ECCOMAS Congress 2016
VII European Congress on Computational Methods in Applied Sciences and Engineering
M. Papadrakakis, V. Papadopoulos, G. Stefanou, V. Plevris (eds.)
Crete Island, Greece, 5–10 June 2016

NEW EDUCATIONAL MATLAB TOOL TO EXPLAIN TWO DIMENSIONAL FINITE ELEMENT METHOD TO GRADUATE STUDENTS

Rafet Sisman[1] **and Abdurrahman Sahin**[2]

[1] Yildiz Technical University
Istanbul
e-mail: rafet@yildiz.edu.tr

[2] Yildiz Technical University
Istanbul
abdsahin@yildiz.edu.tr

Keywords: Educational Tool, 2D Finite Element Method, 2D Rectangular Finite Elements, Gauss Quadrature Integration, Lagrangian Shape Functions

Abstract. *In this study a simple code is developed in powerful programming software MATLAB. Graduate engineering students are generally expected to cope with Finite Element Method in two and three dimensions during their Finite Element Courses. However textbooks of that courses, sometimes, do not have enough solved examples for students, because in two and three dimensions calculations are difficult to do manually. At this step a computer program which includes infinite number of examples, as user defines them, and their solution steps, is of crucial importance. The produced MATLAB code has these features as it can display stiffness matrix with or without boundary conditions, deformed shape of the body, displacement, strain and stress distributions over the surface. This code also makes it possible to analyze bodies as it is a plane stress or plane strain body. Besides, this MATLAB code can generate a step by step solution report for students to comprehend the Finite Element procedure. Since this code is a simple and educational tool, only one-piece rectangular bodies can be analyzed. The body to be analyzed can be meshed into various number of rectangular elements. These rectangular elements utilize bilinear Lagrangian shape functions, and Gauss Quadrature technique is used to deal with integrations. In this study four Gauss integration points are used as two in X direction and two in Y direction, which yields the exact integration as approximation functions are bilinear.*

1 INTRODUCTION

Finite Element Method (FEM) courses are essential for most engineering students, especially mechanical engineering and civil engineering students. FEM courses in graduate level generally include two dimensional problems. However, textbooks of those courses are short of solved problems and solutions are not explanatory, for example only necessary terms of stiffness matrix are determined because of its difficulty to do manually. Those deficiencies become motivation for this study. As an outcome of this study, a simple software is developed via MATLAB [1] to create two dimensional FEM examples. There are a number of important advantages of this software. First of all, this software makes it possible for user to create infinite number of examples with their FEM solutions. FEM solutions of those problems are written to a text file as a solution report which includes all necessary matrices. Also those matrices are written to a Microsoft Excel file for convenience of user. Deformed shape of the system, stress and strain distributions are also among the visual outputs.

Two dimensional FEM problems includes plane stress problems and plane strain problems [2]. The developed software can handle both of these two kinds of problems. Planar problems to be solved with the developed software must include one-part rectangular geometries. Since the motivation of this study emerges from educational necessities, that level of simplicity is adopted.

Meshing algorithm of the developed software is also simple, only rectangular meshing is available. Four node rectangular elements are created during meshing process. To determine element stiffness matrix for those four node rectangular elements numerical integration is used. Using Lagrangian shape functions for rectangular elements with four nodes, and utilizing Gauss Quadrature technique for numerical integrations yields the exact solution when using four Gauss points per element [3].

2 TWO DIMENSIONAL FEM FUNDAMENTALS

2.1 Constitutive Matrix

In two dimensional FEM, problems can be either plane stress problems or plane strain problems. The difference between these two kinds of problems can be illustrated as shown in Figure 1. If the planar geometry, which is considered in FEM problem, has a small thickness with respect to other two dimensions then it can be said that this problem is a plane stress problem. If that thickness is large enough, which causes zero strain through that direction, with respect to other two dimensions then it can be said that this problem is a plane strain problem [3].

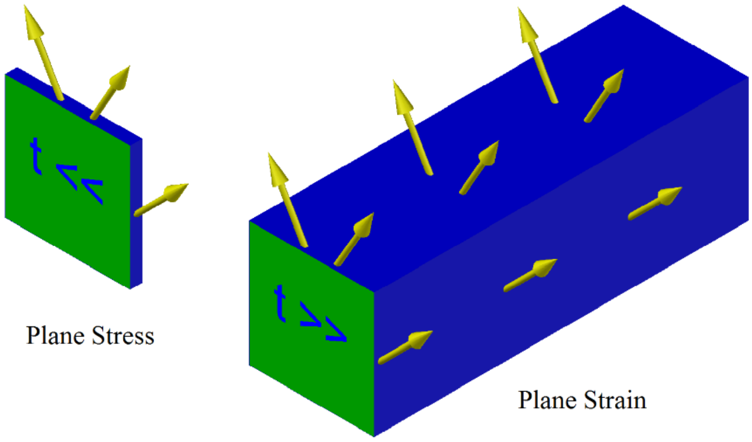

Figure 1 – Plane stress and plane strain geometries

During Finite Element procedures that difference is defined by the constitutive matrix or stress-strain relation matrix, denoted by $[D]$. The constitutive matrix for plane stress problems is given in equation (1), and constitutive matrix for plane strain problem is given in equation (2) where E is Young's modulus and υ is Poisson's ratio [3].

$$[D]=\frac{E}{(1-\upsilon^2)}.\begin{bmatrix} 1 & \upsilon & 0 \\ \upsilon & 1 & 0 \\ 0 & 0 & \frac{1-\upsilon}{2} \end{bmatrix} \tag{1}$$

$$[D]=\frac{E}{(1+\upsilon).(1-2\upsilon)}.\begin{bmatrix} 1-\upsilon & \upsilon & 0 \\ \upsilon & 1-\upsilon & 0 \\ 0 & 0 & \frac{1-2\upsilon}{2} \end{bmatrix} \tag{2}$$

In the developed software, user is expected to define material properties. For two dimensional elastic isotropic materials, Young's modulus and Poisson's ratio are necessary and sufficient parameters to define material. In the software user defines these two parameters, besides user defines problem state as plane stress or plane strain.

2.2 Element Stiffness Matrix

As stated before rectangular elements with four nodes are used for the FEM analysis. Those rectangular elements utilize Lagragian shape functions and uses Gauss Quadrature technique for numerical integrations. Lagragian shape functions are given in Figure 2, and Gauss point coordinates and weights are given in Figure 3 [3]. Numerical integration with those shape functions and four Gauss points yields exact integration result.

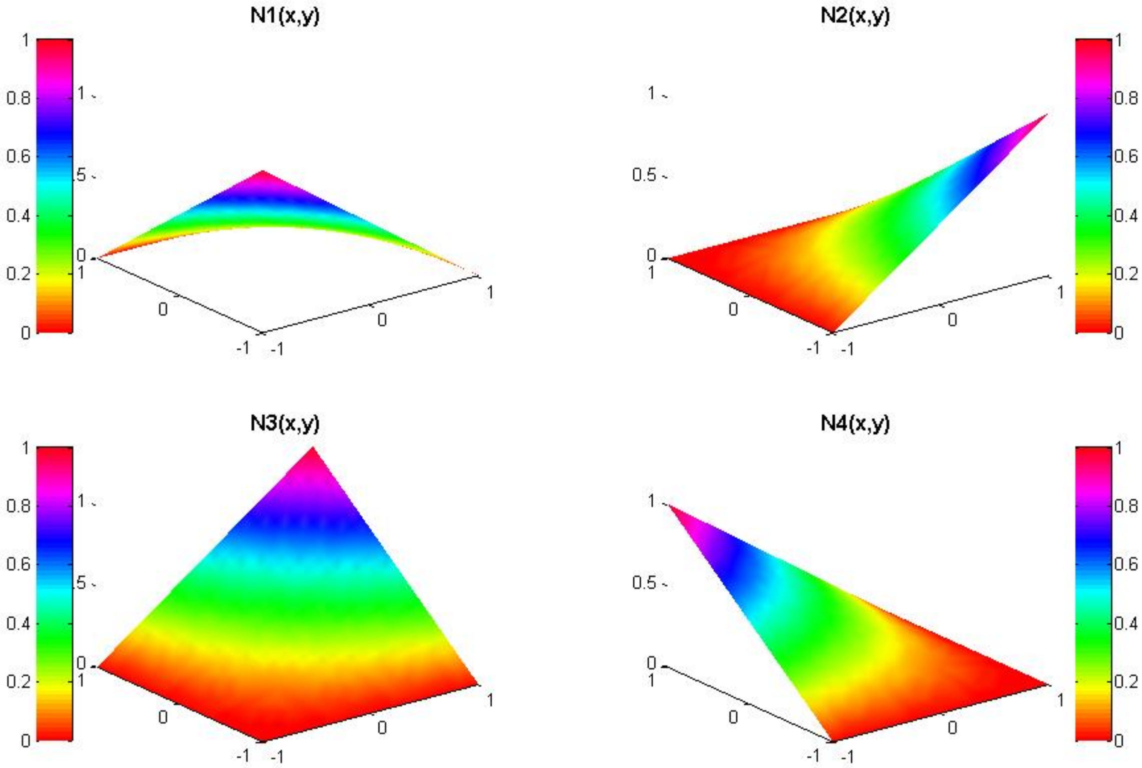

Figure 2 – Lagragian shape functions for rectangular element with 4 nodes

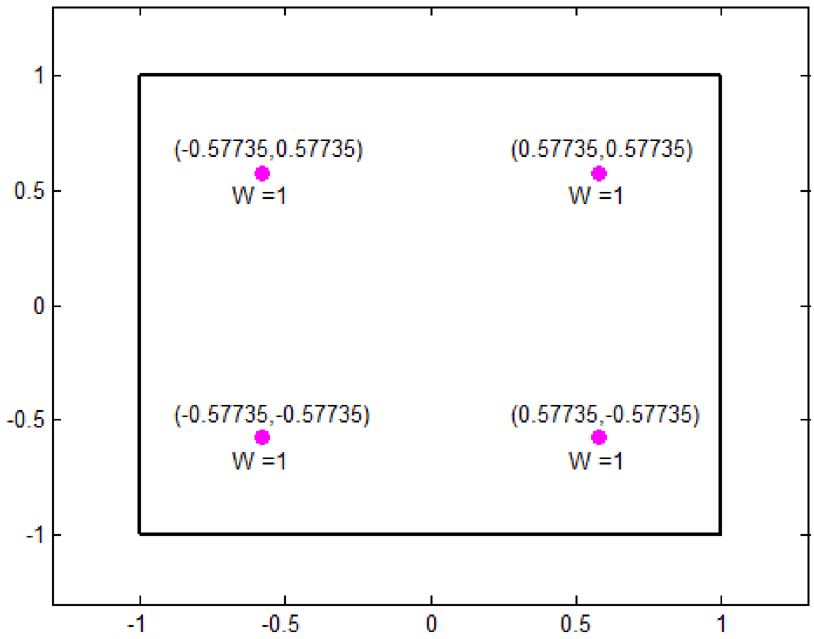

Figure 3 – Gauss quadrature point coordinates and weights

Considering Figure 2 and Figure 3 together, it can be seen that node numbering starts from bottom left node and goes on in counter clockwise direction.
Element stiffness matrix for a rectangular element with four nodes can be determined by the formulae given in equation (3) where $[D]$ is constitutive matrix and $[B]$ is strain-displacement

transformation matrix. Strain-displacement transformation matrix is defined as shown in (4) [3].

$$[k]=t.\iint_{x,y}[B]^T.[D].[B].dxdy \tag{3}$$

$$[B]=\begin{bmatrix} \frac{\partial N_1}{\partial x} & 0 & \frac{\partial N_2}{\partial x} & 0 & \frac{\partial N_3}{\partial x} & 0 & \frac{\partial N_4}{\partial x} & 0 \\ 0 & \frac{\partial N_1}{\partial y} & 0 & \frac{\partial N_2}{\partial y} & 0 & \frac{\partial N_3}{\partial y} & 0 & \frac{\partial N_4}{\partial y} \\ \frac{\partial N_1}{\partial y} & \frac{\partial N_1}{\partial x} & \frac{\partial N_2}{\partial y} & \frac{\partial N_2}{\partial x} & \frac{\partial N_3}{\partial y} & \frac{\partial N_3}{\partial x} & \frac{\partial N_4}{\partial y} & \frac{\partial N_4}{\partial x} \end{bmatrix} \qquad [\varepsilon]=[B].[u] \tag{4}$$

2.3 Topology Matrix and Global Stiffness Matrix

To form global stiffness matrix in FEM topology matrices can be used, despite requiring large memory. Since this study introduces an educational tool for FEM, it is necessary to determine topology matrix. Topology matrix relates nodal degrees of freedoms with each other briefly, by utilizing topology matrix global stiffness matrix can be determined easily as shown in equation (5) where $[C]$ is topology matrix and $diag[k]$ is a matrix which have element stiffness matrices on the diagonal [3].

$$[K]=[C].diag[k].[C]^T \tag{5}$$

2.4 Boundary Conditions and Solution

In two dimensional FEM problems, nodes have two degrees of freedom as translation in X direction and translation in Y direction. Therefore natural boundary conditions can include restrains in X direction and restrains in Y direction. Also the external forces at nodes can be defined in X direction and Y direction. In the developed software, a user friendly interface makes it possible to define nodal forces and nodal restrains in two directions. For simplicity, each direction represented with a color, X direction is represented with red, Y direction is represented with blue and both directions represented with green. For example, if user defines a nodal force in X direction than it is displayed as a red dot. Those symbols and their meanings are given in Table 1.

Function of Button	Symbol of Button
Nodal force in X direction	●
Nodal force in Y direction	●
Nodal force in both directions	●
Restrain in X direction	▶
Restrain in Y direction	▲
Restrain in both directions	■

Table 1 – Symbols for supports and nodal loads

After applying boundary conditions to global stiffness matrix and load vector, nodal displacements can be determined by using the equation (6) where $\{F\}$ is load vector.

$$\{U\}=[K]^{-1}.\{F\} \tag{6}$$

Determined nodal displacements can be used to calculate element strain and stress values by utilizing equation (4). Using the Lagragian shape functions given in Figure 2 results in linear strains and linear stresses [3]. However, strain and stress values at element centroid are considered during visualization of strains and stresses.

3 EDUCATIONAL TOOL DESCRIPTION

3.1 Introducing the Program Interface

The graphical user interface of the developed software has three menus as "Analysis", "Output" and "Report" and has eight buttons, the interface and dialog boxes which are used during problem definition are show in Figure 4. A brief explanation of problem definition is given below.

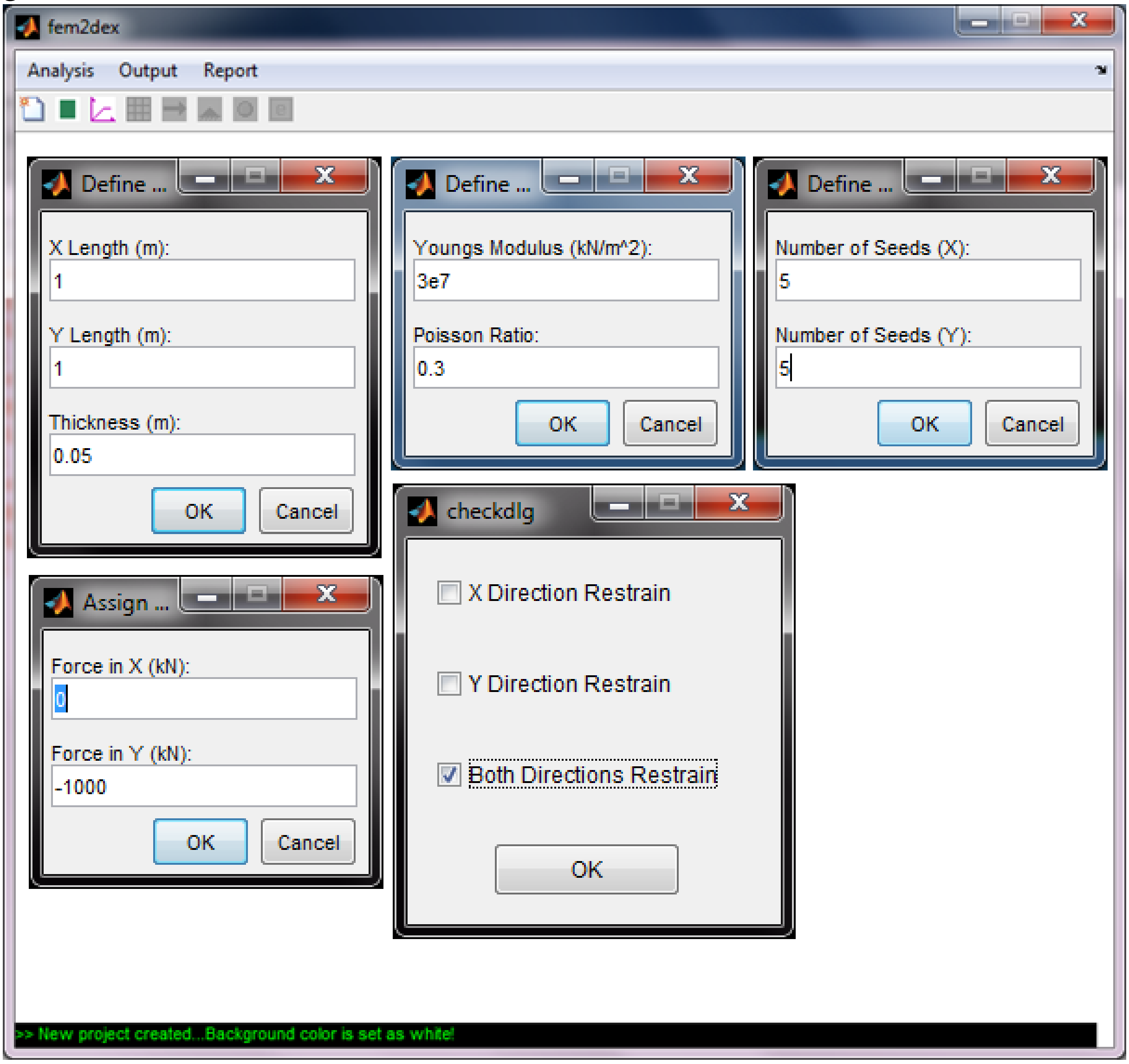

Figure 4 – Graphical user interface of the developed software

By clicking "Define Geometry" button, user can define two lateral dimensions and thickness of the rectangular part. Then, by clicking "Define Material" button, user can define material properties. When geometry definition completed, meshing tool becomes enabled. By using "Define Mesh" button user can input number of seeds in X directions and number of seeds in

Y direction. After defining mesh, user can assign nodal forces and restrains to nodes generated after meshing. The dialog windows for geometry definition, material definition, mesh definition, force assignment and restrain assignment are also shown in Figure 4. Figure 5 shows the functions and icons of the buttons mentioned above.

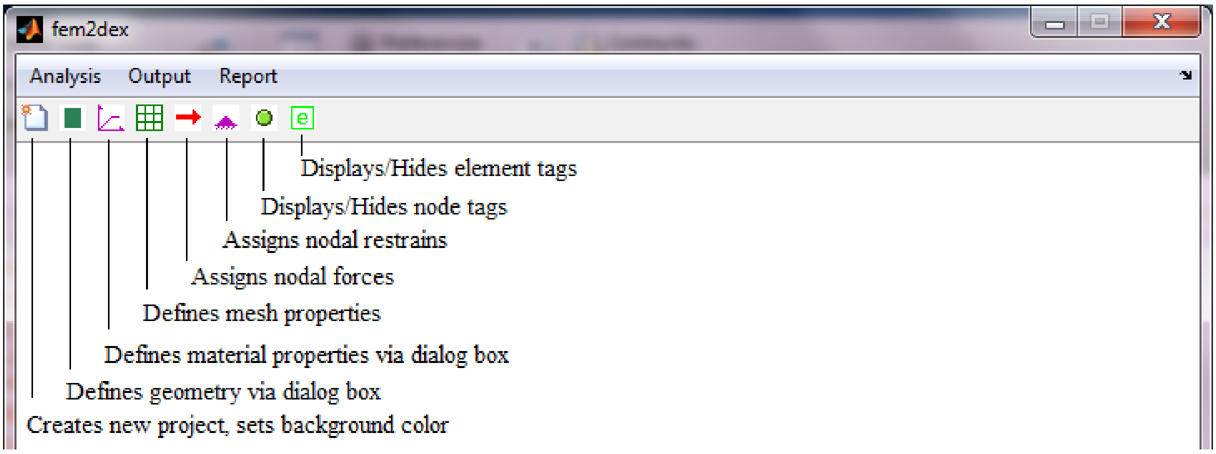

Figure 5 – Button icons and functions

3.2 Example Problem Definition and Solution

A simple rectangular plate problem and a simply supported beam problem are solved to illustrate capabilities of the developed software. In the first problem a coarse mesh preferred to explain the software briefly. In the second problem a fine mesh preferred to display visual capabilities of the software. For the plate problem, a rectangular plate with dimensions $1\,\text{m}\times 1\,\text{m}\times 0.05\,\text{m}$ is meshed into 4 elements as two in X direction and two in Y direction. A material with Young's modulus $E = 3.10^7\ \text{kN/m}^2$ and Poisson's ratio $\upsilon = 0.3$ is defined. The bottom corner nodes of the rectangular plate are restrained in both X and Y directions and two equal nodal forces with a value of $1000\ \text{kN}$ are assigned to the top corner nodes in X direction. For the beam problem, a beam with dimensions $5\,\text{m}\times 0.5\,\text{m}\times 0.25\,\text{m}$ is meshed into 40 elements as 10 in X direction and 4 in Y direction. The same material with plate problem is used. The bottom corner nodes of the beam are restrained in both X and Y directions and a $1000\ \text{kN}$ nodal load assigned to the top-mid node of the beam in Y direction. Problem geometries, mesh properties and boundary conditions of these two problems are shown in Figure 6 and Figure 7 respectively.

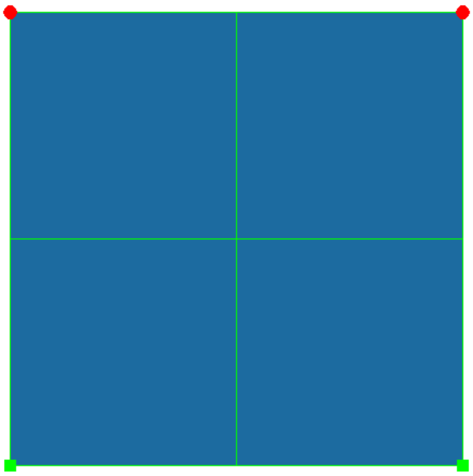

Figure 6 – Plate problem geometry, mesh and boundary conditions

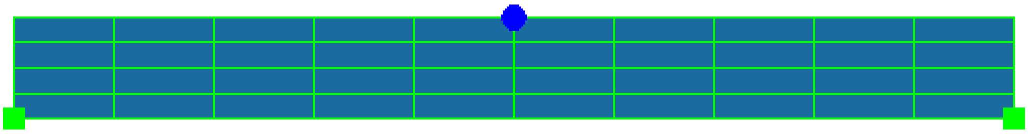

Figure 7 – Beam problem geometry, mesh and boundary conditions

As thicknesses of these two geometries are smaller with respect to other two dimensions of the geometries, both of the problems are solved as plane stress problems. Deformed shape of both geometries are shown in Figure 8 and Figure 9 respectively. Also stress distributions for beam problem are shown in Figure 10. Stress distributions of the plate problem are not given here, since it would not be satisfying because of the coarse mesh defined.

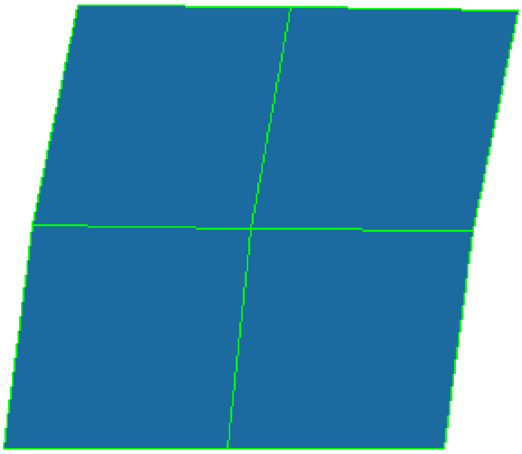

Figure 8 – Deformed shape of the plate

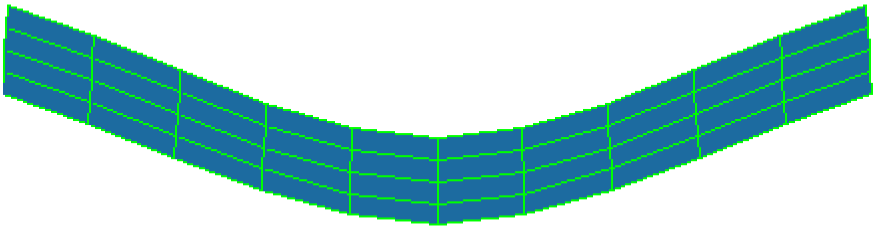

Figure 9 – Deformed shape of the beam

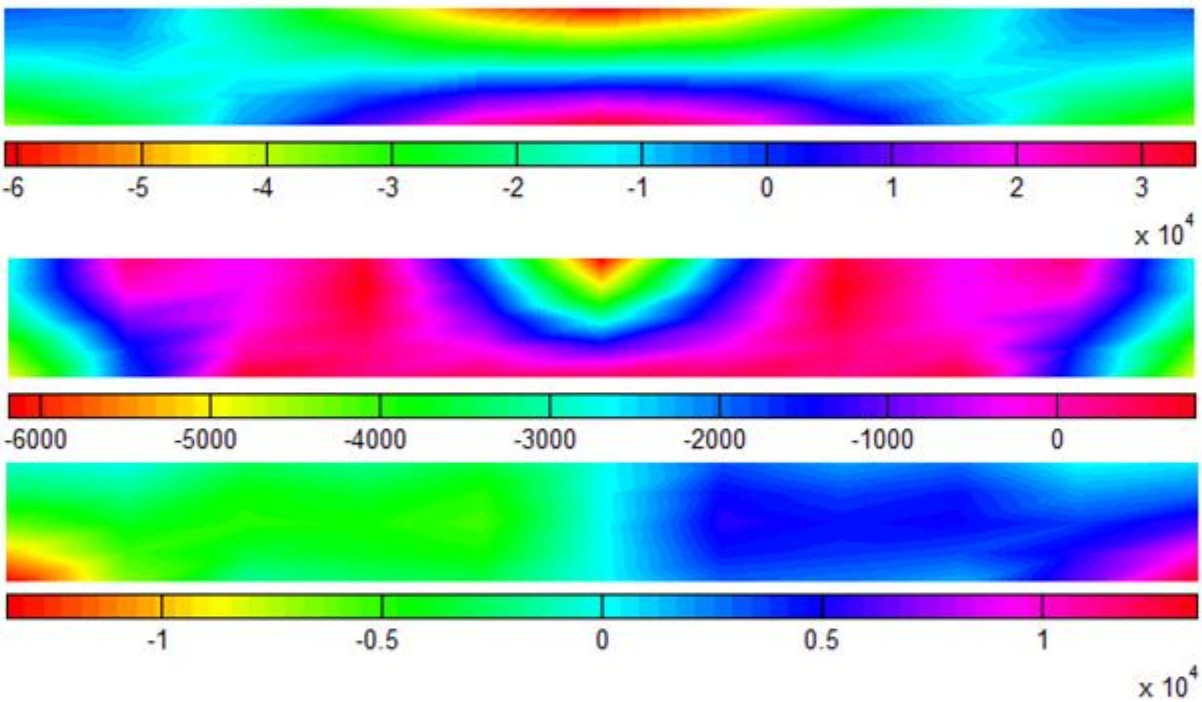

Figure 10 – Stress distributions (kN/m^2) for σ_X , σ_Y and τ_{XY} respectively

3.3 Example Problem Outputs

As stated before the developed software can generate a solution report which includes material properties, problem state and all matrices of FEM procedure. Those matrices can also be exported to Microsoft Excel sheets. To illustrate these capabilities, constitutive matrix and element stiffness matrix of plate problem are given in equations (7) and (8).

$$[D]=\begin{bmatrix} 32967032,97 & 9890109,89 & 0,00 \\ 9890109,89 & 32967032,97 & 0,00 \\ 0,00 & 0,00 & 11538461,54 \end{bmatrix} \tag{7}$$

$$[k]=\begin{bmatrix} 741758,24 & 267857,14 & -453296,70 & -20604,40 & -370879,12 & -267857,14 & 82417,58 & 20604,40 \\ 267857,14 & 741758,24 & 20604,40 & 82417,58 & -267857,14 & -370879,12 & -20604,40 & -453296,70 \\ -453296,70 & 20604,40 & 741758,24 & -267857,14 & 82417,58 & -20604,40 & -370879,12 & 267857,14 \\ -20604,40 & 82417,58 & -267857,14 & 741758,24 & 20604,40 & -453296,70 & 267857,14 & -370879,12 \\ -370879,12 & -267857,14 & 82417,58 & 20604,40 & 741758,24 & 267857,14 & -453296,70 & -20604,40 \\ -267857,14 & -370879,12 & -20604,40 & -453296,70 & 267857,14 & 741758,24 & 20604,40 & 82417,58 \\ 82417,58 & -20604,40 & -370879,12 & 267857,14 & -453296,70 & 20604,40 & 741758,24 & -267857,14 \\ 20604,40 & -453296,70 & 267857,14 & -370879,12 & -20604,40 & 82417,58 & -267857,14 & 741758,24 \end{bmatrix} \tag{8}$$

Topology matrix, load vector, displacement vector and global stiffness matrices can also be illustrated via the developed software.

4 CONCLUSION

The developed software meets the aim of this study, which is to prepare an educational software to create two dimensional FEM examples and solutions easily. The product of this study can easily be used for FEM courses in graduate level. A large variety of examples from basic examples to complicated examples and their solutions can be prepared with the developed software. Outputs of the software are also very impressive. A solution report in text format can be generated after completing the analysis. Visual outputs also meet demands, visual capabilities of the software suppress many commercial FEM software.

Results given by the software are compared with a reliable FEM software which is a well-known earthquake engineering simulation tool named as OpenSees [4]. The comparison ended up with a good harmony of the results which verifies the developed software.

REFERENCES

[1] MATLAB 2014a, The MathWorks, Inc., Natick, Massachusetts, 2014.

[2] Mase, G. T., Mase, G. E., *Continuum Mechanics for Engineers*, CRC Press, 1999.

[3] Desai, C. S., Kundu, T., *Introductory Finite Element Method*, CRC Press, 2001.

[4] McKenna, F., Fenves, G. L., Scott, M. H., and Jeremic, B., *Open System for Earthquake Engineering Simulation (OpenSees)*. Pacific Earthquake Engineering Research Center, University of California, Berkeley, CA, 2000.