Modelling of Isogeometric Analysis for Plane Stress Problem Using Matlab GUI

Norliyana Farzana Binti Zulkefli, Ahmad Razin Bin Zainal Abidin

Faculty of Civil Engineering, Universiti Teknologi Malaysia, Malaysia

arazin@utm.my

Keywords: Isogeometric analysis; plane stress; MATLAB.

Abstract. Isogeometric analysis (IGA) is a recently developed method in computational mechanics that offers the possibility of integrating the analysis and the design process into a single and unified process compared to other numerical methods such as finite element method (FEM) that is more widely used. While many research studies focused more on the performance of IGA analysis, not many of them were looking into the graphical aspect, which is more related to CAD. Therefore, in this study, modelling of IGA particularly for plane stress problem was developed using MATLAB graphical user interface (GUI) in order to understand the unique formulation of IGA. The first objective was to identify the IGA formulation to analyse a plane stress problem. The formulation was then used to guide the execution of "Plane Stress – IGA" codes in MATLAB program. Finally, a dedicated interface for the "Plane Stress – IGA" program was developed using GUI-MATLAB, considering all the inputs and outputs required for a plane stress analysis. In achieving the three objectives of this study, a basic step-by-step to develop a computer program, known as a PDC (Program Development Cycle) was adopted. Basically, it involves identifying the problem background through literature reviews, designing the sequences of program using flowcharts, testing and debugging the codes, and lastly, developing the user-interface. The successfully developed "Plane Stress – IGA" program shows that the IGA formulation is a user-friendly computational method that can be developed using MATLAB software. The process flow of IGA calculation is similar to FEM where it requires geometry data, material properties, meshing parameters, loading and support conditions before the complete calculation can be executed. Compared to FEM, IGA has more advantages on the graphical aspects where the stress plots can be easily done without the need to deal with element-by-element data as in the FEM. It is hoped that the developed program can increase the understanding of the IGA formulation through the guided input and output process created on the MATLAB interface. In fact, the dedicated GUI makes it easier for users to deal directly with the plane stress analysis program without having to change the codes in MATLAB scripts.

Introduction

Background of the Study

By definition, plane stress is a general two-dimensional type of plane analysis in the theory of plane elasticity, apart from plane strain. Plane stress happened when there is merely non-zero components act in one plane of the material particle. Therefore, plane stress element replicates structure with thin body which is subjected to in-plane loading or boundary stresses. Plane stress solid structure can only be analysed in the x and y directions. On the other hand, the z direction would be irrelevant in the analysis due to it small dimension compared to x and y direction. Practically, plane stress is used to represent shear walls, load bearing walls, steel web and so forth. Plane stress problem can be solved through an analytical approach, but it is very often in engineering analysis that plane stress problems are solved using numerical approach considering for example finite element method (FEM).

The FEM has developed into a key, indispensable technology in the modelling and simulation of advanced engineering systems in various fields like housing, transportation, communications, and so on [5]. As stated by Liu and Quek, FEM is a numerical method seeking an approximated solution of the distribution of field variables in the problem domain that is difficult to obtain analytically. The FEM was first used to solve problems of stress analysis to determine the distribution of some field variable like the displacement in stress analysis. Hence, FEM can analyse problem related to plane stress solids and structures. There are plenty of well developed FEM packages built to solve most of engineering problems related to solids and structures. Nevertheless, FEM has its own drawbacks and limitations. For instances, analyst has to spend most of the time in mesh creation which is not cost effective. Besides this, FEM requires the need of recover accuracy of stresses in post processing stage and posses difficulty in adaptive analysis in ensuring high accuracy and limitation in analyzing of problems under large deformation, crack and simulating breakage of material [7].

As a result, Isogeometric Analysis (IGA) is used instead of Finite Element Analysis (FEA). Isogeometric analysis (IGA) represents a recently developed technology in computational mechanics that offers the possibility of integrating methods for analysis and Computer Aided Design (CAD) into a single, unified process. The implications to practical engineering design scenarios are profound, since the time taken from design to analysis are greatly reduced, leading to dramatic gains in efficiency [6]. IGA seeks to unify the field of Computer Aided Design (CAD) and numerical analysis such as FEM and Meshfree, hence bridging the gap of CAD and Computer Aided Engineering (CAE). Among the computational geometry technologies used in IGA, Non-Uniform Rational B-Splines (NURBS) is most widely used in engineering design. The pre-eminence of NURBS in engineering design as compared to other computational geometry technologies is generally because of the convenient for free-form surface modelling. For example, it can represent exactly all conic sections. It also exist many efficient and numerically stable algorithms to generate NURBS objects. Since NURBS is the most popular computational geometry in CAD, it is selected to be the basis function in the derivative of domain equation and analysis [7].

In this study, the plane stress problem is solved by a numerical method that is Isogeometric Analysis (IGA). The modelling is made by using a matrix laboratory (MATLAB) program and also a graphical user interface (GUI) feature available in MATLAB. MATLAB is used because it is a very powerful software package that has many built-in tools for solving problems and developing graphical illustrations. The simplest method for using the MATLAB product is interactively when an expression is entered by the user and MATLAB responds immediately with a result. It is also possible to write scripts and programs in MATLAB, which are essentially groups of commands that are executed sequentially [1]. For that reason, MATLAB is used to solve the plane stress problem instead of other mathematical and graphical software with numerical, graphical, and programming abilities. What's more, by using GUI, the plane stress problem can be solved easily by anyone through the interface created in GUI. So, IGA and FEM program can be created in MATLAB to solve the plane stress problem.

Statement of the Problem

Even though, there are a vast of research done in the Computer Aided Design (CAD), but there is still less research done between the CAD and Computer Aided Engineering (CAE) relationship. Consequently, illustrate the communication gap between CAD and CAE which has been present. Therefore, this study is done to fill the gap between CAD and CAE besides analyse the problem at hand. At this preliminary stage of study, a few problem of concern are as follows:

1. Solve the plane stress structure problem using Isogeometric Analysis (IGA).
2. Resolve the problem using matrix laboratory program (MATLAB).
3. Report the IGA for plane stress based on MATLAB.

Objectives of the Study

The objectives of this study are as follows:

1. To identify the formulation for plane stress problem using Isogeometric Analysis (IGA).
2. To execute a “Plane Stress – IGA” program for the plane stress problem using MATLAB scripts.

3. To develop a dedicated interface for the “Plane Stress – IGA” program using GUI in MATLAB.

Significance of the Study

The significance of this study is as follows:

1. The develop program would acts as a starting point to reduce problem regarding CAD – CAE communication.
2. IGA is more efficient in CAE to provide a simple program for analysing plane stress structure.

Scope of the Study

The scope of this study is as follows:

1. The plane stress problem would be based on the formulation in the theory of plane elasticity.
2. The IGA for the plane stress problem would be solving using the program develops in MATLAB.

Literature Review

This section discuss more on the previous study regarding plane stress problem solve using IGA, which will then be applied in MATLAB.

Properties and Differences of FEM and IGA

In this section we overview the main properties of the most common basis functions used in FEM and IGA and of the corresponding parameterizations applied for the discretization of the continuum geometry. In both FEM and IGA, uni- and bi-variate parameterizations represent contact curves/surfaces respectively in 2Dand 3Dsettings, and these are inherited respectively from the bi- and tri-variate parameterizations of the continuum in a straightforward fashion. In the following, ds and dp denote the dimensions of the physical and of the parametric space, respectively. Within an isoparametric approach, the same parameterizations are adopted for the field of the unknowns. In the large deformation framework, this leads to a significant impact of the properties of the basis on the quality of the results [3].

As a comparison, FEM and IGA methods depart at the stage of mesh creation. Both methods start to diverge during the construction of shape functions. In FEM, the shape functions are constructed using predefined elements and the shape functions are the same for the entire element. On the other hand, the shape functions constructed in IGA are based on knot vector of the patch of the entire domain. This means the NURBS parameter space is local to patches rather than local to elements in Finite Element Analysis (FEA). Both analysis methods follow the same procedure once the global discretized system equation is established [7].

The use of basis, NURBS to model the geometry and serve as the basis for the solution space of numerical method is the main idea of IGA. This intention of using the same basis for geometry design and analysis is called isoparametric concept and it is common in finite element analysis. The fundamental difference between this new concept of isogeometric analysis and the traditional isoparametric FEA is that, in classical FEA, the basis chosen to approximate the unknown solution fields is then used to approximate known geometry. Contradictorily, IGA turns the idea around by selecting a basis capable of exactly representing the known geometry and uses it as a basis to approximate the solution field [7].

NURBS have been a mainstay of geometric design for years due to the flexibility and precision in its nature. As NURBS will be the basis of analysis, it can improve upon the traditional piecewise polynomial basis functions from FEM, providing unprecedented accuracy and robustness across wide array of applications (Cottrell, Hughes and Bazilevs, 2009). The key concept outlined by Hughes et al. (2005) was to employ NURBS not only as a geometry discretization technology, but also as a discretization tool for analysis, attributing such methods to the field of 'Isogeometric Analysis' [7].

Isogeometric Analysis for Plane Stress

The subject of the research is the study of plane stress isogeometric finite element according to the canon of Isogeometric Analysis. Isogeometric Analysis is a group of FEM techniques assuming the same functions for element geometry description (structures) and displacement state description (shape functions). Geometry description and shape function are usually used by B- Splines or T-Splines [4]. Application of this type of functions enables the universal element geometry description. By means of a small amount of parameters we are able to descri- be complexly shaped element with high precision. Thanks to the usage of B-Splines the construction description in CAD systems and FEM system was standardized [4]. According to Zbigniew Kacprzyk and Zbigniew Trybocki (2013), their study on isogeometric plane stress analysis found out that there is high precision of plane stress problem with small amount of meshing. They also mentioned that the finite element rigidity matrix of IGA yield in higher numerical cost as compared to isoparametric phrase [7].

Formulation

Partial Differential Equation (PDE) of plane stress. PDE of plane stress in terms of displacement:

$$[\partial][E][\partial]^T\{u\}^T = \{F\}^T \tag{1}$$

where, $[\partial] = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix}$ is the differential operator matrix, $[E] = \frac{E}{1-v^2}\begin{bmatrix} 1 & v & 0 \\ v & 1 & 0 \\ 0 & 0 & \frac{(1-v)}{2} \end{bmatrix}$ is the material properties, $\{u\}$ is the displacement vector, $\{F\}$ is the force vector.

FEM formulation for plane stress

Degree of Freedom and Shape Functions of plane stress element. Assume polynomial interpolation function for the 4-noded and 8-noded element:

$$U = a_1 + a_2x + a_3y + a_4xy \tag{2a}$$

$$U = a_1 + a_2x + a_3y + a_4x^2 + a_5y^2 + a_6xy + a_7x^2y + a_8xy^2 \tag{2b}$$

In vector forms, Equations (2a) and (2b) can be written as:

$$U = \{x\}\{a\}^T \tag{2c}$$

Degree of freedom (DOF), Ui of Eq. (4) in matrix forms:

$$[x|]\{a\}^T = \{\bar{U}\}^T \tag{2d}$$

where $[x|]$ is the matrix which contains the evaluated values of monomial, $\{\bar{U}\}^T$ is the vector of DOF.

Polynomial coefficients obtained after solving the simultaneous equations:

$$\{a\}^T = [x|]^{-1}\{\bar{U}\}^T \tag{2e}$$

Inserting the coefficients from Eq. (2e) into Eq. (2c):

$$U = \{x\}[x|]^{-1}\{\bar{U}\}^T \tag{2f}$$

Shape functions in vector forms:

$$\{N\} = \{x\}[x|]^{-1} \tag{2g}$$

By performing multiplication in Eq. (2g), the shape functions for 4-noded plane stress element:

$$N_1 = 1 - \frac{x}{a} - \frac{y}{b} + \frac{xy}{ab},\ N_2 = \frac{x}{a} - \frac{xy}{ab},\ N_3 = \frac{xy}{ab},\ N_4 = \frac{y}{b} - \frac{xy}{ab} \tag{2h}$$

The shape functions for 8-noded plane stress element:

$$N_1 = \left(\frac{2}{a^2} - \frac{2y}{a^2 b}\right)x^2 + \left(\frac{5y}{ab} - \frac{3}{a} - \frac{2y^2}{ab^2}\right)x + \frac{2y^2}{b^2} - \frac{3y}{b} + 1,\quad N_2 = \left(\frac{2}{a^2} - \frac{2y}{a^2 b}\right)x^2 + \left(-\frac{1}{a} - \frac{y}{ab} - \frac{2y^2}{ab^2}\right)x,$$
$$N_3 = \frac{2x^2 y}{a^2 b} + \left(\frac{2y^2}{ab^2} - \frac{3y}{ab}\right)x,\ N_4 = \frac{2x^2 y}{a^2 b} + \left(-\frac{2y^2}{ab^2} - \frac{y}{ab}\right)x + \frac{2y^2}{b^2} - \frac{y}{b},\ N_5 = \left(\frac{4y}{a^2 b} - \frac{4}{a^2}\right)x^2 + \left(\frac{4}{a} - \frac{4y}{ab}\right),$$
$$N_6 = \left(\frac{4y}{ab} - \frac{4y^2}{ab^2}\right)x,\ N_7 = \frac{4xy}{ab} - \frac{4x^2 y}{a^2 b},\ N_8 = \left(\frac{4y^2}{ab^2} - \frac{4y}{ab}\right)x - \frac{4y^2}{b^2} + \frac{4y}{b} \tag{2i}$$

Interpolation of displacements, *u* and *v* by the shape functions in vector form:

$$\begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{Bmatrix} \tag{2j}$$

$$\begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & \cdots & N_7 & 0 & N_8 & 0 \\ 0 & N_1 & 0 & N_2 & \cdots & 0 & N_7 & 0 & N_8 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ \vdots \\ u_7 \\ v_7 \\ u_8 \\ v_8 \end{Bmatrix} \tag{2k}$$

Generally represent Eq. (2j) and Eq. (2k) as:

$$\{u\}^T = [N]\{\hat{u}\}^T \tag{2l}$$

Discretization by Galerkin - Weighted Residual Method (WRM). Substitute Eq. (2) into Eq. (1):

$$[\partial][E][\partial]^T[N]\{\hat{u}\}^T = \{F\}^T \tag{3a}$$

Multiplying Eq. (3) with shape function matrix [N] and integrate:

$$\int_x \int_y [N]^T ([\partial][E][\partial]^T [N]\{\hat{u}\}^T - \{F\}^T)\, dy\, dx = 0 \quad (3b)$$

Integration by Parts (IBP). Conducting IBP:

$$\int_x \int_y [N]^T [\partial][E][\partial]^T [N]\{\hat{u}\}^T dy\, dx = \int_x \int_y [N]^T \{F\}^T\, dy\, dx + \int_x [N]^T \{b\}^T ds \quad (4a)$$

Solve with:

$$[k]\{\hat{u}\}^T = \{q\} + \{b\} = \{r\} \quad (4b)$$

where $[k] = \int_x \int_y [N]^T [\partial][E][\partial]^T [N] dy\, dx$, $\{q\} = \int_x \int_y [N]^T \{F\}^T\, dy\, dx$, $\{b\} = \int_x [N]^T \{b\}^T ds$

IGA formulation for plane stress.
NURBS of IGA. Two-dimensional NURBS:

$$R_{i,j}^{p,q}(\xi,\eta) = \frac{N_{i,p}(\xi)M_{j,q}(\eta)\omega_{i,j}}{\sum_{i=1}^{n}\sum_{j=1}^{m} N_{i,p}(\xi)M_{j,q}(\eta)\omega_{i,j}} \quad (5a)$$

where p = polynomial order in x-direction, q = polynomial order in y-direction, ξ = knot vector in x-direction, η = knot vector in y-direction, N = shape functions in x-direction, M = shape functions in y-direction, ω = weighting. Taking the weights are equal to one, Equation (3) will become:

$$R_{i,j}^{p,q}(\xi,\eta) = N_{i,p}(\xi)M_{j,q}(\eta) \quad (5b)$$

Eq. (5b) is exactly the same as B-spline surface basis which is known as a special case for NURBS in IGA. Using a set of knot vector, the B-spline basis functions are defined recursively starting with piecewise constants:

$$p = 0$$

$$N_{i,0}(\xi) = \begin{cases} 1 & if \quad \xi_i \le \xi < \xi_{i+1} \\ 0 & otherwise \end{cases} \quad (5c)$$

For p = 1, 2, 3,…,

$$N_{i,p}(\xi) = \frac{\xi - \xi_i}{\xi_{i+p} - \xi_i} N_{i,p-1}(\xi) + \frac{\xi_{i+p+1} - \xi}{\xi_{i+p+1} - \xi_{i+1}} N_{i+1,p-1}(\xi) \quad (5d)$$

*Knot vectors.*A knot vector in one dimension is an increasing set of coordinates in the parameter space written:

$$\Xi = \{\xi_1, \xi_2, \dots, \xi_{n+p+1}\}$$

where $\xi_i \in R$ is the i^{th} knot, i is the knot index, i = 1, 2,…,n+p+1, p is the polynomial order, n is the number of basis function to construct the B-spline curve.

The knots partition the parameter space into elements. Element boundaries in the physical space are simply the images of knot lines under the B-spline mapping. Non-uniform knot vectors are being used in this project. Knot values can be repeated that is one knot can take on the same value. The multiplicities (repetitive) of knot values have important implications for properties of the basis. A knot vector is said to be free if its first and last knot values appear p+1times. This distinguishes between knots and "nodes" in FEA. Knot vector, polynomial order and number of basic functions plays important role in the determination of partition of patch in B-spline [7]. Thus, the solution in IGA is defined by:

$$S(\xi,\eta) = \sum_{i=1}^{n}\sum_{j=1}^{m} N_{i,p}(\xi)M_{j,q}(\eta)B_{i,j} \tag{5}$$

Hence, $S(\xi,\eta)$ is the displacement or deformation as of $\{\hat{u}\}^T$ in FEA.

Methodology

Program Development Cycle

This research is done by using a systematic process of developing a program which is known as Program Development Cycle (PDC). In PDC, five steps are arranged in order to solve the IGA for plane stress problem which are analyse, design, code, test and debug, and documentation. The following are the steps:

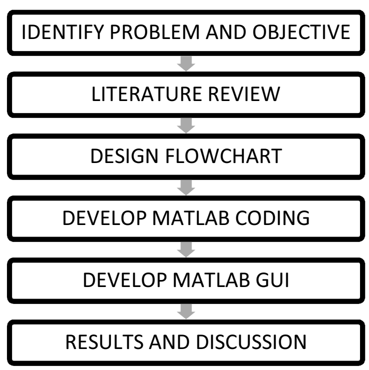

Figure 1: The procedure of works needed in the study

Table 1: The descriptions of procedures in the study

PROCEDURES	DESCRIPTIONS
IDENTIFY PROBLEM AND OBJECTIVE	ANALYSE Analyse the problem of the study
LITERATURE REVIEW	Identify IGA formulation.
DESIGN FLOWCHART	DESIGN Plan the solution. Distinguish the input, process, and output of the study.
DEVELOP MATLAB CODING	CODE The coding is developing based on formulation in literature review. TEST AND DEBUG The coding is run to check for errors and update coding.
DEVELOP MATLAB GUI	Create an interface based on the coding and produce figure output.
RESULTS AND DISCUSSION	DOCUMENTATION Review the objectives of the study based on the outcomes of the study.

Procedures of Study

Flowchart. The flowchart consists of special geometric symbol connected by arrows. In fact, within each symbol there is a phrase representing the activity at each step for the flow of the flowchart. The shape of the symbol indicates the type of operation.

Develop coding in MATLAB. The coding is developed based on previous research regarding the IGA and its formulation. However, some changes have been done to the original coding as to improve the IGA coding.

Develop GUI in MATLAB. After developing the MATLAB coding for the IGA, a MATLAB GUI is constructed to ease user to input data for analysis using MATLAB.

Results and Discussions

Result and Discussion for Objective 1: To identify the formulation for plane stress problem using Isogeometric Analysis (IGA).

Actually, this research is not done to study the IGA formulation in detail but to intergrade the IGA in solving problem related to plane stress either for cantilever nor simply supported structures. Therefore, to understanding how to solve the problem easily, a flowchart is constructed to show the sequence of calculation as well as the expected end result from the calculation by using the previous study formulation of solving IGA using MATLAB.

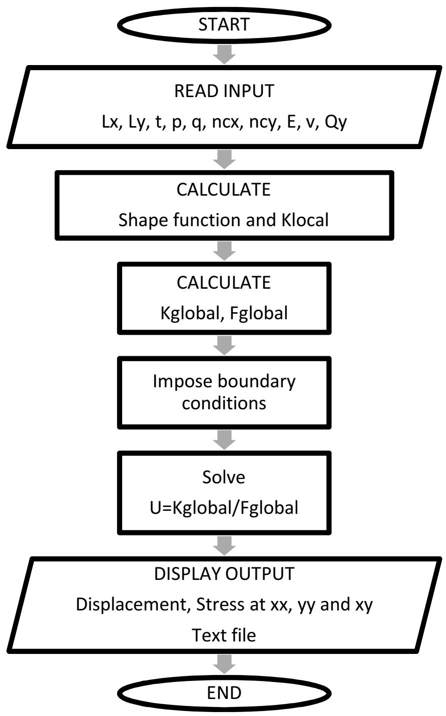

Figure 2: Flowcharts for coding based on the IGA formulation.

Result and Discussion for Objective 2: To execute "Plane Stress – IGA" program for the plane stress problem.

Upon understanding how to solve plane stress problem, MATLAB coding are wrote to develop the "Plane Stress – IGA" program. The coding written are mostly referred to previous study coding with some modification on how to run the coding faster than the previous study coding. At the same time, the new coding is written to make it able for user to try other input data before running the coding. For example, the new coding for the geometry of the structure and its deflection figure helps user to identify the shape of the plane stress problem. Therefore, this creates of a model that can be easily understood by user. *Refer coding in MATLAB.

Result and Discussion for Objective 3: To develop "Plane Stress – IGA" program interface using GUI in MATLAB.

Based on the coding, GUI is generated by altering the coding into guide which is a figure interface. This figure interface which consists of many components such as push button, edit text, axes, and static text helps user to understand the flow on how to solve the plane stress problem using IGA formulation. Thus, this is more efficient than ordinary coding which require user to input data at the command window. Besides, through the GUI created, user can see the results of their input data directly as the axes in the figure generated the plot for all geometry, meshing, displacement, and stress output.

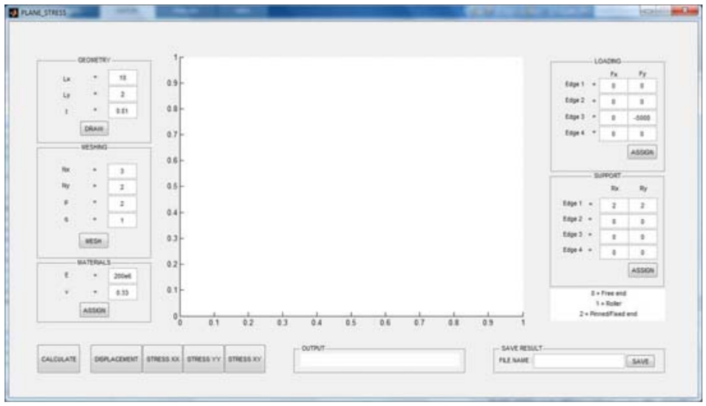

Figure 3: The developed 'PLANE_STRESS' program in MATLAB GUI.

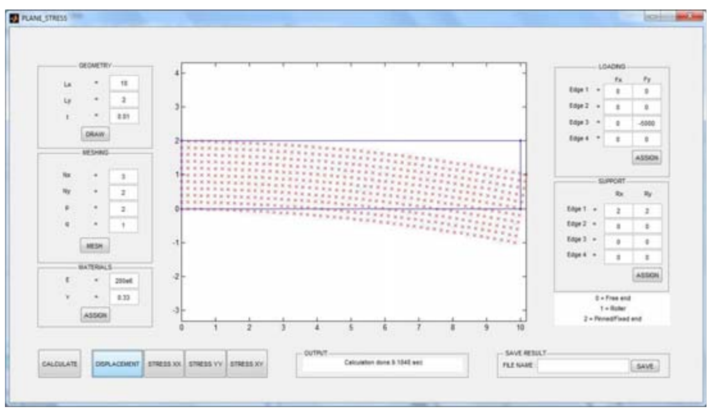

Figure 4: The additional displacement figure based on the plane stress problem on cantilever.

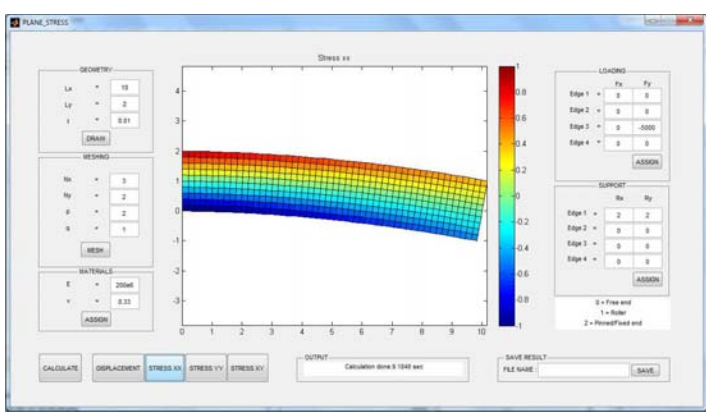

Figure 5: Among the stresses output figure from MATLAB GUI.

Conclusion

The following conclusion can be drawn for this study:

1. The formulation of IGA for analysing a plane stress problem was identified. A process flow to execute the formulation in computer program was designed in a flowchart to get a better understanding on the sequence of calculation for the plane stress problem.
2. The execution of a “Plane Stress – IGA” program was successfully conducted in MATLAB scripts, taking into accounts the step-by-step formulation of plane stress analysis by IGA from the designed flowchart. The coding was extended into a plane stress problem allowing for different boundary conditions and loading conditions.
3. The development of a dedicated “Plane Stress – IGA” program interface using GUI in MATLAB was conducted to make the program more user-friendly, for example, by providing an interface that can guide user to input data and obtain the outputs directly on the axes of the interface. The program also allows the user to save the analysis results for a later use in the future.

This study can be claimed as an initial step for the development of a complete IGA program for structural analyses and other engineering problems. Although the computational programming seems to be harder to be developed compared to the conventional FEM due to the complexity of formulation, especially on the shape function, IGA could result in reduced overall analysis time and possible error in converting CAD files into CAE program/software. In a nutshell, it is worthwhile to invest in developing IGA program as IGA could possibly replace the other conventional numerical techniques currently used in the field of engineering.

References

[1] Attaway, S. (2013). *Matlab: A Practical Introduction to Programming and Problem Solving* (Third Edit). United States: Butterworth-Heinemann.

[2] Cook, R. D., Malkus, D. S., Plesha, M. E., & Witt, R. J. W. (2002). Concept and Applications of Finite Element Analysis.

[3] De Lorenzis, L., Wriggers, P., & Hughes, T. J. R. (2014). Isogeometric contact: A review. *GAMM Mitteilungen*, *37*(1), 85–123.

[4] Kacprzyk, Z., & Trybocki, Z. (2013). Isogeometric plane stress analysis, *1*(August), 7–8.

[5] Liu, G. R., & Quek, S. S. (2013). *The Finite Element Method: A Practical Course* (Second Ed). United States of America: Butterworth-Heinemann.

[6] Nguyen, V. P., Rabczuk, T., & Bordas, S. P. a. (2013). Isogeometric analysis: an overview and computer implementation aspects, 1–95. http://doi.org/10.1016/j.matcom.2015.05.008

[7] Xian, C. Z., Ridza, M., Yasreen, A., & Yassin, M. (n.d.). ISOGEOMETRIC ANALYSIS OF PLANE STRESS STRUCTURE.

[8] Yasreen, A., Yassin, M., Akhbar, a L., Noor, M., Kasiman, H., & Zhafri, M. (2013). Linear and Nonlinear Finite Element Analysis for Solids and (Unedited 1St Draft).