RESEARCH ARTICLE OPEN ACCESS

Development of an Interactive Finite Element Solution Module for 2d Stress Problem Analysis Using Isoparametric Element

K.M. Odunfa, O.P. Akinmolayan
Mechanical Engineering Department, University Of Ibadan, Ibadan, Oyo State
Corresponding Auther: K.M. Odunfa

ABSTRACT
Finite Element Method Has Been Established As One Of The Most Versatile Tool Commonly Employ In Solving Complex Engineering Problems. In This Work, Finite Element Analysis Module For Stress Problem Using Isoparametric Element Was Developed And Utilized In Some Selected Engineering Problems. Two-Dimensional Standard Deflection (Flexural) Equation Was Used In The Modelling. Isoparametric 8-Noded Element With 16 Degree Of Freedom Was Also Adopted In The Model. The Domain Was Discretized Into Six And Nine Elements For The Cases Analyzed. Matlab Code Was Written Based On The Developed Model. Graphic User Interface (Gui) Was Also Developed In Other To Make The Analysis Easier For The User. A Windows Application Was Created From Matlab Standalone App In Gui For Easy Access Of The Module. Case Studies For Curvilinear Structural Element Were Used To Test-Run The Developed Program. Windows Application Was Then Run With The Required Input Data. Global Displacement Were Obtained For Each Nodal Element (For Case One, Node 8 Experienced - 0.3371m And - 0.6663m In X And Y Direction, Respectively While For Case Two, Node 8 Experienced - 0.021m And 0.14m, Respectively). Also, The Global Node Reactions Were Obtained For Each Nodal Element (For Case One, Node 8 Experienced Nodal Reactions Of -29.07kn And 19.98kn In X And Y Direction, While For Case Two, Node 8 Experienced Nodal Reactions Of 2.6kn And 9.7kn, Respectively). Stresses For Each Element (Case One, Element One Experienced 1.81kn/M, 21.28kn/M And -3.36kn/M And For Case Two, Element One Experienced -6.32kn/M, -5.64kn/M And -12.08kn/M). Developed Finite Element Analysis Module For Stress Problem Using Isoparametric Element Established An Improvement Over Previous Result Obtained From The Same Case Studied; Hence The Module Can Be Used For Other Related Problems.

Keywords: Isoparametric Element, Isotropic, Homogenous, Two-Dimensional, Curve-Boundaries.

Date of Submission: 10-04-2018 Date of acceptance: 24-04-2018

I. INTRODUCTION

Finite Element Method In The Last Three To Four Decades, Has Received Much Attention Due To The Increasing Use Of High-Speed Computers And The Growing Emphasis On Numerical Methods For Engineering Analysis. This Is Completely Understandable, Since It Is Not Possible To Obtain Analytical Solutions For Many Practical Engineering Problems. An Analytical Solution Is A Mathematical Or Functional Expression That Can Give The Values Of The Desired Unknown Variables At Any Location In A Continuum, And As A Consequence It Is Valid For An Infinite Number Of Locations In The Body. However, Analytical Solutions Can Be Obtained Only For Certain Simple Problems. This Difficulty, However, Can Be Overcome With The Application Of The Finite Element Method. The Finite Element Method Is An Efficient Tool To Numerically Solve The Engineering Problems. In Fact, Finite Element Method Has Been Applied To Complex Geometries And Orthotropic Problems In The Classical Elasticity. For Problems Involving Non Isotropic Material Properties And Complex Boundary Conditions, One Has To Resort To Numerical Methods That Provide Approximate Solutions With Reasonable Accuracies. In Most Of The Numerical Methods, The Solutions Yield Approximate Values Of The Unknown Variables Only At A Discrete Number Of Points In The Continuum. The Process Of Selecting Finite Number Of Discrete Points In The Continuum Can Be Termed "Discretization". One Way Of Discretizing An Entire Body Or Structure Is To Divide It Into A Set Of Small Bodies, Or Units. The Assemblage Of Such Units Then Represents The Original Body. Instead Of Solving The Problem For The Entire Body In One Operation, The Solutions Could Be Formulated For Each Constituent Unit And Then Combined To Obtain The Solution For The Original Body Or Structure. The Finite Element Method Is Applicable To A Wide Range Of Boundary Value Problems In Engineering. In Boundary Value Problems, Solutions Are Sought In The Region Of The Body, While On The Boundaries The Values Of The Unknown Variables (Or Their Derivatives) Are Prescribed.

All Stress Problems Are, In Theory, Capable Of Being Solved Using The Finite Element Method. (E.G. Pressure Vessels, Cooling Towers, Rocket Nozzles). Limitations To The Finite Element Method Occur When Numerous Elements Are Required To Achieve A Desired Degree Of Accuracy Thus Resulting In Large Computer Core Requirements And/Or Excessive Cost. Prior To 1968, Finite Elements Having Only Linear Variation Of Boundaries Were Available. Thus, When A Curved Geometric Boundary Was To Be Modelled, One Was Forced To Introduce Large Numbers Of Elements To Achieve Acceptable Results. This Required The Solution Of A Greatly Increased Number Of Equilibrium Equations And Was Recognized As A Limiting Factor In The Application Of The Finite Element Method To This Type Of Problem. The Introduction Of The Isoparametric Concept By Ergatoudis Enabled Development Of Elements With Polynomic Variation Of Boundaries And Led To A Reduction In The Number Of Elements Necessary To Idealize Curved Boundaries.

Mathematical Formulation Of A Problem For Continuous Body Is Usually Made Up Of Differential Equations Where Mechanical And Physical Quantities Of The Continuous Body Such As Displacement, Stress, Strain Etc., Are Assumed To Be Continuous Functions Of The Space Coordinate. These Mathematically Governing Differential Equations Must Be Solved To Find Values Of The Desired Quantities At Various Points Within The Continuum. Majority Of Design Problems Fall Outside The Reach Of Closed Solution Due To Complex And Irregular Geometric Forms Of The Continuum, Complexity Of The Loading Pattern, Non-Linearity And Inhomogeneity In Properties Of The Material. Thus, A Developer Must Certainly Resort To An Approximate Numerical Analysis That Provide Approximate Solutions With Reasonable Accuracies. In Most Of The Numerical Methods, The Solutions Yield Approximate Values Of The Unknown Variables Only At A Discrete Number Of Points In The Continuum. More So, Finite Element With Curved Edges And Boundaries Are Best Solved Using Isoparametric Element. Hence, There Are Great Ambiguity In Analyzing Isoparametric Element Problem Due To The Complexity Of The Governing Equation And Methods Used In Analyzing The Problem.

II. LITERATURE REVIEW

The Original Concept Of Finite Element Method For Continual Solid Developed In The Mid- 1950s And Had Been Attributed To Turnel Et Al (1956) Who Applied The Matrix Displacement Technique To Plane Stress Problems Using Triangular And Rectangular Elements. They Derived The Stiffness Matrices Without Basing The Formulation On The Field Equation Of The Continuum. Courant (1943), One Of The Pioneering Mathematician In The Development Of Fem Presented An Approximate Solution Of The Saint-Venant Torsion Of An Irregular Cross Section Based On The Principle Of Minimum Potential Energy Using An Assembly Of Triangular Elements.

Since The Publication Of These Results In Literature, Lots Of Work Have Gone Into Perfecting The Method So As To Attain Both Simplicity And Accuracy In Obtaining Solution To Various Problems In Engineering. The Fem Is Now Widely Accepted As A Method Of Stress Analysis. Progress In The Method Has Been On Three Fronts. All Of Which Contributes To The Strength And Flexibility Of The Method. First Of All, The Relationship Of Fem To Previous Well Established Methods In Continuum Mechanics Has Given It A Firm Foundation. Secondly, The Search For, And Development Of The Many Consistent Elements Has Given It A Wide Area Of Application. And Finally, Extension Of The Methods To The Study Of Nonlinear Behavior In Both Materials And Geometric Non Linearity's Has Resulted In More Realistic Models And Design Methods.

Li Et Al, (2001) Presented A Quadratic Finite Element And Quadratic Finite Strip With Generalized Degrees Of Freedom Based On The Fact That The Local Displacement Fields Of The Elements Should Be Compatible With The Global Displacement Field For The Corresponding System. Though Quadratic Elements And Strips Were Used, They Found Results With Good Accuracy And Desirable Convergence. Compared To The Traditional Finite Elements And Strips This Method Yielded Similar Results With Less Degrees Of Freedom. They Also Found That, When Compared To The Linear Element, This Method Can Yield Results With Better Accuracy. Kikuchi Et Al, (1999) Presented A Modification Of An 8-Node Quadrilateral Element Which Is Widely Used In Finite Element Analysis. They Proposed This Element Which Can Represent Any Cartesian And Isoparametric Quadratic Polynomials When It Is Of Bilinear Isotropic Shape. They Found That The Results Were In Good Agreement With The Basic Formulation Of An 8-Node Element. Moreover, This Element Gave Good Results For Higher Order Elements And For Three-Dimensional (3-D)

Elements. Long Et Al, (2004) Investigated The Effect Of Modified Reduced Quadrature Rules On The Presence Of Spurious Modes In The Stiffness Matrices Of The Q8 Serendipity And Q9 Lagrange Membrane Finite Element. The Alternative Five- And Eight-Point Schemes Were Proposed For Q8 And Q9 Elements, Respectively, That Allowed For The Elimination Of Spurious Modes While Element Accuracy Was Maintained. They Found That The Q8 Element Yielded More Economical Results Using The Five-Point Rule When Compared To The Eight Point Rule. The Q9 Element, However, Produced Inadequate Results Using The Five-Point

Huang (1986) Developed Finite Element Analysis Program For Isotropic And Orthotropic Axisymmetric Micropolar (Cosserat) Elastic Solids. Isoparametric Elements Of 8- And 20-Node Are Used To Solve General Three-Dimensional Problems, And Both 4-And 8-Node Elements Are Used For Two-Dimensional Cases. Three-Dimensional Finite Element Formulation For Cylindrical Coordinate System Is Derived. Corresponding Fortran Programs Are Then Developed. Patch Tests Are Performed For Two-Dimensional Cases To Verify The Applicability Of The Finite Element Method To Non-Rectangular Geometries. Several Two-Dimensional And Three Dimensional Problems For Micropolar Elastic Solids Are Solved To Verify The Formulations And Computer Program. Good Agreements Were Obtained In All Cases, Confirming The Validity Of The Finite Element Method.

Gautam (2006) On Stiffness Matrices Of Isoparametric Four-Node Finite Elements By Exact Analytical Integration Presented An Explicit Algebraic Expressions Needed To Compute Element Stiffness Matrices Using Procedural (Fortran) And Object Oriented (C++) Computer 10 Programs. Numerical Illustrations For A Convex Quadrilateral And A Triangle With A Side Node Are Included. The Wide Controversy Due To Conventional Element Level Approximate Numerical Quadrature Within The Computational Square Domain, In Η And Ξ Coordinates, Is Completely Resolved Here By The Closed Form Analytical Integration Within The Physical Element, In X And Y Coordinates. Janucik (1974) On Development And Applications Of A Quadratic Isoparametric Finite Element For Axisymmetric Stress And Deflection Analysis Presented The Theory And Computer Program For An Axisymmetric Finite Element For Static Stress And Deflection Analysis. The Element Is An Eight Node Isoparametric Quadrilateral Based On The Displacement Method Which Is Capable Of Representing Quadratic Variation Of Element Boundaries And Displacements. Element Stiffness Properties Are Developed For Linear Elastic Small Displacement Theory Using Homogeneous Isotropic Material. Test Cases Are Compared With Theoretical Solutions From The Theory Of Elasticity To Identify Program Capabilities And Limitations.

Akpan, (1990) On Finite Analysis Of Two Dimensional Stress Problems Using Isoparametric Developed Used Principle Of Virtual Work With Two Dimensional Isoparametric Element To Develop Finite Element Matrix Equation For Plane Stress Problems. The Overall Equations Are Integrated Numerically Using Gaussian Quadrature Integration Rule. The Resultant Set Of Equation Are Solved On Digital Computer Using Gaussian Elimination Method

Barlets Et Al. (2004) Implemented A Short Matlab Program To Incorporate A Flexible Isoparametric Finite Element Method. Two-Dimensional Domains With Curved Boundaries Of Elastic Problems Having Quadratic Order Were Considered. They Incorporated Triangular And Quadrilateral Elements Equipped With Varying Quadrature Rules Which Allowed For Mesh Refinement. They Provided Numerical Examples For The Laplace Equation With Mixed Boundary Conditions To Indicate The Flexibility Of The Isoparametric Finite Elements.

III. METHODOLOGY

The Primary Objective Of This Study Is To Develop A Finite Element Analysis Program Utilizing Isoparametric Elements. Different Element Types, However, Influence The Response Of A Structure In Varying Fashions. This Effect Depends On Various Factors Such As The Number Of Nodes Per Elements, The Degrees Of Freedom Associated With That Element, The Displacement Field And The Material Properties. As Such, One Element Is Not Always Superior To Another With Respect To Any Given Analysis. Often, It Is The Experience Of The Finite Element Analyst That Determines The Appropriate Element To Be Used Based On Past Solved Problems. The Purpose Of This Study Is Not To Determine The Appropriate Element To Be Used In An Analysis, But To Develop The Capabilities For The Analyst To Create A Module For Analysis. To Develop Model For The Finite Element Analysis For Stress Problem Using Isoparametric Element, An Understanding Of The Theoretical Development Of Finite Elements Is Necessary. Finite Elements Are Discrete Pieces Of The System That Are Interconnected At Nodal Points. In A Structural Sense, Each Element Contributes To The Stiffness Of The System. The Stiffness, In Conjunction With The Boundary (Or Support) Conditions As Well As Prescribed Loadings, Determines The

Deformations Of The System. Numerical Procedures Are Utilized To Determine The Stiffness For Each Element. Furthermore, Algorithms For Combining Each Element Into An Assembly Of Finite Elements Is Needed To Determine The Structural Stiffness. Finally, Solution Techniques Are Needed To Solve The Structural Equations From A Numerical Perspective.

Assumptions:

The Material Are Homogenous And Isotropic

The Material Is Two Dimensional

All Body And Surface Forces Acting On The Body Act In The X-Y Plane, That Is, They Have No Z Component.

A 8node Isoparametric Plane Stress Is Used For Analysis

General Deflection Equation

From The Euler-Bernoulli Theory Of Bending, At A Point Along A Beam, We Know

$$\frac{1}{R} = \frac{M}{EI} \quad 1$$

Where:

R Is The Radius Of Curvature Of The Point And 1/R Is The Curvature

M Is The Bending Moment At The Point

E Is The Elastic Modulus

I Is The Second Moment Of Area At The Point.

Mathematically, It Can Be Shown That For Large R,

$$\frac{1}{R} = \frac{d^2w}{dx^2} \quad 2$$

Where W Is The Deflection At The Point And X Is The Distance Of The Point Along The Beam. Hence, The Fundamental Equation In Finding Deflection Is

$$\frac{d^2w}{dx^2} = \frac{M_x}{EI_x} \quad 3$$

In Which The Subscripts Show That Both M And Ei Are Function Of X And So May Change Along The Length Of The Beam

Basic Relation

The Displacement (U_x, U_y) Of A Structural Element Particle $P(x, y)$ Is Given By

$$U_x = \frac{dw}{dx}, \quad U_y = \frac{dw}{dy}, \quad U_x = \frac{dw}{dx} \quad 4$$

Relating Strain And Deflection, We Have

$$e_{xx} = \frac{dU_x}{dx}, \quad e_{yy} = \frac{dUy_y}{dy}, \quad 2e_{xy} = \frac{dU_x}{dy} + \frac{dU_y}{dx} \quad 5$$

Then,

$$K_{xx} = \frac{d^2w}{dx^2}, \quad K_{yy} = \frac{d^2w}{dy^2}, \quad K_{xy} = \frac{d^2w}{dxdy} \quad 6$$

Are The Curvatures Of The Deflected Surface.

Calculation Of Moments And Shear Forces

Consider A Structural Element Of $dx\ X\ dy$ And With Thickness T. The Element Is Subjected To And External Uniformly Distributed Load P.

Normal Stress Varies Linearly Along Thickness Of Element From The Equation $\sigma = D(\nabla w)$. Hence, The Moment On The Cross Section Can Be Calculated By Integration.

$$M = \begin{Bmatrix} M_x \\ M_y \\ M_{xy} \end{Bmatrix} = \int_{-t/2}^{t/2} D(\nabla w)\, dt \quad 7$$

On Expansion, We Have

$$M_x = \frac{Et^3}{12(1-v^2)}\left(\frac{d^2w}{dx^2} + v\frac{d^2w}{dy^2}\right) = D_p\left(\frac{d^2w}{dx^2} + v\frac{d^2w}{dy^2}\right)$$

$$M_y = \frac{Et^3}{12(1-v^2)}\left(\frac{d^2w}{dy^2} + v\frac{d^2w}{dx^2}\right) = D_p\left(\frac{d^2w}{dy^2} + v\frac{d^2w}{dx^2}\right)$$

$$M_{xy} = M_{yx} = \frac{Et^3}{12(1+v)}\left(\frac{d^2w}{dxdy}\right) = \frac{D_p(1-v)}{2}\left(\frac{d^2w}{dxdy}\right) \quad 8$$

Where, D_p Is Known As Flexural Rigidity Of An Element And It's Given By

$$D_p = \frac{Et^3}{12(1-v^2)}$$

9

Considering Equilibrium Of The Element, The Equations For Forces Can Be Obtained As

$$\frac{dQ_x}{dx} + \frac{dQ_y}{dy} + p = 0$$

$$\frac{dM_x}{dx} + \frac{dM_{xy}}{dy} = Q_x$$

$$\frac{dM_{xy}}{dx} + \frac{dM_y}{dy} = Q_y$$

10

Substituting The Equation For The Moments, We Have

$$Q_x = -D_p \frac{d}{dx}\left(\frac{d^2w}{dx^2} + v\frac{d^2w}{dy^2}\right)$$

$$Q_y = -D_p \frac{d}{dy}\left(\frac{d^2w}{dx^2} + v\frac{d^2w}{dy^2}\right)$$

11

Finite Element Analysis Using Isoparametric Element

For Two Dimensional Quadratic Isoparametric Element With N Nodes, The Geometric Is Expressed As

$$x(\xi,\eta) = \sum_{i=1}^{n} N_i(\xi,\eta)\, x_i \quad ;$$

$$y(\xi,\eta) = \sum_{i=1}^{n} N_i(\xi,\eta)\, y_i$$

12

Where, $N_i(\xi,\eta)$ Are The Standard Displacement Shape Functions. Equation 3.14 Relates The Cartesian And The Natural Coordinates At Each Point. Such A Relationship Must Be Unique And This Is Satisfied If The Jacobian Of The Transformation Of The Partial Derivatives Of A Function In The Natural And Cartesian Coordinate Systems Has A Constant Positive Sign Over The Element. It Can Be Shown That This Condition Is Satisfied For Linear Quadrilateral Elements If No Internal Angle Between Two Element Sides Is Equal Or Greater Than 180^0. For Quadratic Elements It Is Additionally Required That The Side Nodes Are Located Within The "Middle Third" Of The Distance Between Adjacent Corners. There Are No Practical Rules For Higher Order Quadrilateral Elements And The Constant Sign Of The Determinant Of The Jacobian Matrix Is The Only Possible Verification In This Case. Figure 1 Shows Some Examples Of 2d Isoparametric Elements.

Equation 3.27 Allows Us To Obtain A Relationship Between The Derivatives Of The Shape Functions With Respect To The Cartesian And The Natural Coordinates. In General, N_i Is Expressed In Terms Of The Natural Coordinates ξ *and* η And The Chain Rule Of Derivation Yields

$$\frac{\partial N_i}{\partial \xi} = \frac{\partial N_i}{\partial x}\frac{\partial x}{\partial \xi} + \frac{\partial N_i}{\partial y}\frac{\partial y}{\partial \xi} \quad ;$$

$$\frac{\partial N_i}{\partial \eta} = \frac{\partial N_i}{\partial x}\frac{\partial x}{\partial \eta} + \frac{\partial N_i}{\partial y}\frac{\partial y}{\partial \eta}$$

13

In Matrix Form,

$$\begin{Bmatrix} \frac{\partial N_i}{\partial \xi} \\ \frac{\partial N_i}{\partial \eta} \end{Bmatrix} = \begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} \end{bmatrix} \begin{Bmatrix} \frac{\partial N_i}{\partial x} \\ \frac{\partial N_i}{\partial y} \end{Bmatrix} = J^{(e)} \begin{Bmatrix} \frac{\partial N_i}{\partial x} \\ \frac{\partial N_i}{\partial y} \end{Bmatrix} \qquad 14$$

Where $J^{(e)}$ Is The *Jacobian Matrix* Of The Transformation Of The Derivatives Of N_i In The Natural And Global Axes. The Super Index *E* In J Denotes That This Matrix Is Always Computed At Element Level. We Deduce From Eq. 3.29

$$\begin{Bmatrix} \frac{\partial N_i}{\partial x} \\ \frac{\partial N_i}{\partial y} \end{Bmatrix} = \left[J^{(e)}\right]^{-1} \begin{Bmatrix} \frac{\partial N_i}{\partial \xi} \\ \frac{\partial N_i}{\partial \eta} \end{Bmatrix} =$$

$$\frac{1}{|J^{(e)}|} \begin{bmatrix} \frac{\partial y}{\partial \eta} & -\frac{\partial y}{\partial \xi} \\ -\frac{\partial x}{\partial \eta} & \frac{\partial x}{\partial \xi} \end{bmatrix} \begin{Bmatrix} \frac{\partial N_i}{\partial \xi} \\ \frac{\partial N_i}{\partial \eta} \end{Bmatrix}$$

15

Where $|J^{(e)}|$ Is The Determinant Of The Jacobian Matrix; (Also Simply Called "The Jacobian"). This Determinant Relates The Differential Of Area In The Two Coordinate Systems, I.E.

$$dx\, dy = |J^{(e)}|\, d\xi\, d\eta$$

16

The Terms Of $J^{(e)}$ Are Computed Using The Isoparametric Approximation In Eq. 3.31, I.E.

$$\frac{\partial x}{\partial \xi} = \sum_{i=1}^{n} \frac{\partial N_i}{\partial \xi} x_i \quad ;$$

$$\frac{\partial x}{\partial \eta} = \sum_{i=1}^{n} \frac{\partial N_i}{\partial \eta} x_i$$

17

$$J^{(e)} = \sum_{i=1}^{n} \begin{bmatrix} \frac{\partial N_i}{\partial \xi} x_i & \frac{\partial N_i}{\partial \xi} y_i \\ \frac{\partial N_i}{\partial \eta} x_i & \frac{\partial N_i}{\partial \eta} y_i \end{bmatrix}$$

18

For A Rectangular Element,

$$J^{(e)} = \begin{bmatrix} a & 0 \\ 0 & b \end{bmatrix} \; and \; |J^{(e)}| = ab$$

19

The Strain Matrix Is Obtained As

$$B_{(\xi,\eta)} = \begin{bmatrix} \frac{\partial N_i}{\partial x} & 0 \\ 0 & \frac{\partial N_i}{\partial y} \\ \frac{\partial N_i}{\partial y} & \frac{\partial N_i}{\partial x} \end{bmatrix} = \frac{1}{|J^{(e)}|} \begin{bmatrix} \bar{b}_i & 0 \\ 0 & \bar{c}_i \\ \bar{c}_i & \bar{b}_i \end{bmatrix}$$

20

$$\bar{b}_i = \frac{\partial y}{\partial \eta}\frac{\partial N_i}{\partial \xi} - \frac{\partial y}{\partial \xi}\frac{\partial N_i}{\partial \eta} \quad ;$$

$$\bar{c}_i = \frac{\partial x}{\partial \xi}\frac{\partial N_i}{\partial \eta} - \frac{\partial x}{\partial \eta}\frac{\partial N_i}{\partial \xi}$$

21

The Stiffness Matrix Is Obtained As

$$K_{ij}^{(e)} =$$

$$\iint B^T D B_j\, t\, dxdy =$$

$$\int_{-1}^{+1}\int_{-1}^{+1} B_i^T(\xi,\eta) D B_j(\xi,\eta)\, |J^{(e)}|\, t\, d \quad 22$$

Equation 3.38 Shows That The Integrand Of $K_{ij}^{(e)}$ Contains Rational Algebraic Functions In ξ And η.

An Exception To This Rule Is When The Determinant Of The Jacobian Matrix Is Constant. This Only Occurs For Rectangular Elements And For Straight Side Triangles.

For General Quadrilateral Shapes The Analytical Integration Of $K_{ij}^{(e)}$ In The Natural Coordinate System ξ,η Is Difficult (And In Some Cases Impossible!) And The Best Option Is To Use Numerical Integration.

To Accomplish The Integration As Given In Equation 22, From A Numerical Perspective, Gaussian Quadrature Is Used. The Gauss Product Rule Is Obtained By Successive Application Of A One-Dimensional Gauss Rule. For The Function $\emptyset = \emptyset(\xi,\eta)$ The Quadrature Rule Is Given As

$$\int_{-1}^{+1}\int_{-1}^{+1} \emptyset(\xi,\eta)\; d\xi\; d\eta =$$

$$\sum_{p=1}^{n_p}\sum_{q=1}^{n_q} \emptyset(\xi_p,\eta_q) W_p W_q$$

23

Where n_p And n_q Are The Number Of Integration Points Along Each Natural Coordinate ξ And η Respectively; ξ_p *and* η_q Are The Natural Coordinates Of The pth Integration Point And W_p *And* W_q Are The Corresponding Weights. A Similar Procedure Is Followed To Compute The Equivalent Nodal Force Vectors For Isoparametric Quadrilateral Elements.

$$f_{bi}^{(e)} = \int\int N_i^T \boldsymbol{b}\, t\, dxdy =$$

$$\int_{-1}^{+1}\int_{-1}^{+1} N_i^T \boldsymbol{b} \left|J^{(e)}\right| t\, d\xi\, d\eta \quad 24$$

The Numerical Integration Of The Equivalent Nodal Force Vector Due To Body Forces For Isoparametric Quadrilateral Elements Equation 3.39 Gives

$$f_{bi}^{(e)} = \sum_{p=1}^{n_p}\sum_{q=1}^{n_q} \left(N_i^T \boldsymbol{b} \left|J^{(e)}\right| t\right)_{p,q} W_p W_q \quad 25$$

Steps To Be Followed For The Finite Element Analysis Of A Structural Stress Problem;

Step 1. Discretize The Structure Into A Mesh Of Finite Elements.

Step 2. Compute For Each Element The Stiffness Matrix And The Equivalent Nodal Force Vector Due To External Loads And "K" And "F".

Step 3. Assemble The Stiffness Matrix And The Equivalent Nodal Force Vector For Each Element Into The Global System

$$\boldsymbol{Ka} = \boldsymbol{f} \quad 26$$

$$\boldsymbol{K} = \boldsymbol{A}\boldsymbol{K}^{(e)} \quad ; \quad \boldsymbol{f} = \boldsymbol{A}\boldsymbol{f}^{(e)} + \boldsymbol{p} + \boldsymbol{r}$$

Where A Denotes The Operator For The Global Assembly Of All The Individual Matrices And Vectors For Each Element In The Mesh In Equation 3.43.

"P" Is The Vector Of External Point Forces Acting At The Nodes And "R" Is The Vector Of Nodal Reaction To Be Computed "A Posteriori" Once The Nodal Displacement Are Found.

The Assembly Of The Reaction Vector "R" Into "F" Is Optional, As The Reactions Do Not Influence The Solution For The Nodal Displacements

Step 4. The Nodal Displacements Are Computed By Solving The Equation System Eq.3.43 Where The Prescribed Displacements Must Be Adequately Imposed, I.E.

$$\boldsymbol{a} = \boldsymbol{K}^{-1}\boldsymbol{f} \quad 27$$

The Nodal Reactions Are Obtained At The Prescribed Nodes.

Step 5. The Strains And Stresses Are Computed Within Each Element From The Nodal Displacements As

$$\boldsymbol{\varepsilon} = \boldsymbol{B}\,\boldsymbol{a} \quad ; \quad \boldsymbol{\sigma} = \boldsymbol{D}\,\boldsymbol{B}\,\boldsymbol{a} \quad 28$$

The Nodal Axial Forces For Each Element Can Be Computed From

$$\boldsymbol{q}^{(e)} = \boldsymbol{K}^{(e)}\boldsymbol{a}^{(e)} - \boldsymbol{f}^{(e)} \quad 29$$

Main Programming

Previous Sections Provides All The Necessary Expressions For Programming The Computation Of The Stiffness Matrix And The Equivalent Nodal Force Vector For Each Element. Below Is The Main Program Algorithm.

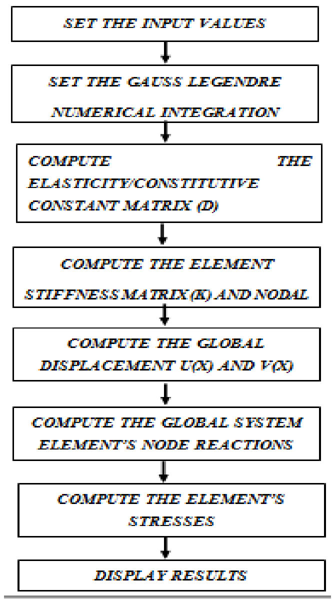


Figure 3: Main Program Flowchart Algorithm

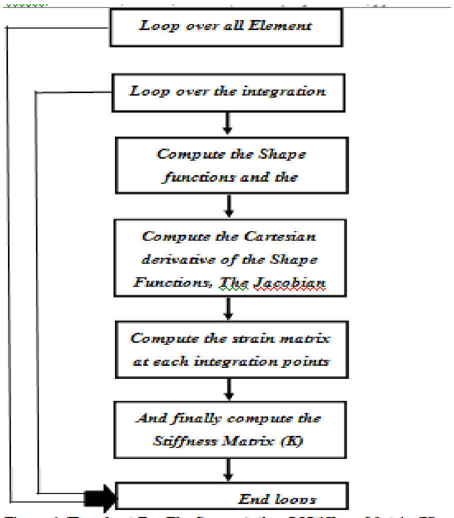


Figure 4: Flowchart For The Computation Of Stiffness Matrix (K)

IV. RESULT AND DISCUSSION

The Computer Program Written And The Model Developed Is Used To Analyze The Problems Below. The Global Displacements, Nodes Reactions And The Stresses For Each Element Are Computed Using The Module. The Program Codes Can Be Found At The Appendices Section.

Case Study To Explain The Work In View

Case 1

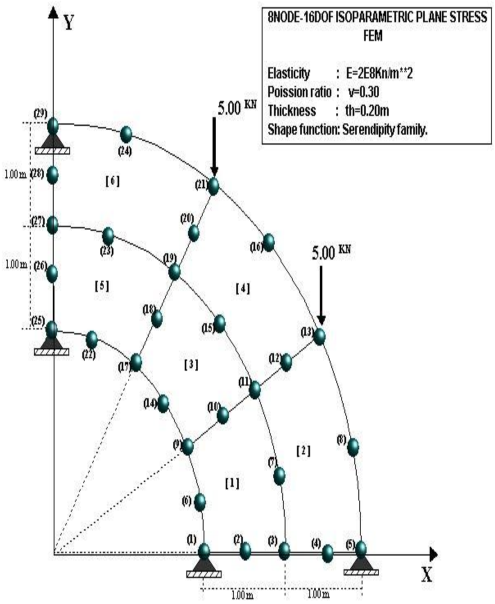


Figure 4.1: An 8node Isoparametric Plane Stress Problem 1

, The Input Value For The "Case 1" Above Are:

- ✓ **Elasticity Constant (E)** = $2\hat{}8KN/m^2$
- ✓ **Poisson Ratio(V)** = 0.3
- ✓ **Plane Element Thickness (Th)** = 0.2m
- ✓ **No Of Element** = 6
- ✓ **Element Node Number For Each Element:**
- ✓ Element Node Number

And In Matlab Syntax, It's Written As

[1 3 11 9 2 7 10 6; 3 5 13 11 4 8 12 7; 9 11 19 17 10 E15 18 14 ; 11 13 21 19 12 16 20 15 ;17 19 27 25 18 23 26 22 ; 19 21 29 27 20 24 28 23]

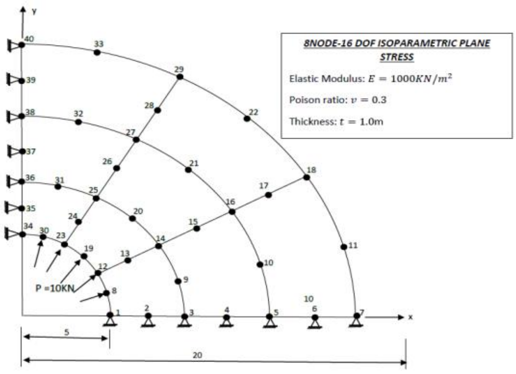
- ✓ **Global System Coordinate`S For Element Node Cartesian Value In The Form X And Y Coordinates**

And In Matlab Syntax,

[1.0 0; ;1.5 0 ; 2.0 0 ; 2.5 0 ;3.0 0 ; 0.9659258 0.258819 ; 1.931852 0.5176381 ; 2.897778 0.7764571 ; 0.8660254 0.5 ; 1.299038 0.75 ; 1.732051 1 ; 2.165064 1.25 ; 2.598076 1.5 ; 0.7071068 0.7071068 ; 1.414214 1.414214 ; 2.12132 2.12132 ; 0.5 0.8660254 ; 0.75 1.299038 ; 1 1.732051 ; 1.25 2.165064 ; 1.5 2.598076 ; 0.258819 0.9659258 ; 0.5176381 1.931852 ; 0.7764571 2.897778 ; 6.123032e-17 1 ; 9.184548e-17 1.5 ; 1.224606e-16 2 ; 1.530758e-16 2.5 ; 1.83691e-16 3]

- ✓ **No Of Fixed Nodes** = 4
- ✓ **Fixed Nodal Number** = [1 5 25 29]
- ✓ **Nodes With External Force** = [13 21]
- ✓ **External Force (Kn) In X And Y Coordinates**= [0 -5; 0 -5]

Case 2;



The Input Value For The "Case 2" Above Are:

- ✓ **Elasticity Constant (E)** = $1000KN/m^2$
- ✓ **Poisson Ratio(V)** = 0.3
- ✓ **Plane Element Thickness (Th)** = $1m$
- ✓ **No Of Element** = 9
- ✓ **Element Node Number For Each Element:**

In Matlab Syntax, It's Written As

[1 2 3 9 14 13 12 8; 3 4 5 1 16

15 14 9; 5 6 7 11
18 17 16 10; 12 13 14
20 25 24 23 19; 14
15 16 21 27 26 25 20;
16 17 18 22 29
28 27 21; 23 24
25 31 36 35 34 30; 25
26 27 32 38 37 36
31; 27 28 29 33 40 39
38 32]

- ✓ **Global System Coordinate`S For Element Node Cartesian Value In The Form X And Y Coordinates**

And In Matlab Syntax,
[5 0;6.667 0;8.333 0;10.667 0;13 0;16.5 0;20 0;4.830 1.294;8.049 2.157;12.557 3.365;19.319 5.176;4.33 2.5;5.774 3.333;7.217 4.167;9.238 5.333;11.258 6.5;14.289 8.250;17.321 10;3.536 3.536;5.893 5.893; 9.192 9.192;14.142 14.142;2.5 4.330;3.333 5.774;4.167 7.217;5.333 9.238;6.5 11.258;8.25 14.289;10.000 17.321;1.294 4.830;2.157 8.048;3.365 12.557;5.176 19.319;0 5;0 6.667;0 8.333;0 10.667;0 13;0 16.5;0 20]

- ✓ **No Of Fixed Nodes** = 14
- ✓ **Fixed Nodal Number** = [1 2 3 4 5 6 7 34 35 36 37 38 39 40]
- ✓ **Nodes With External Force** = [8 12 19 23 30]
- ✓ **External Force (Kn) In X And Y Coordinates**= [9.659 2.588; 8.66 5; 7.071 7.071; 5 8.66;2.588 9.658]
- ✓

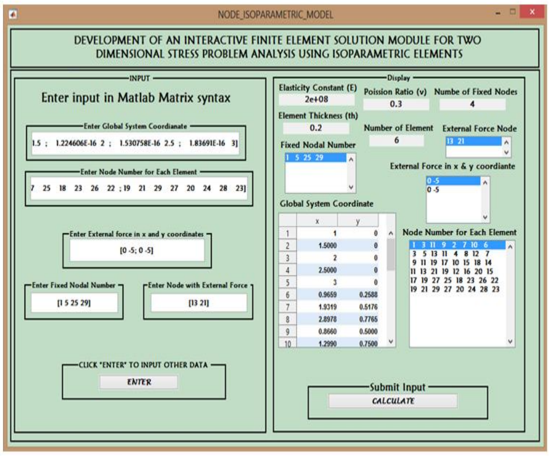
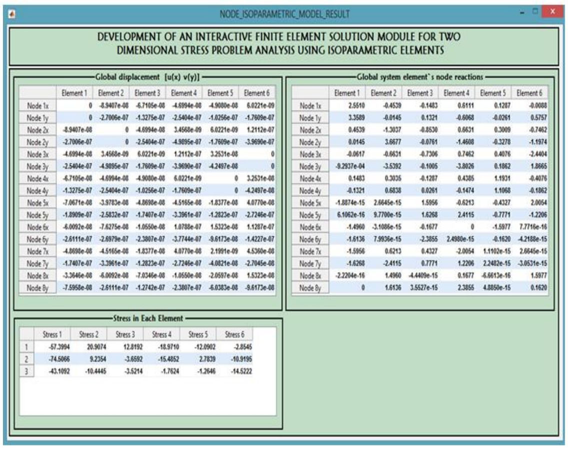


Plate 1: Gui Input Environment Showing The Input Value For "**Case 1**" After Insertion

The Graphic User Interface (Gui) Showing The Input Values And The Output Result

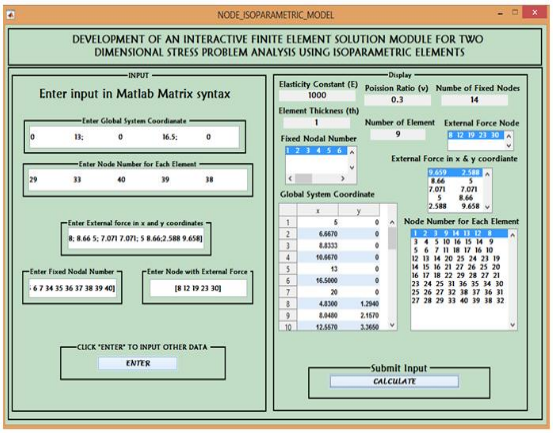
A Windows Application Which Is Developed From Matlab Gui Is Created To Enable A User Friendly Environment For The Input And Output Of The Data. Below Are The Input And Output Environment Generated From The Two Case Studies Above.

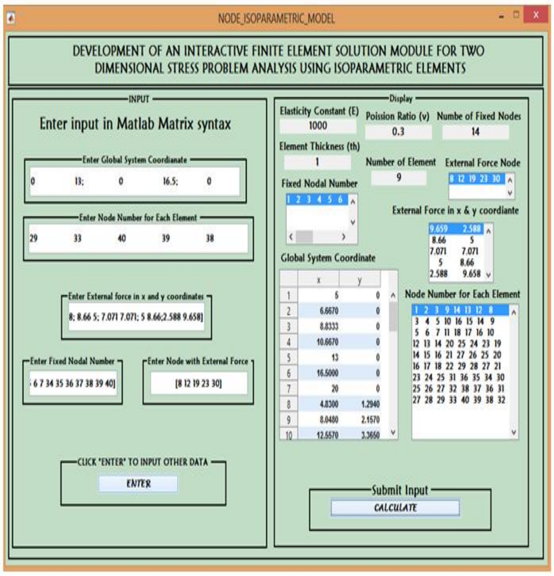
Plate 2: Gui Output Environment Showing The Results For "**Case 1**"

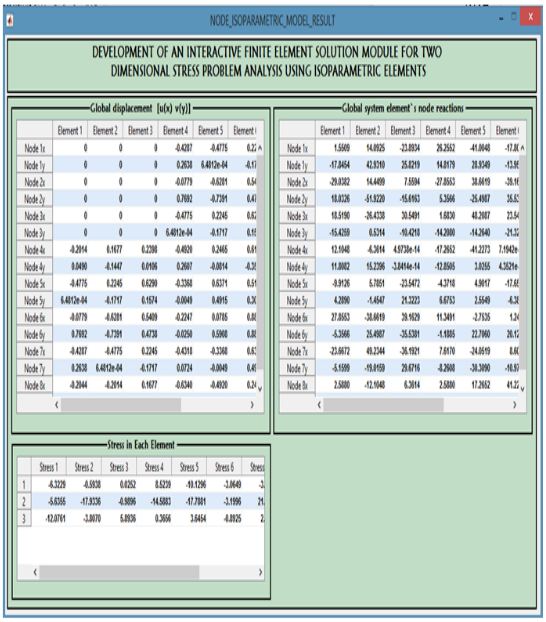


Case Two Input And Output Data

Plate 3: Gui Input Environment Showing The Input Value For "**Case 2**" After Insertion







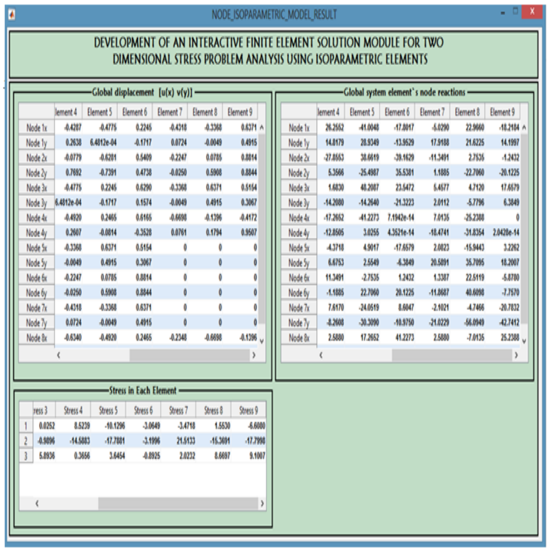


Plate 4: Gui Output Environment Showing The Results For "**Case 2***"

The Result Generated From The Figures Above Are As Follow:
The Global System Displacement (U(X) &V(Y)) For Each Element For The Two Case Studies
The Global Node Reactions In X & Y Direction For Each Element
The Stress For Each Of The Element
It Can Be Deuced From The Result That;

The Fixed Points, That's, The Fixed Nodes Which Are Not Considered In The Problem Defined In A Previous Work Which Tentative Affects The Correctness And Accuracy Of The Results Obtained For The Displacement And Reactions For Each Element. In This Present Work, The Fixed Point Are Rightly Considered Which Enhanced The Accuracy Of The Work Output. The Model And Program Used To Develop And Analyzed The Previous Similar Work Is Ibm Which Has Several Limitation In Computing Some Certain Mathematical Expression And Also Analyzing Some Of The Complex Governing Equations. This Also Contributed To The Accuracy Of The Output Generated. In The Present Work, A More Enhanced And Latest Software Is Used. This Helps In The Correctness Of The Limitation Mentioned Above And Also Systematically Makes The Output More Accurate The Model Developed In Previous Similar Work Was Just Used To Solve A Sample Of Isoparametric Problems. But The Present Work Developed Grants The Enablement To Solve Several Two Dimension Isoparametric Problem Cases. An Addition Of A Graphic User Interface Makes It Possible For A User With No Knowledge Of A Programming Software To Analyze And Solve A Stress Problem Using Isoparametric Element. The Interface Is User Friendly And Very Easy To Use. And Lastly, A Window App Was Developed. This Makes It Possible For An Analyst Or A User To Have Access To The Module Without Installing The Matlab Program Software.

V. CONCLUSIONS

The Results Obtained In This Study Are Of High Level Of Accuracy Compared To Previous Work. It Can Also Be Deduced That The Module Developed Can Be Used To Analyze Different Cases Of Two Dimensional Stress Problem With Curved Boundaries. This Poses A Great Breakthrough In The Analysis Of Finite Element Using Isoparametric Element. In Summary, The Use Of A Computer Program Is Essential In Analysis Of Finite Element Using Isoparametric Element. Having Successfully Developed And Simulated, A New Model With Less Simplified Assumption Is Recommended. In Real Life Situation, It Could Be Difficult To Encounter A Complete Structural Element With Materials Across It Been Homogenous And Isotropic. Hence, A Model With Different Material Properties In A Structural Element Is Recommended. A Three Dimensional Case With Higher Number Of Node Considered, Is Recommended For New Model. A New Model That Allows The Analysis Of Four, Eight, Nine, Twelve, Etc., Simultaneously In One Model Is Recommended.

REFERENCES

[1] Akpan, U. O., 1990. Finite Analysis Of Two Dimensional Stress Problems Using Isoparametric. Msc. Thesis. University Of Ibadan, Nigeria.
[2] Alberty, J., Carsetensen, C., Funken, S.A. And Klose, R., 2002. Matlab

Implementation Of The Finite Element Method In Elasticity. Journal Ofcomputing, Vol. 69, No. 3, Pp. 239 - 263.

[3] Ariman, T., 1968. Circular Micropolar Plates. Ingenieurarchir, Vol. 37, Pp. 156-160.

[4] Bartels, S., Carstensen, C. And Hecht, A., 2004. P2q2iso2d = 2d Isoparametric Fem In Matlab. Journal Of Computation And Applied Mathematics, Vol. 192, No. 2, Pp. 219 - 250.

[5] Caprani, C., 2008. General Deflection Equation. Deflection Of Flexural Member, Structural Engineering, Pp. 4.

[6] Chandenduang, C., Becker, A.A., 2001. Boundary Element Formulation For Two-Dimensional Creep Problems Using Isoparametric Quadratic Elements. Computers And Structures, Vol. 81, Pp. 1611–1618.

[7] Christoph L., 2016. High Order Unfitted Finite Element Methods On Level Set Domains Usingisoparametric Mappings. Computer. Methods In Applied Mechanics And Engineering, Vol. 300, Pp. 716–733

[8] Clough, R. And Rashid, Y., 1965, Finite Element Analysis Of Axisymmetric Solids. Journal Of Engineering Mechanics, Amer. Soc. Civil Engrs., Pp. 71-85.

[9] Courant, R., 1943. Variation Methods For Solution Of Problems Of Equilibrium And Vibration. Bulletin Of The American Mathematical Society, Vol. 49, Pp. 1-23.

[10] Dario, N., Bradley, W. 1973, A Comparison Of First And Second Order Axially Symmetric Finite Elements. Inter. Journal For Num. Meth. In Eng., Vol. 5, Pp. 573-583.

[11] Desai, C, Abel, J., 1972. Introduction To The Finite Element Method. Nostrand Reinhold Company, New York, First Edition.

[12] Ergatoudis, J., 1966. Quadrilateral Elements In Plane Analysis And Introduction To Solid Analysis. Msc Thesis, University College Of Swansea, Wales.

[13] Ergatoudis, J., 1968. Isoparametric Finite Elements In Two And Three Dimensional Stress Analysis. Phd. Thesis, University College Of Swansea, Wales.

[14] Fraeijs De Veubeke, B., 1968. A Conforming Finite Element For Plate. Int. Ji. Of Solid And Structure, Vol. 4, Pp. 95-108.

[15] Fuang-Yuan H., 1986. Two And Three-Dimensional Isoparametric Finite Elements For Axisymmetric Micropolar Elasticity. Phd Thesis, Faculty Of The Graduate School Of The New Jersey Institute Of Technology.

[16] Gautam D., 2006. Stiffness Matrices Of Isoparametric Four-Node Finite Elements By Exact Analytical Integration. Columbia University, New York, Ny.

[17] Gauthier, R.D., 1974. Analytical And Experimental Investigations In Linear Isotropic Micropolar Elasticity. Doctoral Dissertation, Univ. Of Colorado.

[18] Gauthier R.D. And Jahsman, W.E., 1975. A Quest For Micropolar Elastic Constants, J. Of Applied Mechanics, Pp.369-374.

[19] Gupta, A., Mohraz, B. 1972. A Method Of Computing Numerically Integrated Stiffness Matrices. Inter. Journal For Num. Meth. In Eng., Vol. 5, Pp. 83-89.

[20] Henrik F.H. And Christian G.H., 2007. Efficient Finite Element Formulation For Analysis And Optimization Of Laminated Composite Shell Structures. Department Of Mechanical Engineering, Aalborg University.

[21] Hermann, L.R., 1965. Elastic Equation For Incompressible And Nearly Compressible Materials By A Variation Theorem. Aiaa Ji., Vol. 3, No. 10.

[22] Irons, B., 1966. Numerical Integration Applied To Finite Element Methods. Conference On The Use Of Digital Computers In Structural Engineering, University Of Newcastle, England.

[23] Janucik, F.X., 1974. Development And Applications Of A Quadratic Isoparametric Finite Element For Axisymmetric Stress And Deflection Analysis. Thesis, Rochester Institute Of Technology.

[24] Kaloni, R.N., And Ariman, T., 1967. Stress Concentration Effects In Micropolar Elasticity. Zamp, Vol. 18, Pp. 130-141.

[25] Kikuchi, F., Okabe, M. And Fujio, H., 1999. Modification Of The 8-Node Serendipity Element. Journal Of Finite Element Analysis Design, Vol. 37, No. 04, Pp. 325 - 339.

[26] Kishida, M., Sasaki, K., Hanzawa, H., 1982. One Solution Of Three Dimensional Boundary Value Problems In The Couple-Stress Theory Of Elasticity. A.S.M.E. J. Of Applied Mechanics, Vol.49, Pp. 519-524.

[27] Kokou D. And Marie F., 2005. A High Order Isoparametric Finite Element Method For The Computation Of Waveguide Modes. Computer. Methods In Applied Mechanics And Engineering. Vol. 194, Pp. 837–858.

[28] Li, Q.S., Yang, L.F. And Li, G.Q., 2001. The Quadratic Finite Element/Strip With Generalized Degrees Of Freedom And Their

Application. Journal Of Finiteelement Analysis And Design, Vol. 37, No. 04, Pp. 325 - 339.
[29] Long, C.S. And Groenwold, A.A., 2004. Reduced Modified Quadrature For Quadratic Membrane Finite Elements. International Journal For Numerical Methods In Engineering, Vol. 61, No. 06, Pp. 837 - 855.
[30] Martin, H., Carey, G., 1973. Introduction To Finite Element Analysis. (Mcgraw Hill Book Company, Inc., New York, First Edition.
[31] Melosh, R.J., 1963. Basic For Derivation Of Matrices For Direct Stiffness Method. Aiaa Ji, Vol 1, No. 7, Pp. 1631-1637.
[32] Mindlin, R.D. And Tiersten, H.F., 1962. Effects Of Couple Stress In Linear Elasticity. Arch. Rat. Mech. Anal. Vol.11, 1962, Pp. 415-448.
[33] Moxey, D., Green, M.D., Sherwin, S.J., Peiro J., 2014. An Isoparametric Approach To High-Order Curvilinear Boundary-Layer Meshing. . Computer. Methods In Applied Mechanics And Engineering, Vol. 283. Pp. 636–650
[34] Oñate, E., 2009. Isoparametric 2d Solid Elements. Numerical Integration And Applications. Structural Analysis With The Finite Element Method. Linear Statics. Volume 1. Basis And Solids
[35] Rameez I., 2007. Development Of A Finite Element Program Incorporating Advanced Element Types. Msc Thesis, Faculty Of The Graduate College Of The Oklahoma State University.
[36] Salem, A.Z.I., Canaan, S.A. And Saigal, S. (2001). Mid-Node Admissible Spaces For Quadratic Triangular 2d Finite Elements With One Edge Curved. Journal Of International Journal For Numerical Methods In Engineering, Vol. 50, No. 01, Pp. 181 - 197.
[37] Timoshenko, S., 1958. Strength Of Materials, Part Ii (Van Nostrand Reinhold Company, New York), Third Edition.
[38] Timoshenko, S. And Goodier, J., 1970. Theory Of Elasticity (Mcgraw Hill Book Company, Inc., New York.
[39] Turner, M.J., Clough, R.J., Martin, H.C., And Topp, L.J., 1956. Stiffness And Deflection Analysis Of Complex Structure. J. Of Aero. Sc., Pp. 805-823
[40] Wilson, E., 1965. Structural Analysis Of Axisymmetric Solids. Aiaa Journal, Vol. 3, No. 12, Pp. 2269-2274.
[41] Wilson, E., Parsons, B., 1973.Trapezoidal Finite Elements - Their Derivation And Use For Axisymmetric Rotating Bodies. Asme Presentation At The Design Engineering Technical Conference, Cincinnati, Ohio, September 9-12. Paper 73 - Det - 47.
[42] Zienkiewicz, O., Irons, B., 1968. The Isoparametric Finite Element System-A New Concept In Finite Element Analysis. Conference On Recent Advances In Stress Analysis: New Concepts And Techniques And Their Practical Application, Royal Aeronautical Society, Pp. 35-40.
[43] Zienkiewicz, O., Irons, B. 1970, Isoparametric Elements. Finite Element Techniques In Structural, Mechanics, H. Tottenham And C. Brebbia, Eds., Southhampton England Stress Analysis Publishers, Southhampton University, Pp. 283-301.
[44] Zienkiewicz, O., 1971. The Finite Element Method In Engineering Science. (Mcgraw Hill Book Company, Inc., London), Second Edition.
[45] Zienkiewicz, O., Irons, B., Ergatoudis, J., Ahmad, S., Scott, F. 1972. Isoparametric And Associated Element Families For Two And Three Dimensional Analysis. Finite Element Methods In Stress Analysis, Ivarholand And Kolbein Bell, Eds., Trondheim, Norway: Tapir Publishing; The Technical University Of Norway, Pp. 383-432.

K.M. Odunfa "Development of an Interactive Finite Element Solution Module for 2d Stress Problem Analysis Using Isoparametric ElementInternational Journal of Engineering Research and Applications (IJERA) , vol. 8, no. 4, 2018, pp. 67-78