrix is complicated in the total Lagrangian formulation than the Updated Lagrangian formulation because the effect of the term involving the initial displacement.

2.5 Stress and Strain Measures

The description of deformation and the measure of strain are essential parts of nonlinear finite element continuum mechanics. In general structural
components or continuum bodies will exhibit large strain when undergoing a geometrically nonlinear deformation process.

The main Stress measures are (Bathe et. el., 1975):

1- Engineering stress
   Defined in terms of the original area and the original geometry
2- Cauchy (true) stress
   Defined in terms of current area and current deformed geometry.
3- 1st Piola-Kirchhoff stress
   Defined in terms of the original area and the current deformed geometry.
4- 2nd Piola- Kirchhoff stress
   Defined in terms of the initial area and current deformed geometry. For small strain, the 2nd Piola-Kirchhoff stress can be interpreted as the true stress related to local axes that rotate with the material.

The corresponding Strain measures are:

1- Engineering strain
   \[ e_E = \frac{(L - L_0)}{L_0} \]  
   A measure preferred by structural engineers, and work conjugate to the engineering stress measure.
2- Logarithmic (natural)strain
   \[ e_L = \ln\left(\frac{L}{L_0}\right) \]  
   A measure that is incremental in form and work conjugate to the Cauchy stress measure.
   This measure is typically used for large strain analysis.
3- Green Lagrange strain
   \[ e_G = \frac{(L^2 - L_0^2)}{2L_0} \]  
   It is work conjugate to the 2nd Piola-Kirchhoff stress measure.
2.6 Brief description of the finite element method

In solid mechanics the unknown field variable is displacement. In a continuum, these unknowns are infinite. The finite element procedure reduces such unknowns to a finite number by dividing the solution region into small parts called elements and by expressing the unknown field variables in terms of assumed approximating shape functions within each element. The approximating functions are defined in terms of field variables of specified nodal points. Once these are found the field variables at any point can be found by using the interpolating shape functions.

After selecting elements nodal unknowns, the next steps in finite element analysis is to determine element properties for each element. The element stiffness matrix and load vector are given mathematically, by the relationship:

\[ [K]_e \{\delta\}_e = \{F\}_e \]  \hspace{1cm} (2.4)

Where
\[ [K]_e \] is the element stiffness matrix
\[ \{\delta\}_e \] is element nodal displacement vector and \[ \{F\}_e \] is element nodal force vector.

The element of stiffness matrix \( k_{ij} \) represents the force in coordinate direction, I, due to a unit displacement in coordinate direction j. Four methods are available for formulating these element properties: direct approach, variation approach, weighted residual approach and energy balance approach. Any one of these methods can be used for assembling element properties. In solid mechanics variation approach is commonly employed to assemble stiffness matrix and nodal force vector (Bhavikatti, 2005).

Element properties are used to assemble global properties/ structural properties to get system equation;

\[ [K]\{\delta\} = \{F\} \]  \hspace{1cm} (2.5)

Where
\[ [K] \] is the global stiffness matrix
\[ \{\delta\} \] is global nodal displacement vector and \( \{F\} \) is global nodal force vector.
Then the boundary conditions are imposed. And the solution of these simultaneous equations gives the nodal unknowns. Using these nodal values additional calculations are made to get the required gradients e.g. stresses and strains.

Thus, the steps involved in the finite element analysis are:

1. Select suitable field variables and the elements.
2. Discritize the continua
3. Select interpolation functions.
4. Find the element properties.
5. Assemble element properties to get global properties.
6. Impose the boundary conditions.
7. Solve the system of equations to get the nodal unknowns.
8. Make additional calculations to get the required values.

2.7 Previous Studies

There are many applications of plane stress/strain element in different fields of analysis.

Pidia, Yang and Soedel, 1989 used large strain 8-node plane stress Isoparametric finite element for prediction of rubber fraction. The formulation is based on total Lagrangian and incremental formulation.

Mohamed, 1983 used both Green strain and engineering strain measures to solve thin beam problems. He also, proposed total Lagrangian modified incremental equations for a two-dimensional state of stress based on the engineering strains.

Cook, 1995 stated that geometric nonlinearity (as opposed to material nonlinearity) arises when deformations are large enough to significantly affect applied load. The goal of analysis is to construct the nonlinear relation between applied load and the resulting deformation. He assumed that the surfaces are sufficiently thin that large displacements are possible without yielding of the material.
Zienkiewicz, 2000 introduced the geometric nonlinear analysis using the Lagrangian formulation, with incremental procedure combined with Newton-Raphson iterative techniques.

Mohammed, 2009 performed the geometric nonlinear analysis of isoparametric Quadrilateral Elements and the formulation of 4-noded quadrilateral isoparametric element. He also developed finite element analysis code to model the geometric nonlinear solid element, and verified of the code by some examples and comparison with analytical solution.

Mohamed and Adam, 2003 presented a finite element formulation for large deformation analysis of shells; the three-dimension 8-noded isoparametric finite element was adopted. A Total Lagrangian formulation by using Engineering strain and Green strain were used in derivation of the geometric element stiffness matrix. The nonlinear equilibrium equations were solved by combined incremental load and Newton-Raphson method.

Mellierr, 1969 presented a thesis on a finite element analysis for geometrically nonlinear large displacement problems in thin elastic plates and shells. In his study only stable equilibrium configuration was considered Engineering strains were assumed to remain small, the geometrical stiffness of the resulting eighteen degree of freedom triangular element was derived from a purely geometrical stand point. The geometrical stiffness was linked with the standard small displacement stiffness and was used in the linear incremental approach to obtain numerical solution to the large displacement problems.

Akasha and Mohamed, 2012 developed a formulation for geometric nonlinear plane stress/strain based on logarithmic strain. That formulation was coupled with modifying a formulation based on a engineering strain. A geometric nonlinear Total Lagrangian formulation was adopted for two – dimensional elasticity using 4-node plane finite elements. The solution of nonlinear equation was obtained by Newton-Raphson method.

AbdElazeez, 2015 developed a formulation for geometric nonlinear finite element of cantilever beam based on Green strain. The formulation was
implemented on computer by using MATLAB program. The incremental method was used in solving nonlinear equilibrium equations.

2.8 Nonlinear solution techniques

2.8.1 Introduction

The finite element formulation for nonlinear problems requires solution techniques to solve the system of nonlinear equations. The solution procedure may even influence the formulation of the problem.

Different methods are available according to the way to calculate the displacement increment, as follows:

2.9.2 Newton-Raphson Method

This method is popular in numerical analysis to find the roots of nonlinear equations. Basically, most numerical methods for solving a system of nonlinear equations assume an initial estimate, $u^0$ and find its increment, $\Delta u$ so that new estimate, $u^0 + \Delta u$, is close to the solution to equation (3.33). In order to find the increment, the nonlinear equations are locally approximated by linear once. This process is repeated until the original nonlinear equations are satisfied as shown in Fig (2.2).

![Newton-Raphson method](image)

**Fig (2.2)** Newton-Raphson method
2.9.3 Modified Newton-Raphson Method

At the iteration process the Newton-Raphson Method requires forming the Jacobian matrix and the system of linearized equations should be solved for each increment of solution. Computationally, these are expensive tasks. In the finite element framework, building the tangent stiffness matrix and solving the matrix equation are the two most computationally intensive procedures. The modified Newton-Raphson method is an attempt to make these procedures less expensive. Instead of formulating a new tangent stiffness matrix at each iteration, the initial tangent stiffness matrix is repeatedly used in all iterations as shown in Fig (2.3).

![Diagram](image)

**Fig (2.3)** Modified Newton-Raphson method

2.9.4 Incremental Force Method

The idea of the incremental force method is to apply the load in increments. Within each load increment, the procedure is the same as the standard Newton-Raphson method. The next load increment is applied after the solution corresponding to the current load increment has converged. The converged solution at each increment is then used as an initial estimate of the next increment as shown in Fig (2.4).
Fig (2.4) Incremental force method

The following chapter presents the finite element formulation of 4-node element and theory of geometrically nonlinear plane stress/strain problems.
Chapter Three

Geometrically Non-linear Finite Element Analysis of Plane Stress/Strain Problems

3.1 Introduction

Geometric nonlinearity arises when change in geometry whether large or small, has a significant effect on the load deformation behavior. Geometric nonlinearity includes deformation-dependent boundary conditions and loading.

As a direct consequence of geometric nonlinearity, the stiffness matrix of the finite element model is not constant. It is a function of the residual displacement. Therefore, an iterative procedure is required to obtain the equilibrium state.

For a given load, there are two main Lagrangian formulations depending on the configuration to which the variables involved in each step are referred. These are:

1. The Total Lagrangian formulation where all variables are referred to initial configuration in Fig (3.1).

2. The Updated Lagrangian formulation where all the variables are referred to the configuration at the beginning of the load step considered in Fig (3.2).
3.2 Formulations of The 4-node Isoparametric Plane Stress/Strain Element:

3.2.1 Introduction

The bilinear quadratic element as shown in Fig (3.3) and Fig (4.3) is a two dimensional finite element with both local and global coordinates. It is characterized by linear shape functions in each of the x and y directions. This element can be used for plane stress or plain strain problems in elasticity. It is a generalization of the 4-node rectangular element (Pattan, 2008).

A Total Lagrangian formulation based on 4-node isoparametric plane stress/strain using Green's strain with 2\textsuperscript{nd} Piola-Kirchhoff is adopted in this chapter.

![Fig (3.3) 4-node Isoparametric Element](image-url)
3.2.2 Stress-Strain relations:

In the two-dimensions the stresses and stain components are defined as (Akasha, 2009 and Mohamed et. el., 2013):

\[
\{\varepsilon\} = \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ Y_{xy} \end{bmatrix} \quad \text{For Green's strain} \quad (3.1)
\]

Consider the material is an isotropic, the stresses are defined by:

\[
\{\sigma\} = \begin{bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{bmatrix} = [D]\{\varepsilon\} \quad \text{For Green's strain} \quad (3.2)
\]

Where D is elasticity matrix, in case of isotropic material,

\[
[D] = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & 0 \\ \nu & 1-\nu & 0 \\ 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix} \quad \text{For plane strain case} \quad (3.3a)
\]

\[
[D] = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix} \quad \text{For plane stress case} \quad (3.3b)
\]

It is using the shape function for the geometry for defining the geometry at any point within the element in terms of nodal vectors as:

\[
x = \sum_{i=1}^{4} N_i x_i \quad , \quad y = \sum_{i=1}^{4} N_i y_i \quad (3.4a)
\]
\[
\begin{bmatrix}
\{x\} & N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 & 0
\end{bmatrix}
= \begin{bmatrix}
N_{1} \{X^e\}
\end{bmatrix}
\]

Where the \( N_i \) the shape function element as:

\[
N_1 = \frac{(1-r)(1-s)}{4}, \quad N_2 = \frac{(1+r)(1-s)}{4}, \quad N_3 = \frac{(1+r)(1+s)}{4}, \quad N_4 = \frac{(1-r)(1+s)}{4}
\]

since \( r \) and \( s \) are the natural coordinates.

The displacement at any point is defined in terms of nodal displacements as:

\[
\begin{bmatrix}
\{u\} & N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 & 0
\end{bmatrix}
= \begin{bmatrix}
N \{d^e\}
\end{bmatrix}
\]

The relation between the Cartesian and natural coordinates can be obtained using chain rule of partial differential as:

\[
\begin{bmatrix}
\frac{\partial}{\partial r} & \frac{\partial}{\partial s}
\end{bmatrix}
= \begin{bmatrix}
\frac{\partial}{\partial x} & \frac{\partial}{\partial y}
\end{bmatrix}
\begin{bmatrix}
\frac{\partial}{\partial x} & \frac{\partial}{\partial y}
\end{bmatrix}
\]

\[
= [J]
\]

Where \([J]\) is Jacobian matrix

It relates derivatives of function in local coordinate system to derivative in global coordinate system which obtained by solving (3.7) as:
\[
\begin{pmatrix}
\frac{\partial x}{\partial x} \\
\frac{\partial y}{\partial y}
\end{pmatrix} = [J]^{-1}
\begin{pmatrix}
\frac{\partial}{\partial r} \\
\frac{\partial}{\partial s}
\end{pmatrix}
\] (3.8)

Since \(N_i\) is function in natural coordinates \((r,s)\) the Jacobian matrix can be evaluated as:

\[
[J] = \begin{vmatrix}
\frac{\partial N_1}{\partial r} & \frac{\partial N_2}{\partial r} & \frac{\partial N_3}{\partial r} & \frac{\partial N_4}{\partial r} \\
\frac{\partial N_1}{\partial s} & \frac{\partial N_2}{\partial s} & \frac{\partial N_3}{\partial s} & \frac{\partial N_4}{\partial s}
\end{vmatrix}
\begin{bmatrix}
x_1 \\
x_2 \\
x_3 \\
x_4
\end{bmatrix}
\begin{bmatrix}
y_1 \\
y_2 \\
y_3 \\
y_4
\end{bmatrix}
\] (3.9)

Using the values of \(N_i\) and substituting their derivatives in (3.9) to gives:

\[
[J] = \begin{vmatrix}
\frac{\partial N_1}{\partial r} & \frac{\partial N_2}{\partial r} & \frac{\partial N_3}{\partial r} & \frac{\partial N_4}{\partial r} \\
\frac{\partial N_1}{\partial s} & \frac{\partial N_2}{\partial s} & \frac{\partial N_3}{\partial s} & \frac{\partial N_4}{\partial s}
\end{vmatrix}
\begin{bmatrix}
\frac{x_1}{4} \\
\frac{x_2}{4} \\
\frac{x_3}{4} \\
\frac{x_4}{4}
\end{bmatrix}
\begin{bmatrix}
y_1 \\
y_2 \\
y_3 \\
y_4
\end{bmatrix}
\] (3.10)

The inverse of Jacobin \([J]^{-1}\) is:

\[
[J]^{-1} = \begin{vmatrix}
\frac{\partial r}{\partial x} & \frac{\partial r}{\partial s} \\
\frac{\partial s}{\partial x} & \frac{\partial s}{\partial s}
\end{vmatrix} = \frac{1}{\text{det}[J]}
\begin{vmatrix}
\frac{\partial y}{\partial r} & -\frac{\partial y}{\partial s} \\
-\frac{\partial x}{\partial r} & \frac{\partial x}{\partial s}
\end{vmatrix}
\]

\[
[J]^{-1} = [J^*] = \begin{bmatrix}
J_{11}^* & J_{12}^* \\
J_{21}^* & J_{22}^*
\end{bmatrix}, \text{ where } \text{det}[J] = \frac{\partial x}{\partial r} \times \frac{\partial y}{\partial s} - \frac{\partial x}{\partial s} \times \frac{\partial y}{\partial r}
\]

And \(J_{11}^*, J_{12}^*, J_{21}^*, J_{22}^*\) are the elements of Jacobin inverse matrix

The derivatives of displacement in Cartesian coordinates can be expressed in terms of natural coordinates as:
\[
\begin{pmatrix}
\frac{\partial u}{\partial x} \\
\frac{\partial u}{\partial y} \\
\frac{\partial v}{\partial x} \\
\frac{\partial v}{\partial y}
\end{pmatrix}
= \begin{bmatrix}
J_{11}^* & J_{12}^* & 0 & 0 \\
J_{21}^* & J_{22}^* & 0 & 0 \\
0 & 0 & J_{11}^* & J_{12}^* \\
0 & 0 & J_{21}^* & J_{22}^*
\end{bmatrix}
\begin{pmatrix}
\frac{\partial u}{\partial r} \\
\frac{\partial u}{\partial s} \\
\frac{\partial v}{\partial r} \\
\frac{\partial v}{\partial s}
\end{pmatrix}
\]

(3.11)

### 3.2.3 Incremental Formulations for Geometrically Nonlinear Plane Stress/Strain Element

The Green strains \( \varepsilon \) and their variation \( \delta \varepsilon \) are defined in terms of the nodal variables in finite element representation as (Kim, 2014 and Bathe, et. al., 1975):

\[
\{ \varepsilon \} = \begin{pmatrix}
\varepsilon_x \\
\varepsilon_y \\
\gamma_{xy}
\end{pmatrix} = \begin{bmatrix}
\frac{\partial u}{\partial x} & \frac{\partial v}{\partial y} & 0 & 0 \\
0 & \frac{\partial u}{\partial y} & \frac{\partial v}{\partial x} & 0 \\
\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} & \frac{\partial u}{\partial y} & \frac{\partial v}{\partial x} & \frac{\partial v}{\partial y}
\end{bmatrix}
\begin{pmatrix}
\frac{\partial u}{\partial x} \\
\frac{\partial u}{\partial y} \\
\frac{\partial v}{\partial x} \\
\frac{\partial v}{\partial y}
\end{pmatrix}
\]

+ \frac{1}{2}
\begin{bmatrix}
\frac{\partial u}{\partial x} & \frac{\partial v}{\partial y} & 0 & 0 \\
0 & \frac{\partial u}{\partial y} & \frac{\partial v}{\partial x} & 0 \\
\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} & \frac{\partial u}{\partial y} & \frac{\partial v}{\partial x} & \frac{\partial v}{\partial y}
\end{bmatrix}
\begin{pmatrix}
\frac{\partial u}{\partial x} \\
\frac{\partial u}{\partial y} \\
\frac{\partial v}{\partial x} \\
\frac{\partial v}{\partial y}
\end{pmatrix}

(3.12)

Or \( \{ \varepsilon \} = \{ \varepsilon_0 \} + \{ \varepsilon_L \} \)

(3.13)

Where \( \{ \varepsilon_0 \} \) is the linear strain vector, \( \{ \varepsilon_L \} \) is non-linear strain component

Let

\[
[A] = \begin{bmatrix}
\frac{\partial u}{\partial x} & \frac{\partial v}{\partial y} & 0 & 0 \\
0 & 0 & \frac{\partial u}{\partial x} & \frac{\partial v}{\partial y} \\
\frac{\partial u}{\partial y} & \frac{\partial v}{\partial y} & \frac{\partial u}{\partial x} & \frac{\partial v}{\partial x}
\end{bmatrix}
\]

(3.14)

Where \([A]\) is the matrix containing displacement derivatives w. r. t Cartesian coordinates
And  \[
\{\theta\} = \begin{bmatrix}
\frac{\partial u}{\partial x} \\
\frac{\partial u}{\partial y} \\
\frac{\partial v}{\partial x} \\
\frac{\partial v}{\partial y}
\end{bmatrix}
\tag{3.15}
\]

Where:

\{\theta\} is vector containing displacement derivatives \textit{w.r.t.} Cartesian Coordinates.

Then the non-linear strain is:

\[
\{\varepsilon_L\} = \frac{1}{2} [A] \{\theta\}
\tag{3.16}
\]

The displacements are defined in terms of nodal parameters using the appropriate shape functions, thus

\[
u = \sum_{i=1}^{4} N_i u_i \text{ and } v = \sum_{i=1}^{4} N_i v_i \quad \text{or} \quad \begin{bmatrix} u \\ v \end{bmatrix} = [N]\{a\}
\tag{3.17}
\]

And there derivatives:

\[
\{\theta\} = [G]\{a\}
\tag{3.18}
\]

Where:

\[
\{a\} = [u_1 \ v_1 \ u_2 \ v_2 \ u_3 \ v_3 \ u_4 \ v_4]^T
\tag{3.19}
\]

Is vector of nodal displacements and:

\[
[G] = \begin{bmatrix}
\frac{\partial N_1}{\partial x} & 0 & \frac{\partial N_2}{\partial x} & 0 & \frac{\partial N_3}{\partial x} & 0 & \frac{\partial N_4}{\partial x} & 0 \\
0 & \frac{\partial N_1}{\partial y} & 0 & \frac{\partial N_2}{\partial y} & 0 & \frac{\partial N_3}{\partial y} & 0 & \frac{\partial N_4}{\partial y} \\
0 & 0 & \frac{\partial N_1}{\partial x} & \frac{\partial N_2}{\partial x} & \frac{\partial N_3}{\partial x} & \frac{\partial N_4}{\partial x} \\
0 & 0 & 0 & \frac{\partial N_1}{\partial y} & \frac{\partial N_2}{\partial y} & \frac{\partial N_3}{\partial y} & \frac{\partial N_4}{\partial y}
\end{bmatrix}
\tag{3.20}
\]

\([G]\) is the matrix containing shape function derivatives \textit{w.r.t.} Cartesian coordinates.
From equations (3.12) and (3.17) \( \{ \varepsilon_0 \} \) can be obtained as:

\[
\{ \varepsilon_0 \} = \begin{pmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial y} \end{pmatrix} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \end{bmatrix} \begin{pmatrix} u \\ v \end{pmatrix}
\]

(3.21)

or \( \{ \varepsilon_0 \} = [B_0] \{ a \} \)

In the Total Lagrangian formulations, the infinitesimal strain-displacement transformation \([B_0]\) becomes (Kim, 2014 and Bathe, et al., 1975):

\[
[B_0] = \begin{bmatrix} \frac{\partial N_1}{\partial x} & 0 & \frac{\partial N_2}{\partial x} & 0 & \frac{\partial N_3}{\partial x} & 0 & \frac{\partial N_4}{\partial x} & 0 \\ 0 & \frac{\partial N_1}{\partial y} & 0 & \frac{\partial N_2}{\partial y} & 0 & \frac{\partial N_3}{\partial y} & 0 & \frac{\partial N_4}{\partial y} \\ \frac{\partial N_1}{\partial y} & \frac{\partial N_1}{\partial x} & \frac{\partial N_2}{\partial y} & \frac{\partial N_3}{\partial x} & \frac{\partial N_4}{\partial y} & \frac{\partial N_4}{\partial x} \end{bmatrix}
\]

(3.22)

Taking the variation of equation (3.16) gives:

\[
\delta \{ \varepsilon_L \} = \frac{1}{2} [A] \delta \{ \theta \} + \frac{1}{2} [A] \delta \{ \theta \} = [A] \delta \{ \theta \} = [A][G] \delta \{ a \} = [B_L] \delta \{ a \}
\]

(3.23)

In which

\([B_L] = [A][G]\) Or

The linear strain matrix \([B_L]\) in total Lagrangian formulation becomes (Kim, 2014 and Bathe, et al., 1975):

\[
[B_L] = \begin{bmatrix} F_{11} \frac{\partial N_1}{\partial x} + F_{12} \frac{\partial N_1}{\partial y} \\ F_{21} \frac{\partial N_1}{\partial y} + F_{22} \frac{\partial N_1}{\partial y} \\ F_{11} \frac{\partial N_2}{\partial x} + F_{12} \frac{\partial N_2}{\partial y} \\ F_{21} \frac{\partial N_2}{\partial y} + F_{22} \frac{\partial N_2}{\partial y} \\ F_{11} \frac{\partial N_3}{\partial x} + F_{12} \frac{\partial N_3}{\partial y} \\ F_{21} \frac{\partial N_3}{\partial y} + F_{22} \frac{\partial N_3}{\partial y} \\ F_{11} \frac{\partial N_4}{\partial x} + F_{12} \frac{\partial N_4}{\partial y} \\ F_{21} \frac{\partial N_4}{\partial y} + F_{22} \frac{\partial N_4}{\partial y} \end{bmatrix}
\]

(3.24)
Where \( \{F_{ij}\} \) is the deformation gradient vector which can be written as:

\[
\{F\} = \{F_{11} \ F_{12} \ F_{21} \ F_{22}\}^T = \{1 + \frac{\partial u}{\partial x} \ \frac{\partial u}{\partial y} \ \frac{\partial v}{\partial x} \ \frac{\partial v}{\partial y}\}^T
\]  

Using equations (3.22) and (3.23) and taking the variation of equation (3.13) gives:

\[
\delta\{\varepsilon\} = [B_0]\delta\{a\} + [B_L]\delta\{a\} = ([B_0] + [B_L])\delta\{a\} = [B]\delta\{a\}
\]  

In which

\[
[B] = [B_0] + [B_L]
\]  

### 3.3 Formulation of Geometrically Nonlinear Plane Stress/Strain Element:

In this section the 2\textsuperscript{nd} Piola-Kirchhoff stresses and the Green Lagrangian strain are used in total Lagrangian formulations. The solution of the resulting nonlinear equations is obtained by incremental and Newton-Raphson method.

### 3.3.1 Stress-Strain relations

\[
[\sigma] = \begin{bmatrix}
\sigma_x \\
\sigma_y \\
\tau_{xy}
\end{bmatrix} = [D]\begin{bmatrix}
\varepsilon_x \\
\varepsilon_y \\
\gamma_{xy}
\end{bmatrix} \quad \text{or} \quad [D]\{\varepsilon\}
\]  

Where \([D]\) is given by (3.3) for plane stress and plane strain.

For Green's strain and 2nd Piola-Kirchhoff stresses
3.3.2 Strain-Displacement Relations:

![Diagram](image)

**Fig (3.5) Deformed and Undeformed shape**

Referring to Fig (3.4), the position vector for point P(x, y) after deformation is:

$$\vec{R} = (x + u)i + (y + v)j$$  \hspace{1cm} (3.29)

In which $u$ and $v$ are components of displacement in the global axes directions.

The displacement gradient vectors are given from equation (3.29) by:

$$\vec{g}_x = \frac{\partial \vec{R}}{\partial x} = \left(1 + \frac{\partial u}{\partial x}\right)i + \frac{\partial v}{\partial x}j$$

$$\vec{g}_y = \frac{\partial \vec{R}}{\partial y} = \left(\frac{\partial u}{\partial y}\right)i + (1 + \frac{\partial v}{\partial x})j$$  \hspace{1cm} (3.30)

### 3.3.3 Green Lagrangian Strain

The Green's strains are defined by a vector:
\{\varepsilon\} = \left\{ \begin{array}{l}
\varepsilon_x \\
\varepsilon_y \\
\gamma_{xy}
\end{array} \right\} = \left\{ \begin{array}{l}
\frac{1}{2} (g_x \cdot g_x - 1) \\
\frac{1}{2} (g_y \cdot g_y - 1) \\
g_x \cdot g_y
\end{array} \right\} \quad (3.31)

From equation (3.30)

\begin{align*}
g_x \cdot g_x &= \left[ (1 + \frac{\partial u}{\partial x}) i + \frac{\partial v}{\partial x} j \right] \cdot \left[ (1 + \frac{\partial u}{\partial x}) i + \frac{\partial v}{\partial x} j \right] = \left( 1 + \frac{\partial u}{\partial x} \right)^2 + \left( \frac{\partial v}{\partial x} \right)^2 \\
&= 1 + 2 \frac{\partial u}{\partial x} + \left( \frac{\partial u}{\partial x} \right)^2 + \left( \frac{\partial v}{\partial x} \right)^2
\end{align*}

Similarly:

\begin{align*}
g_y \cdot g_y &= \left[ \left( \frac{\partial u}{\partial y} \right) i + (1 + \frac{\partial v}{\partial y}) j \right] \cdot \left[ \left( \frac{\partial u}{\partial y} \right) i + (1 + \frac{\partial v}{\partial y}) j \right] = \left( \frac{\partial u}{\partial y} \right)^2 + (1 + \frac{\partial v}{\partial y})^2 \\
&= 1 + 2 \frac{\partial v}{\partial y} + \left( \frac{\partial u}{\partial y} \right)^2 + \left( \frac{\partial v}{\partial y} \right)^2
\end{align*}

And

\begin{align*}
g_x \cdot g_y &= \left[ (1 + \frac{\partial u}{\partial x}) i + \frac{\partial v}{\partial x} j \right] \cdot \left[ \left( \frac{\partial u}{\partial y} \right) i + (1 + \frac{\partial v}{\partial y}) j \right] \\
&= \left( 1 + \frac{\partial u}{\partial x} \right) \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \left( 1 + \frac{\partial v}{\partial y} \right) \\
&= \frac{\partial u}{\partial y} + \frac{\partial u}{\partial y} \frac{\partial u}{\partial x} + \frac{\partial v}{\partial x} + \frac{\partial v}{\partial x} \frac{\partial v}{\partial y} \\
&= \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \frac{\partial u}{\partial x} + \frac{\partial v}{\partial x} \frac{\partial v}{\partial y}
\end{align*}

Subsisting in equation (3.31) the strains become:

\begin{align*}
\{\varepsilon\} = \left\{ \begin{array}{l}
\varepsilon_x \\
\varepsilon_y \\
\gamma_{xy}
\end{array} \right\} = \left[ \begin{array}{c}
\frac{\partial u}{\partial x} + \frac{1}{2} \left( \frac{\partial u}{\partial x} \right)^2 + \frac{1}{2} \left( \frac{\partial v}{\partial x} \right)^2 \\
\frac{\partial v}{\partial y} + \frac{1}{2} \left( \frac{\partial u}{\partial y} \right)^2 + \frac{1}{2} \left( \frac{\partial v}{\partial y} \right)^2 \\
\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \frac{\partial u}{\partial x} + \frac{\partial v}{\partial x} \frac{\partial v}{\partial y}
\end{array} \right] \quad (3.32)
\end{align*}
3.3.4 Tangent Stiffness Matrix due to Green Strains:

From virtual work, the equilibrium equations can be written as:

\[ \{ \varphi \} = \int_v [B]^T \{ \sigma \} dv - \{ f \} = 0 \]  \hspace{1cm} (3.33)

Where:

\{ \varphi \} \text{ is the difference between external and internal generalized forces}

\{ f \} \text{ is the vector of external forces}

\{ \sigma \} \text{ is the } 2^{nd} \text{ Piola-Kirchhoff stresses vector}

The stiffness matrix is obtained by taking the variation of equation (3.33) w.r.t the nodal variables \{a\}

\[ \delta \{ \varphi \} = \int_v [B]^T \delta \{ \sigma \} dv + \int_v \delta [B]^T \{ \sigma \} dv \] \hspace{1cm} (3.34)

The discretized element of volume \( dv = h \ det \ J \ dr \ ds \)

Where \( h \) is the thickness of the element

Using equation (3.26) and (3.24)

\[ \{ \sigma \} = \begin{bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{bmatrix} = [D] \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix} \text{ or } \{ \sigma \} = [D] \{ \varepsilon \} 

And

\[ \delta \{ \sigma \} = [D] \delta \{ \varepsilon \} = [D][B] \delta \{ a \} \]

Therefore

\[ \int_v [B]^T \delta \{ \sigma \} dv = \left( \int_v [B]^T [D][B] dv \right) \delta \{ a \} = ([K_0] + [K_L]) \delta \{ a \} \] \hspace{1cm} (3.35)

Where the matrix \([K_0]\) represents the small displacement stiffness matrix and is given by:
\[ [K_0] = \int [B_0]^T [D] [B_0] d\nu \]  

(3.36)

And the matrix \([K_L]\) is the linear strain stiffness matrix

\[ [K_L] = \int ([B_0]^T [D] [B_L] + [B_L]^T [D] [B_0] + [B_L][D][B_L]) d\nu \]  

(3.37)

Where \([K_L]\) is the linear strain stiffness matrix

From equation (3.25)


By taking the variation of \([B]^T\) and using equation (3.23) we have:


Therefore

\[ \int \nu \delta [B]^T \{\sigma\} d\nu = \int \nu [G]^T \delta [A]^T \{\sigma\} d\nu = [K_\sigma] \delta \{a\} \]

(3.38)

Where \([K_\sigma]\) is a symmetric matrix depending on the stress level and is known as the initial stress matrix

Using equations (3.26), (3.14) & (3.18)
\[
[A]^T \{\sigma\} = \begin{bmatrix}
\frac{\partial u}{\partial x} & 0 & \frac{\partial u}{\partial y} \\
\frac{\partial v}{\partial x} & 0 & \frac{\partial v}{\partial y} \\
0 & \frac{\partial u}{\partial y} & \frac{\partial u}{\partial x} \\
0 & \frac{\partial v}{\partial y} & \frac{\partial v}{\partial x}
\end{bmatrix}
\begin{bmatrix}
\sigma_x \\
\sigma_y \\
\gamma_{xy}
\end{bmatrix} = \begin{bmatrix}
\frac{\partial u}{\partial x} \\
\frac{\partial v}{\partial x} \\
0 \\
0
\end{bmatrix}
\begin{bmatrix}
\sigma_x + \gamma_{xy} \\
\sigma_x + \gamma_{xy} \\
\sigma_y + \gamma_{xy} \\
\sigma_y + \gamma_{xy}
\end{bmatrix}
\]

Or \([A]^T \{\sigma\} = [M]\{\theta\} = [M][G]\{a\} \quad (3.39)

Where \([M]\) is initial stress matrix and is given by:

\[
[M] = \begin{bmatrix}
\sigma_x[l] & \tau_{xy}[l] \\
\tau_{xy}[l] & \sigma_y[l]
\end{bmatrix}, \quad [l] = \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix} \quad (3.40)
\]

Taking the variation of Eq. (3.39) gives

\[
\delta[A]^T \{\sigma\} = [M][G]\delta\{a\} \quad (3.41)
\]

Substitute equation (3.40) in (3.38) to get:

\[
\int_v [G]^T \delta[A]^T \{\sigma\} dv = \left( \int_v [G]^T [M][G] dv \right) \delta\{a\} = [K_\sigma]\delta\{a\} \quad (3.42)
\]

Therefore, \([K_\sigma] = \int_v [G]^T [M][G] dv \quad (3.43)\]

Where \([G]\) is the Non-linear strain-displacement transformation matrix written as:
\[
\begin{bmatrix}
\frac{\partial N_1}{\partial x} & 0 & \frac{\partial N_2}{\partial x} & 0 & \frac{\partial N_3}{\partial x} & 0 & \frac{\partial N_4}{\partial x} & 0 \\
0 & \frac{\partial N_1}{\partial y} & 0 & \frac{\partial N_2}{\partial y} & 0 & \frac{\partial N_3}{\partial y} & 0 & \frac{\partial N_4}{\partial y} \\
0 & 0 & \frac{\partial N_1}{\partial x} & 0 & \frac{\partial N_2}{\partial x} & 0 & \frac{\partial N_3}{\partial x} & 0 \\
0 & 0 & 0 & \frac{\partial N_1}{\partial y} & 0 & \frac{\partial N_2}{\partial y} & 0 & \frac{\partial N_3}{\partial y}
\end{bmatrix}
\]  
(3.44)

From equations (3.35), (3.38) & (3.34)

\[
\delta \{\varphi\} = ([K_0] + [K_L] + [K_\sigma])\delta \{a\} = [K_T]\delta \{a\}
\]

Where:

\[
[K_T] = [K_0] + [K_L] + [K_\sigma]
\]  
(3.45)

Is the total tangent stiffness matrix

**3.4 Incremental Equilibrium Equations**

From equation (3.45) the tangent stiffness matrix takes the form:

\[
[K_T] = [K_0] + [K_L(a_0)] + [K_\sigma]
\]

Where:

\[
[K_0] + [K_L(a_0)] = \int_v [B]^T[D][B]dv
\]  
(3.46)

\[
\]  
(3.47)

The displacement increments \(\{\Delta a_0^i\}\) are evaluated by using \([K_T]\) and residuals as:

\[
\{\Delta a_0^i\} = -[K_T]^{-1}\{\varphi^i\}
\]  
(3.48)

The strain increments are defined by:

\[
\{\Delta \varepsilon^i\} = \left[[B_0] + [B_L(a_0^i)] + \frac{1}{2}[B_L(\Delta a_0^i)]\right]\{\Delta a_0^i\}
\]  
(3.49)

The stresses increments are given by:
\[ \{ \Delta S_0^i \} = \{ D \} \{ \Delta \varepsilon_0^i \} \quad (3.50) \]

The total stresses are:

\[ \{ s_0^{i+1} \} = \{ s_0^i \} + \{ \Delta s_0^i \} \quad (3.51) \]

The nodal residual forces for the next iteration are evaluated as flows:

\[ \{- \varphi^{i+1} \} = \{ R \} - \int_v \{ B \}^T \{ s^{i+1} \} dv \quad (3.52) \]

And

\[ [B] = [B_0] + [B_L (a_0^{i+1})] \quad (3.53) \]
Chapter four

Implementation and Application of Program and Verification of Results

4.1 Description of Computer Program

4.1.1 Introduction

A computer program for linear and nonlinear analysis of plane stresses and plane strains by finite element method is presented in this chapter. The program was developed and written by MATLAB program language. The programs have nine subroutines or functions. The basic functions are, LINA2D and NONLA2D. The function LINA2D was developed for linear formulation, and the NONLA2D was developed for nonlinear formulation based on Green strain presented in chapter three.

The formulation stated in chapter three, was implemented in the standard MATLAB (2010b), and the major goals were to achieve good results, and compare them with known results. In the program the 4-noded 2-dimensional isoparametric element was used.

The main program procedures and the all functions are explained in the following sections, the listing of the main program and the subroutines are presented in Appendices A and B.

4.1.2 Main program procedures:

The basic stages of implementation of MATLAB programming are as follows:

1- read input data which include:

   - Mesh generation function

   - Element external nodal force EXTFORCE
- Material properties (Passion ratio, Elastic modulus)
- Program parameters (ITER, TOL, INC)
- Load increments (Start increment, End increment ...)
- Boundary conditions

2- Call FORD to calculate elastic modulus matrix [D]

3- Read SHAPLE to compute shape functions, their derivatives and determinant of Jacobian of 4-noded element

4- Before incrementing and Newton-Raphson iteration start, the main function/ program calls CHECK to check the area and connectivity of the elements and the program stops if the determinant is negative or if there is any error in the element connectivity.

5- Begin load increments INC=1, and iteration ITER=1 for nonlinear analysis.

6- If Analtype =1, call LINA2D to calculate element stiffness matrix and assemble it in global matrix for linear analysis.

7- Else if Analtype is not equal to one use NONLINA2D to compute global tangent stiffness matrix KT, and residual force (FORCE) for nonlinear analysis.

8- After calculation of KT and FORCE:

   calculat:

   - displacement increments $\Delta\{U\} = [K]/\{R\}$
   - total displacement vector $\{U\} = \{U\} + \Delta\{U\}$
   - strain increments
   - stresses increments
   - the total stresses
   - the total residual force
a- If solution has not converged or this is not the last iteration then go to next iteration (go to 5).

b- Else continue if solution has converged.

9- call OUTPUT print convergent results

10-call PROUT to print out convergence displacement and stresses

11-calculate direct stresses and shear stresses at integration points of elements

12 – If this is not the last increment go to (5); otherwise:

13-   End program

4.1.3 Program layout

The computer program system is divided into main nine functions/subroutines; each function consists of specific task in the program. The main program flow chart is shown in the Figure (4.1) and the subroutines are explained in the next section.
Fig (4.1) Main program flow chart

ITER = ITER + 1

If Solution converge

CALL OUTPUT

INC = INC + 1

If INC = MAX

CALL PROUT

ITER = ITER + 1

Yes

No

STOP

END

No

Yes

Yes
\[ \Delta \varepsilon_i = [B_0] + [B_L(a_0)] + \frac{1}{2}[B_L(\Delta a_0)] \]

\[ \{\Delta S_0^i\} = [D]\{\Delta \varepsilon_0^i\} \]

\[ \{S_n^{i+1}\} = \{S_n^i\} + \{\Delta S_n^i\} \]

\[ [K_0] + [K_L(a_0)] = \int [B]^T[D][B]dv \]
Fig (4.1a) flow chart for function NONLA2D
Calculate Strain
\[ \{\varepsilon\}_i = [B_0] \{a_i\} \]

Calculate Stress
\[ \{S_0\} = [D]\{\varepsilon\}_i \]

Calculate Linear Tangent Stiffness Matrix
\[ [K_T] = \int [B_0]^T[D][B_0]dv \]

Any Number of Gauss Points?
No

Any Number of Elements?
No

- Calculate Global Tangent Stiffness Matrix
- Calculate Global Force Vector

Solve the System of Linear:
\[ \Delta\{U\} = [K]/\{R\} \]

**Fig (4.1b)** flow chart of function LINA2D
4.1.4 Functions Explanation

1- Input data function

This subroutine consists of the basic data that is required in defining the geometry of 4-noded isoparametric element, connectivity, properties of element, and boundary conditions, external nodal force. It also reads the control parameters.

2- Function MESH

According to dimensions of the structure and specified sub-divisions in the two directions, this subroutine generates a mesh of linear quadrilateral elements.

3- Function FORD

This subroutine calculates the modulus matrix for linear elastic material [D]

4- Function SHAPLE

According to the 4-node isoparametric element, the function computes the shape functions, their derivatives, and determinant of the Jacobian of bilinear element.

5- CHECK function

This subroutine checks the determinant of element Jacobian matrix, and the program stops if the determinant is negative, which indicates that either element has negative area or the element connectivity is not correct.

6- LINA2D function

This is the main function for linear analysis as shown in Fig (4.1b), which is starts when Analtype =1. According to the integration point and weights, function reads the element nodal coordinate and calls the subroutine SHAPLE, FORD, to assemble the strain stiffness matrix and then assemble global stiffness matrix and Global force of linear elastic analysis. The function also calculate the
strain and stresses at integration points, and solve the system of linear analysis to compute the displacements at nodes.

7- NONLIA2D function

The function NONLIA2D as shown in Fig (4.1a) is used to compute or assemble the global tangent stiffness matrix and residual force for nonlinear analysis, and solve the nonlinear equation to calculate the increment displacements, displacements, strain increments, stress increments and total stresses. The function starts when Analtype equals any number other than one.

8- OUTPUT function

This function is prints out iteration of results to MATLAB screen.

9- PROUT function

This function prints out analysis results.

4.2 Verification of Formulation

4.2.1 Introduction

For the linear and nonlinear analyses three numerical examples of large deformation problems were analysed to demonstrate the degree of accuracy that can be obtained by using the linear and the geometrically nonlinear formulation based on Green strain and 4-node plane stress/strain element.

The result of displacements, stresses at the integration points and the differences between incremental displacements and incremental stresses are obtained and compared with known results.
4.2.2 Linear analysis

4.2.2.1 Cantilever plate under concentrated load at the end

The linear formulation was tested by analyzing the cantilever plate beam subjected to load at free end. The cantilever is of dimension \( L=3000\text{mm} \), \( D=300\text{mm} \) and thickness \( h=60\text{mm} \) as shown in Fig (4.2). The numerical values of material property parameter are: Young's modulus \( E=210\times10^3\, N/\text{mm}^2 \), Poisson's ratio \( = 0.3 \). The structure is modeled with different numbers of isoparametric elements. The result obtained for displacements and stresses are compared with known results. Graphical comparison of results of displacement at free end and mid-span, stresses at integration point are presented in figures Fig (4.2) to (4.8). Tables (4.1) to (4.5) show the displacements at free end for different load increment and different numbers of elements. Table (4.6) and Fig (4.5) present the deflected (deformed) shape along center line. Figures (4.6) to (4.8) and Tables (4.7) to (4.9) show the stresses at integration point one as shown in Fig (3.4) and El (1) in Fig (4.2).

![Fig (4-2) Cantilever plate beam with vertical load at free end](image-url)
Table (4-1) Vertical displacement ($10^{-1}$mm) at free end and effect of mesh

<table>
<thead>
<tr>
<th>Load(N)</th>
<th>Elements</th>
<th>20EL</th>
<th>40EL</th>
<th>60EL</th>
<th>80EL</th>
<th>100EL</th>
<th>120EL</th>
<th>140EL</th>
<th>160EL</th>
<th>180EL</th>
<th>200EL</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh</td>
<td>2*10</td>
<td>2*20</td>
<td>2*30</td>
<td>2*40</td>
<td>2*50</td>
<td>2*60</td>
<td>2*70</td>
<td>2*80</td>
<td>2*90</td>
<td>2*100</td>
<td></td>
</tr>
<tr>
<td>Aspect</td>
<td>2</td>
<td>1</td>
<td>0.667</td>
<td>0.5</td>
<td>0.4</td>
<td>0.334</td>
<td>0.285</td>
<td>0.25</td>
<td>0.222</td>
<td>0.2</td>
<td></td>
</tr>
<tr>
<td>Mesh</td>
<td>4*5</td>
<td>4*10</td>
<td>4*15</td>
<td>4*20</td>
<td>4*25</td>
<td>4*30</td>
<td>4*35</td>
<td>4*40</td>
<td>4*45</td>
<td>4*50</td>
<td></td>
</tr>
<tr>
<td>Aspect</td>
<td>8</td>
<td>4</td>
<td>2.667</td>
<td>2</td>
<td>1.6</td>
<td>1.333</td>
<td>1.143</td>
<td>1</td>
<td>0.888</td>
<td>0.8</td>
<td></td>
</tr>
</tbody>
</table>

**Fig (4-3)** Effect of mesh and aspect ratio on accuracy of result

Table (4-2) Vertical displacement at free end in point A

<table>
<thead>
<tr>
<th>Load(N)</th>
<th>Vertical displacement(mm)</th>
<th>60E</th>
<th>80E</th>
<th>100E</th>
<th>120E</th>
<th>140E</th>
<th>160E</th>
<th>180E</th>
<th>200E</th>
<th>Exact Sol</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>3000</td>
<td>0.8957</td>
<td>.9120</td>
<td>.9197</td>
<td>.9239</td>
<td>.9265</td>
<td>.9282</td>
<td>.9294</td>
<td>.936</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>6000</td>
<td>1.791</td>
<td>1.824</td>
<td>1.839</td>
<td>1.848</td>
<td>1.853</td>
<td>1.856</td>
<td>1.859</td>
<td>1.860</td>
<td>2</td>
<td></td>
</tr>
<tr>
<td>9000</td>
<td>2.687</td>
<td>2.736</td>
<td>2.759</td>
<td>2.772</td>
<td>2.780</td>
<td>2.785</td>
<td>2.788</td>
<td>2.791</td>
<td>3</td>
<td></td>
</tr>
<tr>
<td>18000</td>
<td>5.374</td>
<td>5.472</td>
<td>5.518</td>
<td>5.544</td>
<td>5.559</td>
<td>5.569</td>
<td>5.576</td>
<td>5.581</td>
<td>6</td>
<td></td>
</tr>
<tr>
<td>24000</td>
<td>7.166</td>
<td>7.296</td>
<td>7.357</td>
<td>7.392</td>
<td>7.412</td>
<td>7.426</td>
<td>7.435</td>
<td>7.442</td>
<td>8</td>
<td></td>
</tr>
</tbody>
</table>

44
**Fig (4-4)** Vertical displacement at free end in point A

Table (4-3) Displacement at free end point A (200EI, 5INC)

<table>
<thead>
<tr>
<th>Increment</th>
<th>Load 24000N V(mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>1</td>
<td>1.488</td>
</tr>
<tr>
<td>2</td>
<td>2.977</td>
</tr>
<tr>
<td>3</td>
<td>4.465</td>
</tr>
<tr>
<td>4</td>
<td>5.953</td>
</tr>
<tr>
<td>5</td>
<td>7.442</td>
</tr>
</tbody>
</table>

Table (4-4) Displacement at free end point A (200EI, 10INC)

<table>
<thead>
<tr>
<th>Increment</th>
<th>Load 24000N V(mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>1</td>
<td>0.7442</td>
</tr>
<tr>
<td>2</td>
<td>1.488</td>
</tr>
<tr>
<td>3</td>
<td>2.233</td>
</tr>
<tr>
<td>4</td>
<td>2.977</td>
</tr>
<tr>
<td>5</td>
<td>3.721</td>
</tr>
<tr>
<td>6</td>
<td>4.465</td>
</tr>
<tr>
<td>7</td>
<td>5.209</td>
</tr>
<tr>
<td>8</td>
<td>5.953</td>
</tr>
<tr>
<td>9</td>
<td>6.698</td>
</tr>
<tr>
<td>10</td>
<td>7.442</td>
</tr>
</tbody>
</table>
Table (4-5) Displacement at free end point A (200 El, 15INC)

<table>
<thead>
<tr>
<th>Increment</th>
<th>Load 24000N V(mm)</th>
<th>Increment</th>
<th>Load 24000N V(mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
<td>11</td>
<td>5.457</td>
</tr>
<tr>
<td>1</td>
<td>0.4961</td>
<td>12</td>
<td>5.953</td>
</tr>
<tr>
<td>2</td>
<td>0.9922</td>
<td>13</td>
<td>6.449</td>
</tr>
<tr>
<td>3</td>
<td>1.488</td>
<td>14</td>
<td>6.946</td>
</tr>
<tr>
<td>4</td>
<td>1.984</td>
<td>15</td>
<td>7.442</td>
</tr>
<tr>
<td>5</td>
<td>2.480</td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>2.976</td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>3.474</td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>3.969</td>
<td></td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>4.465</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>4.961</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table (4-6) Deflected shape along center line (100El)

<table>
<thead>
<tr>
<th>Node</th>
<th>Load 18000N</th>
<th>Load 21000N</th>
<th>Load 24000 N</th>
<th>Node</th>
<th>Load 18000N</th>
<th>Load 21000N</th>
<th>Load 24000 N</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>80</td>
<td>1.854</td>
<td>2.163</td>
<td>2.472</td>
</tr>
<tr>
<td>5</td>
<td>0.0009645</td>
<td>0.001125</td>
<td>0.001286</td>
<td>83</td>
<td>1.983</td>
<td>2.313</td>
<td>2.644</td>
</tr>
<tr>
<td>8</td>
<td>0.0117</td>
<td>0.01303</td>
<td>0.01489</td>
<td>86</td>
<td>2.115</td>
<td>2.467</td>
<td>2.820</td>
</tr>
<tr>
<td>11</td>
<td>0.0277</td>
<td>0.03242</td>
<td>0.03705</td>
<td>89</td>
<td>2.25</td>
<td>2.624</td>
<td>2.999</td>
</tr>
<tr>
<td>14</td>
<td>0.0506</td>
<td>0.05904</td>
<td>0.06748</td>
<td>92</td>
<td>2.387</td>
<td>2.785</td>
<td>3.183</td>
</tr>
<tr>
<td>17</td>
<td>0.07949</td>
<td>0.09274</td>
<td>0.106</td>
<td>95</td>
<td>2.528</td>
<td>2.949</td>
<td>3.37</td>
</tr>
<tr>
<td>20</td>
<td>0.1143</td>
<td>0.1334</td>
<td>0.1524</td>
<td>98</td>
<td>2.67</td>
<td>3.115</td>
<td>3.56</td>
</tr>
<tr>
<td>23</td>
<td>0.1549</td>
<td>0.1808</td>
<td>0.2066</td>
<td>101</td>
<td>2.815</td>
<td>3.285</td>
<td>3.754</td>
</tr>
<tr>
<td>26</td>
<td>0.2012</td>
<td>0.2348</td>
<td>0.2683</td>
<td>104</td>
<td>2.963</td>
<td>3.457</td>
<td>3.951</td>
</tr>
<tr>
<td>29</td>
<td>0.2531</td>
<td>0.2952</td>
<td>0.3374</td>
<td>107</td>
<td>3.112</td>
<td>3.631</td>
<td>4.15</td>
</tr>
<tr>
<td>32</td>
<td>0.3103</td>
<td>0.362</td>
<td>0.4137</td>
<td>110</td>
<td>3.264</td>
<td>3.808</td>
<td>4.352</td>
</tr>
<tr>
<td>35</td>
<td>0.3928</td>
<td>0.4349</td>
<td>0.4971</td>
<td>113</td>
<td>3.417</td>
<td>3.987</td>
<td>4.556</td>
</tr>
<tr>
<td>38</td>
<td>0.4405</td>
<td>0.5139</td>
<td>0.5873</td>
<td>116</td>
<td>3.572</td>
<td>4.168</td>
<td>4.763</td>
</tr>
<tr>
<td>41</td>
<td>0.5131</td>
<td>0.5986</td>
<td>0.6841</td>
<td>119</td>
<td>3.729</td>
<td>4.351</td>
<td>4.972</td>
</tr>
<tr>
<td>44</td>
<td>0.5907</td>
<td>0.6891</td>
<td>0.7875</td>
<td>122</td>
<td>3.887</td>
<td>4.535</td>
<td>5.183</td>
</tr>
<tr>
<td>47</td>
<td>0.6729</td>
<td>0.7851</td>
<td>0.8973</td>
<td>125</td>
<td>4.047</td>
<td>4.721</td>
<td>5.395</td>
</tr>
<tr>
<td>50</td>
<td>0.7598</td>
<td>0.8865</td>
<td>1.013</td>
<td>128</td>
<td>4.207</td>
<td>4.908</td>
<td>5.610</td>
</tr>
<tr>
<td>53</td>
<td>0.8512</td>
<td>0.9931</td>
<td>1.135</td>
<td>131</td>
<td>4.369</td>
<td>5.097</td>
<td>5.825</td>
</tr>
<tr>
<td>56</td>
<td>0.947</td>
<td>1.105</td>
<td>1.263</td>
<td>134</td>
<td>4.531</td>
<td>5.287</td>
<td>6.042</td>
</tr>
<tr>
<td>59</td>
<td>1.047</td>
<td>1.221</td>
<td>1.396</td>
<td>137</td>
<td>4.695</td>
<td>5.477</td>
<td>6.260</td>
</tr>
<tr>
<td>62</td>
<td>1.151</td>
<td>1.343</td>
<td>1.535</td>
<td>140</td>
<td>4.859</td>
<td>5.669</td>
<td>6.478</td>
</tr>
<tr>
<td>65</td>
<td>1.259</td>
<td>1.469</td>
<td>1.679</td>
<td>143</td>
<td>5.023</td>
<td>5.861</td>
<td>6.698</td>
</tr>
<tr>
<td>68</td>
<td>1.371</td>
<td>1.599</td>
<td>1.828</td>
<td>146</td>
<td>5.185</td>
<td>6.053</td>
<td>6.918</td>
</tr>
<tr>
<td>71</td>
<td>1.486</td>
<td>1.734</td>
<td>1.982</td>
<td>149</td>
<td>5.354</td>
<td>6.243</td>
<td>7.138</td>
</tr>
<tr>
<td>74</td>
<td>1.605</td>
<td>1.873</td>
<td>2.14</td>
<td>152</td>
<td>5.518</td>
<td>6.438</td>
<td>7.357</td>
</tr>
<tr>
<td>77</td>
<td>1.728</td>
<td>2.016</td>
<td>2.304</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Fig (4-5) Deformed shape along center line

Table (4-7) Stress in x-direction at integration point (INT1, El (1))

<table>
<thead>
<tr>
<th>Load(N)</th>
<th>Stresses (10^3N/mm²)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>160EL</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>3000</td>
<td>1.027</td>
</tr>
<tr>
<td>6000</td>
<td>2.142</td>
</tr>
<tr>
<td>9000</td>
<td>3.344</td>
</tr>
<tr>
<td>12000</td>
<td>4.633</td>
</tr>
<tr>
<td>15000</td>
<td>6.004</td>
</tr>
<tr>
<td>18000</td>
<td>7.477</td>
</tr>
<tr>
<td>21000</td>
<td>9.024</td>
</tr>
<tr>
<td>24000</td>
<td>10.66</td>
</tr>
</tbody>
</table>

Fig (4-6) Stress in x-direction at integration point (INT 1, El 1)
Table (4-8) Stress in y-direction at integration point (INT 1, El (1))

<table>
<thead>
<tr>
<th>Load(N)</th>
<th>Stresses (10^3N/mm2)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>160EL</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>3000</td>
<td>0.339</td>
</tr>
<tr>
<td>6000</td>
<td>0.7646</td>
</tr>
<tr>
<td>9000</td>
<td>1.277</td>
</tr>
<tr>
<td>12000</td>
<td>1.876</td>
</tr>
<tr>
<td>15000</td>
<td>2.561</td>
</tr>
<tr>
<td>18000</td>
<td>3.333</td>
</tr>
<tr>
<td>21000</td>
<td>4.192</td>
</tr>
<tr>
<td>24000</td>
<td>5.138</td>
</tr>
</tbody>
</table>

Fig (4-7) stress in y-direction at integration point (INT 1, El (1))

Table (4-9) Shear stress at integration point (INT 1, El (1))

<table>
<thead>
<tr>
<th>Load(N)</th>
<th>Shear stresses (10^2N/mm2)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>160EL</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>3000</td>
<td>-1.918</td>
</tr>
<tr>
<td>6000</td>
<td>-3.836</td>
</tr>
<tr>
<td>9000</td>
<td>-5.754</td>
</tr>
<tr>
<td>12000</td>
<td>-7.673</td>
</tr>
<tr>
<td>15000</td>
<td>-9.591</td>
</tr>
<tr>
<td>18000</td>
<td>-11.51</td>
</tr>
<tr>
<td>24000</td>
<td>-15.35</td>
</tr>
</tbody>
</table>
4.2.2.2 Cantilever plate beam under pure bending

A cantilever plate subjected to pure bending moment is considered, the cantilever is of dimensions L=3000mm, D=300mm and thickness h=60mm as shown in figure (4.9). The numerical values of material property parameter are: Young's modulus $E=210 \times 10^3 N/mm^2$, Poisson's ratio $\nu = 0.3$. The structure is modeled with different number of isoparametric elements. The mesh is of equal size of 150mmx150mm, to check the accuracy of results of displacements and stresses they are compared for different numbers of elements, and with exact solutions. The results are presented in Figures (4.10) to (4.15) and tables (4.10) to (4.18). The vertical displacements with different numbers of elements are shown in tables (4.10) to (4.14) and figures (4.10) to (4.11). Tables (4.15) and Fig (4.12) show the deflected (deformed) shape along center line. The stresses at integration point are presented in tables (4.16) to (4-18) and Figures (4.13) to (4.15)
Fig (4.9) Cantilever plate under pure bending

Table (4-10) Vertical displacement (x10^-2) at free end with aspect ratio

<table>
<thead>
<tr>
<th>Load(N)</th>
<th>Max load 3000N</th>
</tr>
</thead>
<tbody>
<tr>
<td>Elements</td>
<td>20EL</td>
</tr>
<tr>
<td>Mesh</td>
<td>2*10</td>
</tr>
<tr>
<td>Aspect</td>
<td>2</td>
</tr>
<tr>
<td>Mesh</td>
<td>4*5</td>
</tr>
<tr>
<td>Aspect</td>
<td>8</td>
</tr>
</tbody>
</table>

Fig (4-10) Effect of mesh and aspect ratio on accuracy of result
### Table (4-11) Vertical displacement at free end point A

<table>
<thead>
<tr>
<th>Load(N)</th>
<th>60E</th>
<th>80E</th>
<th>100E</th>
<th>120E</th>
<th>140E</th>
<th>160E</th>
<th>180E</th>
<th>200E</th>
<th>Exact</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>3000</td>
<td>0.06678</td>
<td>0.06798</td>
<td>0.06855</td>
<td>0.06887</td>
<td>0.06918</td>
<td>0.0693</td>
<td>0.0693</td>
<td>0.071428</td>
<td></td>
</tr>
<tr>
<td>6000</td>
<td>0.1336</td>
<td>0.1360</td>
<td>0.1371</td>
<td>0.1377</td>
<td>0.138</td>
<td>0.1384</td>
<td>0.138</td>
<td>0.1387</td>
<td>0.142857</td>
</tr>
<tr>
<td>9000</td>
<td>0.203</td>
<td>0.2039</td>
<td>0.2057</td>
<td>0.2066</td>
<td>0.207</td>
<td>0.2076</td>
<td>0.207</td>
<td>0.208</td>
<td>0.214285</td>
</tr>
<tr>
<td>12000</td>
<td>0.2671</td>
<td>0.2719</td>
<td>0.2742</td>
<td>0.2755</td>
<td>0.276</td>
<td>0.2767</td>
<td>0.277</td>
<td>0.2773</td>
<td>0.285714</td>
</tr>
<tr>
<td>15000</td>
<td>0.3339</td>
<td>0.3399</td>
<td>0.3428</td>
<td>0.3443</td>
<td>0.345</td>
<td>0.3459</td>
<td>0.346</td>
<td>0.3467</td>
<td>0.357142</td>
</tr>
<tr>
<td>18000</td>
<td>0.4007</td>
<td>0.4079</td>
<td>0.4113</td>
<td>0.4132</td>
<td>0.414</td>
<td>0.4151</td>
<td>0.415</td>
<td>0.416</td>
<td>0.428571</td>
</tr>
<tr>
<td>21000</td>
<td>0.4674</td>
<td>0.4759</td>
<td>0.4799</td>
<td>0.4821</td>
<td>0.483</td>
<td>0.4843</td>
<td>0.484</td>
<td>0.4853</td>
<td>0.500000</td>
</tr>
<tr>
<td>24000</td>
<td>0.534</td>
<td>0.5438</td>
<td>0.5488</td>
<td>0.5514</td>
<td>0.552</td>
<td>0.5535</td>
<td>0.554</td>
<td>0.5547</td>
<td>0.571428</td>
</tr>
</tbody>
</table>

### Fig (4-11) Vertical displacement at free end point A

Table (4-12) Displacement at free end point A (200 El, 4inc)

<table>
<thead>
<tr>
<th>Increment</th>
<th>Load 24000N</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>V(mm)</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>1</td>
<td>1.04</td>
</tr>
<tr>
<td>2</td>
<td>2.08</td>
</tr>
<tr>
<td>3</td>
<td>3.12</td>
</tr>
<tr>
<td>4</td>
<td>4.16</td>
</tr>
</tbody>
</table>
Table (4-13) Displacement at free end point A (200 El, 8inc)

<table>
<thead>
<tr>
<th>Increment</th>
<th>Load 18000N</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>V(mm)</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>1</td>
<td>0.52</td>
</tr>
<tr>
<td>2</td>
<td>1.04</td>
</tr>
<tr>
<td>3</td>
<td>1.56</td>
</tr>
<tr>
<td>4</td>
<td>2.08</td>
</tr>
<tr>
<td>5</td>
<td>2.6</td>
</tr>
<tr>
<td>6</td>
<td>3.12</td>
</tr>
<tr>
<td>7</td>
<td>3.64</td>
</tr>
<tr>
<td>8</td>
<td>4.16</td>
</tr>
</tbody>
</table>

Table (4-14) Displacement at free end point A (200 El, 12inc)

<table>
<thead>
<tr>
<th>Increment</th>
<th>Load 18000N</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>V(mm)</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>1</td>
<td>0.347</td>
</tr>
<tr>
<td>2</td>
<td>0.695</td>
</tr>
<tr>
<td>3</td>
<td>1.041</td>
</tr>
<tr>
<td>4</td>
<td>1.387</td>
</tr>
<tr>
<td>5</td>
<td>1.733</td>
</tr>
<tr>
<td>6</td>
<td>2.08</td>
</tr>
<tr>
<td>7</td>
<td>2.427</td>
</tr>
<tr>
<td>8</td>
<td>2.77</td>
</tr>
<tr>
<td>9</td>
<td>3.12</td>
</tr>
<tr>
<td>10</td>
<td>3.467</td>
</tr>
<tr>
<td>11</td>
<td>3.813</td>
</tr>
<tr>
<td>12</td>
<td>4.16</td>
</tr>
</tbody>
</table>

Table (4-15) Deflected shape along center line (60El)

<table>
<thead>
<tr>
<th>Node</th>
<th>Load 12000N</th>
<th>Load 15000N</th>
<th>Load 18000 N</th>
<th>Node</th>
<th>Load 12000N</th>
<th>Load 15000N</th>
<th>Load 18000 N</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>80</td>
<td>20.0</td>
<td>25.07</td>
<td>30.0</td>
</tr>
<tr>
<td>5</td>
<td>0.01772</td>
<td>0.02215</td>
<td>0.02658</td>
<td>83</td>
<td>21.63</td>
<td>27.04</td>
<td>32.44</td>
</tr>
<tr>
<td>8</td>
<td>0.1066</td>
<td>0.1333</td>
<td>0.1599</td>
<td>86</td>
<td>23.26</td>
<td>29.08</td>
<td>34.9</td>
</tr>
<tr>
<td>11</td>
<td>0.2536</td>
<td>0.317</td>
<td>0.3804</td>
<td>89</td>
<td>24.96</td>
<td>31.20</td>
<td>37.4</td>
</tr>
<tr>
<td>14</td>
<td>0.4602</td>
<td>0.5753</td>
<td>0.609</td>
<td>92</td>
<td>26.71</td>
<td>33.30</td>
<td>40.0</td>
</tr>
<tr>
<td>17</td>
<td>0.7264</td>
<td>0.908</td>
<td>1.09</td>
<td>20</td>
<td>1.052</td>
<td>1.315</td>
<td>1.578</td>
</tr>
<tr>
<td>23</td>
<td>1.437</td>
<td>1.796</td>
<td>2.156</td>
<td>26</td>
<td>1.882</td>
<td>2.352</td>
<td>2.822</td>
</tr>
<tr>
<td>29</td>
<td>2.386</td>
<td>2.982</td>
<td>3.578</td>
<td>32</td>
<td>2.949</td>
<td>3.686</td>
<td>4.424</td>
</tr>
<tr>
<td>35</td>
<td>3.572</td>
<td>4.465</td>
<td>5.358</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Fig (4-12) Deformed shape along center line

Table (4-16) Stress in x-direction at integration point (El 1, INT 1)

<table>
<thead>
<tr>
<th>Load(N)</th>
<th>Stresses (10^-1N/mm²)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>160EL</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>3000</td>
<td>1.041</td>
</tr>
<tr>
<td>6000</td>
<td>2.081</td>
</tr>
<tr>
<td>9000</td>
<td>3.122</td>
</tr>
<tr>
<td>12000</td>
<td>4.163</td>
</tr>
<tr>
<td>15000</td>
<td>5.204</td>
</tr>
<tr>
<td>18000</td>
<td>6.245</td>
</tr>
<tr>
<td>21000</td>
<td>7.286</td>
</tr>
<tr>
<td>24000</td>
<td>8.327</td>
</tr>
</tbody>
</table>
**Fig (4-13)** stress in x-direction at integration point (El 1, INT 1)

**Table (4-17)** Stress in y-direction at integration point (El 1, INT 1)

<table>
<thead>
<tr>
<th>Load(N)</th>
<th>Stresses (10^-1N/mm²)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>160EL</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>3000</td>
<td>0.1909</td>
</tr>
<tr>
<td>6000</td>
<td>0.3819</td>
</tr>
<tr>
<td>9000</td>
<td>0.5728</td>
</tr>
<tr>
<td>12000</td>
<td>0.763</td>
</tr>
<tr>
<td>15000</td>
<td>0.9548</td>
</tr>
<tr>
<td>18000</td>
<td>1.146</td>
</tr>
<tr>
<td>21000</td>
<td>1.337</td>
</tr>
<tr>
<td>24000</td>
<td>1.528</td>
</tr>
</tbody>
</table>

**Fig (4-14)** stress in y-direction at integration point (El 1, INT 1)
4.2.2.3- Simply Supported Plate Beam under Distributed Uniform Load

A beam with two-simple supported end is considered. The beam is of length L=200mm, height D=10mm, and thickness 1mm, as shown in Fig (4.16). The numerical values for material property parameters are Young's modulus, E=210GPa, Poisson's ratio, v=0.3. The beam is modeled with a different numbers of mesh sizes.

The variation of the displacement and max deflection at the mid-span with different numbers of load increment are computed using Green formulation, and compared with the exact solution. Also the stresses at integration point

Table (4-18) Shear Stress at integration point (El 1, INT 1)

<table>
<thead>
<tr>
<th>Load(N)</th>
<th>Shear stresses (10^-1N/mm²)</th>
<th>160EL</th>
<th>180EL</th>
<th>200EL</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td></td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>3000</td>
<td></td>
<td>-0.1257</td>
<td>-1.549</td>
<td>-0.1792</td>
</tr>
<tr>
<td>6000</td>
<td></td>
<td>-0.2514</td>
<td>-0.3097</td>
<td>-0.3584</td>
</tr>
<tr>
<td>9000</td>
<td></td>
<td>-0.3771</td>
<td>-0.4646</td>
<td>-0.5376</td>
</tr>
<tr>
<td>12000</td>
<td></td>
<td>-0.5029</td>
<td>-0.6195</td>
<td>-0.7168</td>
</tr>
<tr>
<td>15000</td>
<td></td>
<td>-0.6286</td>
<td>-0.7743</td>
<td>-0.896</td>
</tr>
<tr>
<td>18000</td>
<td></td>
<td>-0.7543</td>
<td>-0.9292</td>
<td>-1.075</td>
</tr>
<tr>
<td>21000</td>
<td></td>
<td>-0.88</td>
<td>-1.084</td>
<td>-1.254</td>
</tr>
<tr>
<td>24000</td>
<td></td>
<td>-1.06</td>
<td>-1.23</td>
<td>-1.434</td>
</tr>
</tbody>
</table>

Fig (4-15) Shear stress at integration point (El 1, INT 1)
calculated for the different loading increment. Tables (4.20) and Fig (4.17) show the vertical displacements at mid-span, Table (4.12) and Fig (4.18) show the Deformation shape along center line. The stresses at integration point presented in Tables (4.22) to (4.23), Fig (4.19) to (4.20).

\[ Y \]

\[ Q/ m \]

\[ D=10\text{mm} \]

\[ L=200\text{mm} \]

**Fig (4-16)** simple supported plate beam under distributed uniform load

<table>
<thead>
<tr>
<th>Load(N)</th>
<th>Max load 1000N</th>
</tr>
</thead>
<tbody>
<tr>
<td>Elements</td>
<td>80EL 100EL 120EL 140EL 180EL 220EL 260EL 300EL</td>
</tr>
<tr>
<td>U(mm)</td>
<td>0 0 0 0 0 0 0 0</td>
</tr>
</tbody>
</table>

**Table (4-19) Vertical displacement at the mid span**

<table>
<thead>
<tr>
<th>Load(N)</th>
<th>Vertical displacement(mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>220EL</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>1000</td>
<td>-4.243</td>
</tr>
<tr>
<td>3000</td>
<td>-12.73</td>
</tr>
<tr>
<td>4000</td>
<td>-16.97</td>
</tr>
<tr>
<td>5000</td>
<td>-21.21</td>
</tr>
<tr>
<td>6000</td>
<td>-25.46</td>
</tr>
<tr>
<td>7000</td>
<td>-29.7</td>
</tr>
<tr>
<td>8000</td>
<td>-33.94</td>
</tr>
</tbody>
</table>

**Table (4-20) Vertical displacement at mid span**
**Fig (4-17)** Vertical displacement at mid span

Table (4-21) Deflected shape along center line (220El)

<table>
<thead>
<tr>
<th>Node</th>
<th>Load 4000N</th>
<th>Load 5000 N</th>
<th>Node</th>
<th>Load 4000N</th>
<th>Load 5000 N</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>0.05241</td>
<td>0.06551</td>
<td>143</td>
<td>-16.94</td>
<td>-21.18</td>
</tr>
<tr>
<td>8</td>
<td>-0.5564</td>
<td>-0.6955</td>
<td>148</td>
<td>-16.82</td>
<td>-21.03</td>
</tr>
<tr>
<td>13</td>
<td>-1.208</td>
<td>-1.51</td>
<td>153</td>
<td>-16.62</td>
<td>-20.78</td>
</tr>
<tr>
<td>18</td>
<td>-1.907</td>
<td>-2.383</td>
<td>158</td>
<td>-16.35</td>
<td>-20.43</td>
</tr>
<tr>
<td>23</td>
<td>-2.639</td>
<td>-3.299</td>
<td>163</td>
<td>-16.0</td>
<td>-20.0</td>
</tr>
<tr>
<td>33</td>
<td>-4.188</td>
<td>-5.236</td>
<td>173</td>
<td>-15.10</td>
<td>-18.88</td>
</tr>
<tr>
<td>38</td>
<td>-4.994</td>
<td>-6.243</td>
<td>178</td>
<td>-14.57</td>
<td>-18.21</td>
</tr>
<tr>
<td>43</td>
<td>-5.814</td>
<td>-7.268</td>
<td>183</td>
<td>-13.99</td>
<td>-17.48</td>
</tr>
<tr>
<td>48</td>
<td>-6.642</td>
<td>-8.303</td>
<td>188</td>
<td>-13.36</td>
<td>-16.69</td>
</tr>
<tr>
<td>63</td>
<td>-9.125</td>
<td>-11.41</td>
<td>203</td>
<td>-11.24</td>
<td>-14.05</td>
</tr>
<tr>
<td>68</td>
<td>-9.935</td>
<td>-12.42</td>
<td>208</td>
<td>-10.47</td>
<td>-13.09</td>
</tr>
<tr>
<td>73</td>
<td>-10.73</td>
<td>-13.41</td>
<td>213</td>
<td>-9.69</td>
<td>-12.11</td>
</tr>
<tr>
<td>78</td>
<td>-11.49</td>
<td>-14.37</td>
<td>218</td>
<td>-8.89</td>
<td>-11.11</td>
</tr>
<tr>
<td>83</td>
<td>-12.23</td>
<td>-15.29</td>
<td>223</td>
<td>-8.079</td>
<td>-10.10</td>
</tr>
<tr>
<td>88</td>
<td>-12.94</td>
<td>-16.18</td>
<td>228</td>
<td>-7.263</td>
<td>-9.079</td>
</tr>
<tr>
<td>93</td>
<td>-13.61</td>
<td>-17.01</td>
<td>233</td>
<td>-6.447</td>
<td>-8.05</td>
</tr>
<tr>
<td>98</td>
<td>-14.23</td>
<td>-17.79</td>
<td>238</td>
<td>-5.636</td>
<td>-7.046</td>
</tr>
<tr>
<td>103</td>
<td>-18.8</td>
<td>-18.50</td>
<td>243</td>
<td>-4.836</td>
<td>-6.044</td>
</tr>
<tr>
<td>108</td>
<td>-15.32</td>
<td>-19.15</td>
<td>248</td>
<td>-4.05</td>
<td>-5.062</td>
</tr>
<tr>
<td>118</td>
<td>-16.17</td>
<td>-20.21</td>
<td>258</td>
<td>-2.544</td>
<td>-3.180</td>
</tr>
<tr>
<td>1123</td>
<td>-16.49</td>
<td>-20.61</td>
<td>263</td>
<td>-1.834</td>
<td>-2.293</td>
</tr>
<tr>
<td>128</td>
<td>-16.73</td>
<td>-20.92</td>
<td>268</td>
<td>-1.159</td>
<td>-1.449</td>
</tr>
<tr>
<td>133</td>
<td>-16.9</td>
<td>-21.12</td>
<td>273</td>
<td>-0.5314</td>
<td>-0.6643</td>
</tr>
<tr>
<td>138</td>
<td>-16.97</td>
<td>-21.21</td>
<td>278</td>
<td>-0.0533</td>
<td>-0.0674</td>
</tr>
</tbody>
</table>
**Fig (4-18)** Deformed shape along center line

Table (4-22) Stress in x-direction at integration point at mid-span (INT 1)

<table>
<thead>
<tr>
<th>Load(N)</th>
<th>Stresses(N/mm²)</th>
<th>220EL</th>
<th>300EL</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>1000</td>
<td>0.0367</td>
<td>0.04779</td>
<td></td>
</tr>
<tr>
<td>2000</td>
<td>0.1818</td>
<td>0.2589</td>
<td></td>
</tr>
<tr>
<td>3000</td>
<td>0.4352</td>
<td>0.6332</td>
<td></td>
</tr>
<tr>
<td>4000</td>
<td>0.7968</td>
<td>1.17</td>
<td></td>
</tr>
<tr>
<td>5000</td>
<td>1.267</td>
<td>1.872</td>
<td></td>
</tr>
<tr>
<td>6000</td>
<td>1.845</td>
<td>2.736</td>
<td></td>
</tr>
<tr>
<td>7000</td>
<td>2.532</td>
<td>3.76</td>
<td></td>
</tr>
<tr>
<td>8000</td>
<td>3.327</td>
<td>4.954</td>
<td></td>
</tr>
</tbody>
</table>

**Fig (4-19)** stress in x-direction at integration point at mid-span (INT 1)
Table (4-23) Stress in x-direction at integration point at mid-span (INT 1)

<table>
<thead>
<tr>
<th>Load(N)</th>
<th>Stresses(N/mm²)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>260EL</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>1000</td>
<td>-0.218</td>
</tr>
<tr>
<td>2000</td>
<td>-0.6949</td>
</tr>
<tr>
<td>3000</td>
<td>-1.43</td>
</tr>
<tr>
<td>4000</td>
<td>-2.425</td>
</tr>
<tr>
<td>5000</td>
<td>-3.678</td>
</tr>
<tr>
<td>6000</td>
<td>-5.189</td>
</tr>
<tr>
<td>7000</td>
<td>-6.90</td>
</tr>
<tr>
<td>8000</td>
<td>-8.989</td>
</tr>
</tbody>
</table>

Fig (4-20) Shear stress in x-direction at integration point at mid-span (INT 1)
4.2.3 Nonlinear analysis:

4.2.3.1 Clamped plate beam under central Point load

A plate beam with two-fixed ends is considered. The plate beam is of length $L=200\text{mm}$, depth $D=10\text{mm}$ and thickness $1\text{mm}$ as shown in Fig (4.15). The numerical values of material property parameter are: Young's modulus $E=210\times10^3\text{N/mm}^2$, Poisson's ratio $\nu = 0.3$. The structure is modeled with different number of elements. The mesh is of rectangular size of $20\times5\text{mm}$. The displacements, direct stresses and shear stresses obtained were compared for different numbers of elements and known results. The results are presented in Tables (4.24) and (4.25) and Fig (4.22) to (4.23). The vertical displacements at mid span are shown in Table (4.24) and Fig (4.22). Table (4.25) and Fig (4.23) show the deflected (deformed) shape along center line.

![Fig (4-21) Clamped plate beam under concentrated load](image)

Table (4-24) Displacement at mid-span

<table>
<thead>
<tr>
<th>Load(N)</th>
<th>GREEN</th>
<th>Ref(3)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>25</td>
<td>0.062</td>
<td>0.062</td>
</tr>
<tr>
<td>50</td>
<td>0.1067</td>
<td>0.126</td>
</tr>
<tr>
<td>100</td>
<td>0.231</td>
<td>0.253</td>
</tr>
<tr>
<td>175</td>
<td>0.50</td>
<td>0.449</td>
</tr>
<tr>
<td>287.5</td>
<td>0.8</td>
<td>0.751</td>
</tr>
<tr>
<td>456.25</td>
<td>1.26</td>
<td>1.122</td>
</tr>
</tbody>
</table>
**Fig (4-22) Vertical displacement at mid-span**

**Table (4-25) Deformed shape along center line (10^-1mm)**

<table>
<thead>
<tr>
<th>Node</th>
<th>Load 287.5N</th>
<th>Load 456.25N</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>5</td>
<td>-0.7487</td>
<td>-1.123</td>
</tr>
<tr>
<td>8</td>
<td>-2.317</td>
<td>-3.885</td>
</tr>
<tr>
<td>11</td>
<td>-4.208</td>
<td>-7.215</td>
</tr>
<tr>
<td>14</td>
<td>-6.154</td>
<td>-10.29</td>
</tr>
<tr>
<td>17</td>
<td>-8.0</td>
<td>-12.67</td>
</tr>
<tr>
<td>23</td>
<td>-10.42</td>
<td>-14.1</td>
</tr>
<tr>
<td>26</td>
<td>-9.515</td>
<td>-14.14</td>
</tr>
<tr>
<td>29</td>
<td>-5.054</td>
<td>-3.626</td>
</tr>
<tr>
<td>32</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>
Fig (4-23) Deformed shape along center line

4.2.3.2 Cantilever plate beam under pure bending

A cantilever plate subjected to pure bending moment is adopted. The cantilever is of dimensions \( L=3000\text{mm}, D=300\text{mm} \) and thickness \( h=60\text{mm} \) as shown in Fig (4.24). The numerical values of material property parameter are: Young's modulus \( E=210 \times 10^3 \text{N/mm}^2 \), Poisson's ratio \( \nu = 0.3 \). The structure is modeled with different numbers of isoparametric elements. The mesh is of equal size of 150\text{mmx150mm}, to check results the displacements and stresses compared for different numbers of elements, and the known result. The results are shown in Tables (4.26) and (4.27) and Fig (4.4.25) and (4.26). Table (4.26) and Fig (4.25) show the vertical displacement at free end. Table (4.27) and Fig (4.25) show the deflected (deformed) shape along center line.

![Diagram of Cantilever Plate](image)

\( \text{Fig (4-24) Cantilever plate under pure bending} \)
Table (4-26) Vertical displacement at free end

<table>
<thead>
<tr>
<th>Load(N)</th>
<th>GREEN</th>
<th>GREEN Ref(3)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>3000</td>
<td>99.23</td>
<td>99.9727</td>
</tr>
<tr>
<td>6000</td>
<td>230.3</td>
<td>199.242</td>
</tr>
<tr>
<td>9000</td>
<td>344.2</td>
<td>297.087</td>
</tr>
<tr>
<td>12000</td>
<td>418</td>
<td>392.848</td>
</tr>
<tr>
<td>15000</td>
<td>493.5</td>
<td>485.959</td>
</tr>
</tbody>
</table>

**Fig (4-25)** Vertical displacement at free end

Table (4-27) Deflected shape along center line

<table>
<thead>
<tr>
<th>Node</th>
<th>GREEN3000N</th>
</tr>
</thead>
<tbody>
<tr>
<td>35</td>
<td>2.53</td>
</tr>
<tr>
<td>38</td>
<td>6.563</td>
</tr>
<tr>
<td>41</td>
<td>11.01</td>
</tr>
<tr>
<td>44</td>
<td>15.79</td>
</tr>
<tr>
<td>47</td>
<td>20.81</td>
</tr>
<tr>
<td>50</td>
<td>26</td>
</tr>
<tr>
<td>53</td>
<td>31.25</td>
</tr>
<tr>
<td>56</td>
<td>36.49</td>
</tr>
<tr>
<td>59</td>
<td>41.62</td>
</tr>
<tr>
<td>62</td>
<td>46.55</td>
</tr>
<tr>
<td>65</td>
<td>51.22</td>
</tr>
<tr>
<td>58</td>
<td>55.53</td>
</tr>
<tr>
<td>71</td>
<td>59.43</td>
</tr>
<tr>
<td>74</td>
<td>62.84</td>
</tr>
<tr>
<td>77</td>
<td>65.72</td>
</tr>
<tr>
<td>80</td>
<td>68.03</td>
</tr>
<tr>
<td>83</td>
<td>69.71</td>
</tr>
<tr>
<td>86</td>
<td>70.7</td>
</tr>
<tr>
<td>89</td>
<td>71.14</td>
</tr>
</tbody>
</table>
4.3 Discussion of Results

With reference to the results obtained from the application of the program, it can be stared that:

1- For the cantilever under tip load at free end the results are presented in table (4.1) to Table (4.9), and the Fig (4.1) to Fig (4.8), which demonstrate the efficiency of the linear analysis program.

Table (4.1) to Table (4.2) and Fig (4.3) show that the vertical displacements at free end for various applied loads and different numbers of elements are in agreement. The effect of aspect ratio and mesh refinement in the results are agreed with those presented by (Bhavikatti, 2005).

Fig (4.4) and Table (4.3) show that the displacement at point A at free end of structure, are in close agreement with that in exact solutions, Also, Fig (4.5) and Table (4.5) to Table (4.6) demonstrate that the deflected shape of the beam along center line and the displacement at free end. The stresses at the integration point show a good agreement for different numbers of elements and various applied loads.
Fig (4.3), Table (4.4), and Table (4.5) show the vertical displacement at free end for 5, 10, and 15 load increments and clearly demonstrate the accuracy of linear analysis results which are the same for all increments.

2- In the second example for linear analysis, the plate beam subjected to pure bending as shown in Fig (4.9) is analyzed. Fig (4.10) to Fig (15) and Table (4.10) to Table (4.18) show that the different values of displacements at the free end and direct stresses and shear stresses at integration points. Also, all values of the displacement and stresses are in very good agreement with exact solution. Fig (4-10) shows clearly the effect of the aspect ratio on the convergence of the results to the exact solution.

The numerical example is the simply supported plate beam under uniform load as in Fig (4-16). Fig (4-17) and Table (4-24) show the vertical displacement at the mid-span which is computed for different elements and different applied loads. The deformed shape along center line of beam is plotted in Fig (4-25), and the stresses at integration point are listed in Tables (4-25) and (4-26). All results are in good agreement with the exact solution.

3- For the nonlinear analysis, two numerical examples tested, cantilever plate beam subjected to pure bending, and clamped plat beam under concentrated load as shown in Fig (4.13) and Fig (4.14) respectively.

For the clamped plate beam, variation of displacement at mid-span with different load increment, and different iterations computed from the Green formulation as shown in Fig (4-22) and Table (4-24). The values of vertical displacements compared with those presented by (Akasha and Mohamed, 2012), these show good agreement. Also Fig (4-23) and Table (4-25) demonstrate the deformed shape of the beam and the effect of nonlinearity on the results.

For the case of the plate beam under pure bending, the displacement computed from Green formulation was compared with the results presented by (Akasha and Mohamed, 2012), as shown in Fig (4-25) and Table (4-26). These
show good agreement. Fig (4-26) and Table (4-28) represent the deformed shape for parts of the beam. The values demonstrate behavior of the structures that subjected to large deformation with different applied loads.
Chapter Five

Conclusions and recommendations

5.1 Summary of work

In this research the overall work can be summarized as follows:

1- The finite element computer program using standard MATLAB (2010b) language for analysis of static linear and nonlinear plane stresses/strain structures have been developed.

2- The geometrically nonlinear formulation has been developed based on Total Lagrangian approach and Green strain using 2D plane stress/strain 4-node isoparametric element.

3- The formulations for linear and geometrically nonlinear plane stress/strain based on Green stress were implemented on computer using MATLAB as program, LINA2D for linear analysis and program NONLIA2D for nonlinear analysis.

4- The computer programs were tested using numerical examples and the results obtained were compared with published results.

5.2 Conclusions

Based on the results of numerical examples, it can be concluded that:

1- For the linear analysis the values of displacement, for any number of elements and various applied loads were in excellent agreement when compared with known results and exact solution. For cantilever under tip load, the maximum difference between the exact solution and program solution is about 6%, for mesh of 200 elements and with aspect ratio equal to 0.2. Also for the cantilever under pure bending and simple supported beam under udl, the maximum percentage differences between exact solution and linear program solution are 2.7% and 0.8% respectively.
2- For the nonlinear analysis the formulation based on Green strain give a good agreement when compared with known results, for various load increments and different iterations numbers. In case of clamped plate beam the difference between program results and those presented by (Akasha and Mohamed, 2012) is about 12%.

5.3 Recommendations

From the research results obtained, the following is recommended:

(1) Recommendation from study:-

1- Use of MATLAB programming language in developing finite element programs for its accuracy and ease of use.

2- Use of the program LINA2D in linear analysis of plane strain/stresses problems because it gives highly accurate results.

(2) Recommendation for future study:-

1- Modification of the program to calculate the average nodal stresses from the elements Gauss point results.

2- Developing the formulation to cover the gravity load and surface load

3- Developing the program to include higher order elements.

4- Extension of the nonlinear formulation to include engineering strain and logarithmic strain.

5- Developing the nonlinear formulations by using Updated Lagrangian formulation approach.

6- Extension of formulations to include nonlinear material, elastoplastic and hyperelastic materials.
REFERENCES


Appendix A

The main function of program

clc
clear
clear all

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% %------the main program to solve the nonlinear program------%
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%

% global a0 %SIGMA
global NUMNP NE  GXY LE NXE NYE dhx dhy X_0 Y_0
% inputs data
L=input('L=');
W=input('W=');
NXE=input('NXE=');
NYE=input('NYE=');
thick=input('thick=');
dhx=L/NXE;
dhy=W/NYE;
X_0=0;
Y_0=W/2;
% nodal coordinates and Element connectivity
[NUMNP, NDOF] = size(GXY);
NE = size(LE,1);
NEQ = NDOF*NUMNP;
% Material properties
PROP=[210000 0.3];
% Prescribed displacements [Node, DOF, Value]
pn1=((NXE+1)*(NYE+1)-2);
pn2=((NXE+1)*(NYE+1)-1);
pn3=(NXE+1)*(NYE+1);
SDISPT=[1 1 0;1 2 0;2 1 0;2 2 0;3 1 0;3 2 0];
% program parameters
TOL=1E-3;
% External forces [Node, DOF, Value]
EXF=input('EXF=');
xyD=input('xyD=');
%Fn1=((NXE+1)*(NYE+1)-2);
%Fn2=(NXE+1)*(NYE+1);
EXTFORCE=[Fn1 xyD EXF;Fn2 xyD -EXF];

% Check element connectivity
XYZON(GXY, LE, NOUT);
% Elastic modulus material properties
MID=1;
D=FORD(PROP,MID);
% initialize local and global matrixes and vectors
FG=zeros(NEQ,1);
GKT = zeros(NEQ,NEQ);
FORCE=zeros(NEQ,1);
EINF=zeros(8,1);
GINF=zeros(NEQ,1);
STRESS=zeros(3,1);
STRESS1=zeros(3,1);
DltS=zeros(3,1);
BL=zeros(3,8);
EKO=zeros(8,8);
EKL=zeros(8,8);
EKTs0=zer0s(8,8);
KET=zeros(8,8);
INC=input('INC=');
FACTOR=1/INC;
ITRA=input('ITER=');
AnaType=input('AnaType=');
a0=zeros(NEQ,1);
Dlt0=zeros(NEQ,1);

for Li=1:INC
% load
FG=zeros(NEQ,1);
% Assemble the global increment and external force vector
if size(EXTFORCE,1) > 0
    LOC = ND0F*(EXTFORCE(:,1)-1)+EXTFORCE(:,2);
    Linc=FACTOR*EXTFORCE(:,3);
    FG(LOC) = FG(LOC) + Linc;
end
Dlt0=zeros(NEQ,1);
for ITER=1:ITRA % Start convergence iteration (Newton-Raphson)
   if AnaType==1
      FORCE=FG;
   else
      FORCE=FG-GINF;
   end
   % Integration points and weights
   XG=[-0.577350269189630;0.577350269189630;0.577350269189630
       -0.577350269189630;0.577350269189630;0.577350269189630
       0.577350269189630-0.577350269189630;0.577350269189630
       0.577350269189630;0.577350269189630;
   WGT=[1 1 1 1];
   GKT = zeros(NEQ,NEQ);
   % LOOP OVER ELEMENTS, MAIN LOOP TO COMPUTE GKT
   for IE=1:NE
      % Element Nodal coordinates
      ELXY=GXY(LE(IE,:),:);
      % from Local to global
      EKT=zeros(8,8);
      % ID0F=zeros(1,8);
for I=1:4
II=(I-1)*NDOF+1;
IDOF=II:II+1=(LE(IE,I)-1)*NDOF+1:(LE(IE,I)-1)*NDOF+2;
end
DSP=a0(IDOF);               % displacement vector
DSPD=Dlta0(IDOF);           % displacement increment vector
%LOOP OVER INTEGRATION POINTS
for LX=1:4
E1=XG(LX,:);
% Determinant and shape function derivatives
[DSF,SHPD,DET] = SHAPEL(E1, ELXY);
FAC=thick*DET*WGT(LX);
%
Bo=[SHPD(1,1) 0          SHPD(1,2) 0         SHPD(1,3) 0
  SHPD(1,4) 0          SHPD(2,1) 0          SHPD(2,2) 0
  SHPD(2,3) 0          SHPD(2,4) 0          SHPD(2,1)
  SHPD(2,2) 0          SHPD(2,3) 0          SHPD(2,4)];
%
G=[SHPD(1,1) 0          SHPD(1,2) 0         SHPD(1,3) 0
  SHPD(1,4) 0          SHPD(1,1) 0          SHPD(1,2) 0
  SHPD(1,3) 0          SHPD(1,4) 0          SHPD(1,1)
  SHPD(2,1) 0          SHPD(2,2) 0          SHPD(2,3) 0
  SHPD(2,4) 0          SHPD(2,1) 0          SHPD(2,2) 0
  SHPD(2,3) 0          SHPD(2,4)];
%
% to forming thata
tha=tha=G*DSP;   % forming th ata
dtha =G*DSPD;   % forming Dlta th ata
% to formin A th ata
A=[tha(1) tha(2) 0 0;0 0 tha(3) tha(4);tha(3)
  tha(4) tha(1) tha(2)];
da=[dtha(1) dtha(2) 0 0;0 0 dtha(3) dtha(4);dtha(3)
  dtha(4) dtha(1) dtha(2)];
% the strain displacement matrix B
BL=A*G;
BLd=dA*G;
B=Bo+BL;
%
% the strain increment
DltaE=(Bo+BL+0.5*BLd)*DSPD;
% Stress incremental
DltaS=D*DltaE;
% Total Stresses
STRESS=STRESS+DltaS;
% stress stiffness matrix
M=[STRESS(1) 0 STRESS(3) 0;0 STRESS(1) 0 STRESS(3);STRESS(3) 0
  STRESS(2) 0;0 STRESS(3) 0 STRESS(2)];
% Tangent stiffness Matrix
EK0=FAC*Bo'*D*Bo;
EKL=FAC*Bo'*D*BL+BL'*D*Bo+BL'*D*BL;
EKTsigma=FAC*G'*M*G; % Element tangent stiffness matrix
if AnaType==1
EKT=EKT+EK0;
else
EKT=EKT+EK0+EKL+EKTsigma;
end
end
GKT(IDOF,IDOF)=GKT(IDOF,IDOF)+EKT; % Global tangent stiffness matrix
end

% Prescribed displacement BC
NDISP=size(SDISPT,1);
if NDISP~=0
FIXEDDOF=NDOF*(SDISPT(:,1)-1)+SDISPT(:,2);
GKT(FIXEDDOF,:)=zeros(NDISP,NEQ);
GKT(:,FIXEDDOF)=zeros(NEQ,NDISP);
GKT(FIXEDDOF,FIXEDDOF)=eye(NDISP);
FORCE(FIXEDDOF)=0;
end
% Check convergence
if (ITER>=0)
Conve=norm(FORCE,inf);
OUTPUT_1( ITER, Conve)
if ITER<ITRA && Conve<TOL
break;
end
end
% Check max iteration
if(ITER>ITRA), error('Iteration limit exceeds');
end

% Solve the system equation(the incremental displacement)
Dlta0= GKT\FORCE;
a0=a0+Dlta0;
% Dlta01=Dlta0;
a01=a0;
GINF=zeros(NEQ,1);
INT=0;
% assemble Global atiffness matrix GINF
for IE=1:NE
% Element Nodal coordinates and incremental displacements
ELXY=GXY(LE(IE,:),:);
% Local to global
EINF=zeros(8,1);
IDOF=zeros(1,8);
for I=1:4
II=(I-1)*NDOF+1;
IDOF(II:II+1)=(LE(IE,I)-1)*NDOF+1:(LE(IE,I)-1)*NDOF+2;
end
DSP1=a01(IDOF);             % displacement vector
DSPD1=Dlta01(IDOF);           % displacement increment vector

%LOOP OVER INTEGRATION POINTS
for LX=1:4
E1=XG(LX,:);       % this values using to calculate component
% Determinant and shape function derivatives
[DSF,SHPD,DET] = SHAPEL(E1, ELXY);
FAC=thick*DET*WGT(LX);
INT=INT+1;
Bo1=[SHPD(1,1) 0          SHPD(1,2) 0         SHPD(1,3) 0
     SHPD(1,4) 0        SHPD(2,1) 0         SHPD(2,2) 0
     SHPD(2,3) 0        SHPD(2,1) 0         SHPD(2,2) 0
     SHPD(2,4) 0        SHPD(2,1) SHPD(1,1)
                     SHPD(2,4) SHPD(1,2) SHPD(2,3)
                     SHPD(2,3) SHPD(2,4)
                     SHPD(1,3) SHPD(1,4)];
G01=[SHPD(1,1) 0         SHPD(1,2) 0         SHPD(1,3) 0
     SHPD(1,4) 0          SHPD(1,1) 0         SHPD(1,2) 0
     SHPD(1,3) 0          SHPD(1,1) 0         SHPD(1,2) 0
     SHPD(2,1) 0          SHPD(2,1) 0         SHPD(2,2) 0
     SHPD(2,4) 0          SHPD(2,4) 0         SHPD(2,2) 0
     SHPD(2,3) 0          SHPD(2,3) 0         SHPD(2,4)];

% to forming thata
thata1=G01*DSP1;    % forming thata
dthata1 =G01*DSPD1;   % forming Dlta thata

% to formin A thata
A1=[thata1(1) thata1(2) 0 0;0 0 thata1(3) thata1(4);thata1(3)
     thata1(4) thata1(1) thata1(2)];
dA1=[dthata1(1) dthata1(2) 0 0;0 0 dthata1(3)
     dthata1(4);dthata1(3) dthata1(4) dthata1(1) dthata1(2)];

% the strain displacement matrix B
BL1=A1*G01;
BLd1=dA1*G01;
B1=Bo+BL1;

% the strain increment
DltE1=(Bo+BL1+BLd1)*DSPD1;

% Stress incremental
DltS1=D*DltE1;
% Total Stresses
STRESS1=STRESS1+DltS1;
SIGMA(:,INT)=STRESS1;
% Residual internal forces
EINF=EINF+FAC*B1'*STRESS1;
end
end
GINF(IDOF) = GINF(IDOF) + EINF;  % Global internal forces
end

% print results
PROUT_1(NOUT, NUMNP, NDOF, NE);
end
Appendix B

Example of input data and print out of results

Input Data:
L=3000
W=300
NXE=5
NYE=2
thick=60
EXF=1500
xyD=1
INC=1
ITER=1
AnaType=1

Output results:

Iteration Residual

Nodal Displacements

<table>
<thead>
<tr>
<th>Node</th>
<th>U</th>
<th>V</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.000e+000</td>
<td>0.000e+000</td>
</tr>
<tr>
<td>2</td>
<td>0.000e+000</td>
<td>0.000e+000</td>
</tr>
<tr>
<td>3</td>
<td>0.000e+000</td>
<td>0.000e+000</td>
</tr>
<tr>
<td>4</td>
<td>5.515e-004</td>
<td>1.116e-003</td>
</tr>
<tr>
<td>5</td>
<td>8.485e-018</td>
<td>1.090e-003</td>
</tr>
<tr>
<td>6</td>
<td>-5.515e-004</td>
<td>1.116e-003</td>
</tr>
<tr>
<td>7</td>
<td>1.110e-003</td>
<td>4.437e-003</td>
</tr>
<tr>
<td>8</td>
<td>1.498e-017</td>
<td>4.417e-003</td>
</tr>
<tr>
<td>9</td>
<td>-1.110e-003</td>
<td>4.437e-003</td>
</tr>
</tbody>
</table>
10  1.667e-003  9.993e-003  
11  2.639e-017  9.971e-003  
12  -1.667e-003  9.993e-003  
13  2.225e-003  1.778e-002  
14  3.739e-017  1.776e-002  
15  -2.225e-003  1.778e-002  
16  2.782e-003  2.779e-002  
17  2.850e-017  2.777e-002  
18  -2.782e-003  2.779e-002  

Element Stress

<table>
<thead>
<tr>
<th>Element</th>
<th>S1</th>
<th>S2</th>
<th>S12</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>4.227e-002</td>
<td>4.912e-003</td>
<td>8.471e-002</td>
</tr>
<tr>
<td></td>
<td>8.454e-002</td>
<td>9.828e-003</td>
<td>1.694e-001</td>
</tr>
<tr>
<td></td>
<td>1.268e-001</td>
<td>1.474e-002</td>
<td>2.541e-001</td>
</tr>
<tr>
<td></td>
<td>1.691e-001</td>
<td>1.966e-002</td>
<td>3.388e-001</td>
</tr>
<tr>
<td>2</td>
<td>4.339e-003</td>
<td>-2.199e-002</td>
<td>4.256e-001</td>
</tr>
<tr>
<td></td>
<td>-1.604e-001</td>
<td>-6.364e-002</td>
<td>5.123e-001</td>
</tr>
<tr>
<td></td>
<td>-3.251e-001</td>
<td>-1.053e-001</td>
<td>5.991e-001</td>
</tr>
<tr>
<td></td>
<td>-4.899e-001</td>
<td>-1.469e-001</td>
<td>6.858e-001</td>
</tr>
<tr>
<td>3</td>
<td>-4.559e-001</td>
<td>-1.715e-001</td>
<td>7.729e-001</td>
</tr>
<tr>
<td></td>
<td>-4.219e-001</td>
<td>-1.961e-001</td>
<td>8.601e-001</td>
</tr>
<tr>
<td></td>
<td>-3.879e-001</td>
<td>-2.206e-001</td>
<td>9.472e-001</td>
</tr>
<tr>
<td></td>
<td>-3.539e-001</td>
<td>-2.452e-001</td>
<td>1.034e+000</td>
</tr>
<tr>
<td>4</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Element</td>
<td>Coefficient 1</td>
<td>Coefficient 2</td>
<td>Coefficient 3</td>
</tr>
<tr>
<td>---------</td>
<td>------------------------</td>
<td>------------------------</td>
<td>------------------------</td>
</tr>
<tr>
<td>5</td>
<td>-9.499e-001</td>
<td>-3.127e-001</td>
<td>1.467e+000</td>
</tr>
<tr>
<td>6</td>
<td>-1.001e+000</td>
<td>-3.836e-001</td>
<td>1.813e+000</td>
</tr>
<tr>
<td>7</td>
<td>-1.445e+000</td>
<td>-4.626e-001</td>
<td>2.160e+000</td>
</tr>
<tr>
<td>8</td>
<td>-1.497e+000</td>
<td>-5.372e-001</td>
<td>2.507e+000</td>
</tr>
<tr>
<td>9</td>
<td>-1.939e+000</td>
<td>-6.097e-001</td>
<td>2.853e+000</td>
</tr>
<tr>
<td></td>
<td>-1.656e+000</td>
<td>-5.553e-001</td>
<td>2.593e+000</td>
</tr>
<tr>
<td></td>
<td>-1.815e+000</td>
<td>-5.733e-001</td>
<td>2.680e+000</td>
</tr>
<tr>
<td></td>
<td>-1.975e+000</td>
<td>-5.914e-001</td>
<td>2.767e+000</td>
</tr>
<tr>
<td>Element</td>
<td>10</td>
<td></td>
<td></td>
</tr>
<tr>
<td>---------</td>
<td>----</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

-1.867e+000 -6.463e-001  3.026e+000
-1.831e+000 -6.646e-001  3.113e+000
-1.991e+000 -6.831e-001  3.200e+000
-2.150e+000 -7.016e-001  3.286e+000
-2.309e+000 -7.201e-001  3.373e+000
-2.468e+000 -7.386e-001  3.460e+000