Software Engineering Development of Finite Element Method Programming Applications in 2D Frame Structures Using Python Programs

N Nazaruddin 1; Richard Siallagan 1

1Department of Mechanical Engineering, Universitas Riau, 28293, Indonesia

nazaruddin@eng.unri.ac.id

Abstract. The application of the Finite Element Method as one of the numerical methods to solve various engineering problems is of course inseparable from the development of computers with various other related fields such as Computer Aided Design (CAD) and Computer Aided Engineering (CAE) which continuously becomes a concentration of interest in the engineering field. Element analysis has so far been in great demand by various industries due to its reliability and speed in terms of optimization in the world of design and analysis. There are many finite element software such as Abaqus, Nastran, Ansys, CosmosWork, LS-Dyna, Pro-Mecanica, SAP2000. However, the application is paid at a relatively high cost and has quite strict licensing rules, of course this is an obstacle for students at the University of Riau who want to use it. From these problems the author is interested in developing a programming application that can be an alternative to analyzing a structure. The solution that the author can propose is to develop an application using the python programming language to analyze a two-dimensional (2D) frame structure that is under load and is in an elastic condition. From the results of the research that has been carried out, the simulation results of the frame A model obtained data on the 2nd frame model of frame A, the horizontal displacement of \(-0.06098\) m, the vertical displacement of \(-0.00002857\) m, and the angular displacement of \(0.00762\) rad. These results are compared with manual solutions, analytical solutions, and previous research, it can be concluded that the results of calculations in the developed application show values that are close to that comparison, where the percentage error is not more than \(3\)%, with the highest percentage difference of \(2.5\)%.

1. Background
The frame is a structure consisting of a horizontal beam that is connected using a certain type of connection to achieve a balance and stability of the planned structure.[3]

Structural analysis must be carried out in the planning of a structural system, through an in-depth study it can be estimated that the structure can withstand the load from various other external factors that appear on the construction.

The more complicated a structure, the more difficult it will be if it is calculated manually and will take a long time and inefficient in the process, so the recommended solution is to use a numerical approach through the Finite Element Method (FEM) in order to obtain a solution that is close to exact [2]. Finite Element Method numerically solves complex calculation processes
through computational approaches by breaking down structural/construction parts into small elements [5].

At this time there are many software based on the Finite Element Method that can analyze a structure quickly and efficiently compared to manual analysis such as Abaqus, Nastran, Ansys, CosmosWork, LS-Dyna, Pro-Mecanica, SAP2000. However, the application is not open (close source) and has a fairly expensive price and the application has quite strict licensing rules. This is of course an obstacle for users, especially students of Mechanical Engineering at the University of Riau who want to use it. Based on these problems, the author intends to develop the application to analyze 2D frames using the Python programming language with the hope that more applications are made by Mechanical Engineering students at the University of Riau to reduce worries in using licensed applications and can meet the need to be able to analyze structures practically.

2. Literature Study

The frame is a frame consisting of two or more construction parts that are connected for stability, generally able to withstand moment forces, shear forces and axial forces. Frames are generally divided into 2, namely as follows:

1. Plane frames, frames that can withstand loads in the flat direction only (x and y axis).

2. Space frame, a frame that can withstand all loading directions (x, y, and z axis).

![Figure 1. Plane Frame](image)

![Figure 2. Space Frame](image)

Analysis of the strength of the frame can be done in various ways, one of which is using the finite element method. The equation for the global stiffness matrix of the frame structure is as follows:
3. Methodology

3.1 Application Development (Coding)

In this study, the program of the application is made sequentially and regularly according to logic, where the coding of the application program uses the python programming language and the Graphic User Interface (GUI) is made using Tk.Inter which is a package that has been built in Python. The following are the stages of the application of the 2D frame structure analysis program:

- Pre-Processing

The pre-processing stage is part of the program which consists of a series of input processes and reads the data which is then stored into the computer's memory (data structure) which is needed for analysis. The process of entering data at this pre-processing stage is grouped in a single frame named “Input Parameter”, where in this parameter input frame, columns (entry box) have been provided to enter all data from the frame structure.

\[
[k] = \frac{E}{L} \begin{bmatrix}
AC^2 + \frac{12L}{S^2} & (A - \frac{12L}{S^2})CS & -\frac{6L}{C} & AC^2 + \frac{12L}{S^2} & -\frac{6L}{C} \\
4L & \frac{6L}{C} & -\frac{6L}{C} & (A - \frac{12L}{S^2})CS & \frac{6L}{C} \\
AC^2 + \frac{12L}{S^2} & (A - \frac{12L}{S^2})CS & -\frac{6L}{C} & 2L & -\frac{6L}{C} \\
\end{bmatrix}
\]

Symmetry

\[
\text{Symmetry}
\]

At this stage the data point coordinates of the nodal data elements, the foundation of data, the data load will be visualized using a package Tk.Inter, so that the shape of the structure of the frame can be seen on the screen. The visualization of this structure corresponds to the data.
entered by the application user. Visualization of the structure in this application looks like Figure 4.

![Structure Visualization](image)

**Figure 4. Structure Visualization**

- **Processing**
  The inputted data stored in computer memory (data structure) is processed in a numerical solution. Starting from the formation of the element stiffness matrix, then the formed element stiffness matrix is arranged in a global position so that a global stiffness matrix is obtained. Next form the force matrix. After the matrix is formed, it is solved by Hooke's law. Based on the matrix operation, the displacement matrix is equal to the inverse of the global stiffness matrix times the force matrix, so that the result of the displacement matrix is obtained. After obtaining the displacement matrix, the support reaction can also be calculated.

- **Post-Processing**
  At this stage, the value of the displacement of each node, the support reaction of each structural element has been obtained. All the results of this analysis process are shown in Figure 5.

![Simulation Results](image)

**Figure 5. Simulation Results**
3.2 Model Frame
The accuracy of the 2D frame analysis application developed in this study will be compared with manual calculations, analytical calculations and also with previous research. The structural model being tested is the rod frame.

![Figure 6. Frame Structural](image)

| Table 1. Frame Structural Properties |
|--------------------------------------|
| Data                                | Value                |
| Cross-sectional area (A)            | 0.04 m²              |
| Modulus of elasticity (E)           | 70 GPa               |
| Inertia (I)                         | 0.0002 m⁴            |
| Total number of node                | 3                    |
| Total number of element             | 2                    |
| Length of element                   | 4 m                  |
| Total number of element             | 2                    |

3.3 Simulation Application Design
In this developed application, it is necessary to carry out regulatory steps, including:

1. Material Data
   The data entered on the data presented material on Table 2.

   | Table 2. Material Data       |
   |-------------------------------|
   | Data                        | Nilai              |
   | Cross-sectional area (A)    | 0.04 m²            |
   | Modulus of elasticity (E)   | 70 GPa             |
   | Inertia (I)                 | 0.0002 m⁴         |

The display of material data input will be like in Figure 7.

![Figure 7. Material Data Input Display](image)
2. Geometry Data
The frame geometry data is a display that is used to input the geometry of the frame such as the number of elements, the number of nodes, the x and y coordinates of each node. The data is entered on the geometry data are presented as Table 3.

| Data            | Value | X(m) | Y(m) | Start Node Number | End Node Number |
|-----------------|-------|------|------|-------------------|-----------------|
| Total Element   | 2     | -    | -    | -                 | -               |
| Total Nodal     | 3     | -    | -    | -                 | -               |
| Node 1          | -     | 0    | 0    | -                 | -               |
| Node 2          | -     | 0    | 4    | -                 | -               |
| Node 3          | -     | 4    | 4    | -                 | -               |
| Element 1       | -     | -    | -    | 1                 | 2               |
| Element 2       | -     | -    | -    | 2                 | 3               |

Display data input geometry would like the Figure 8.

![Figure 8. Geometry Data Input Display](image)

3. Support Data
The data entered on the support data is presented as Table 4.

| Node (Support) | Support Type |
|----------------|--------------|
| 1              | Engsel       |
| 3              | Rol(y)       |
Display data input support would like the Figure 9.

![Figure 9. Support Data Input Display](image)

4. Load Data
The data entered on the data load is presented as Table 5.

**Table 5. Load Data**

| Titik Nodal (Beban) | Beban Vertikal (kN) | Beban Horizontal (kN) | Momen (kN.m) |
|---------------------|---------------------|-----------------------|--------------|
| 2                   | -20                 | 0                     | 0            |

Display data input support would like the Figure 10.

![Figure 10. Load Data Input Display](image)

4. Result and discussion

4.1 *Structure Display Result Windows*
After entering the data, visualization of tested structure will be displayed in the “Graphic” Frame as shown in figure 11 below.
4.2 Result in table

The result of the analysis is a display that functions to display *output listings* such as the displacement of each node, the support reaction. Display the result will be like the Figure 12 below.

![Figure 12. Display of The Result in The Form of a Table](image)

4.3 Manual solution calculation

The strength of the Model Frame is calculated using the finite element method manual solution so that the structural strength is obtained as in tables 6 and 7.
Table 6. Result of Manual Solution Nodal Displacement

| Nodal | Displacement x-axis (m) | y-axis (m) | Ø (rad) |
|-------|------------------------|-----------|--------|
| 1     | 0                      | 0         | 0,019055 |
| 2     | -0,060981              | -0,000029 | 0,007626 |
| 3     | -0,060981              | 0         | -0,003802 |

Table 7. Result of Manual Solution Support Reaction

| Nodal | Support Reaction Fx(kN) | Fy(kN) | M(kN.m) |
|-------|-------------------------|-------|---------|
| 1     | 19,999                  | 20,3  | 0       |
| 3     | 0                       | -19,999 | 0     |

4.4 Result of The Software Simulation
The application simulation result of the analyzed frame structure are shown in table 8 and 9.

Table 8. Result of Application Simulation Nodal Displacement

| Nodal | Displacement x-axis (m) | y-axis (m) | Ø (rad) |
|-------|------------------------|-----------|--------|
| 1     | 0                      | 0         | 0,0190548 |
| 2     | -0,060981              | -0,00002857 | 0,0076261 |
| 3     | -0,060981              | 0         | -0,0038023 |

Table 9. Result of Application Simulation Support Reaction

| Nodal | Support Reaction Fx(kN) | Fy(kN) | M(kN.m) |
|-------|-------------------------|-------|---------|
| 1     | 20                      | 20    | 0       |
| 3     | 0                       | -20   | 0       |

Figure 13. Comparison of Displacement Result Manual Solution and Developed Application
After the results of manual solution calculations, the results will be compared with the results of the design application itself to determine the accuracy of the design application, and graphs such as Figures 13 and 14 are obtained below.

![Comparison of Support Reaction Result Manual Solution and Developped Application](image)

**Figure 14.** Comparison of Support Reaction Result Manual Solution and Developped Application

From all the comparison charts above, it can be seen that the percentage difference does not exceed 1%. The largest percentage in the comparison of the results of the support reaction Solution and the design application, there is a difference of 0.015 % in the moment force at the node 1. This difference in results is due to the difference in the precision of the decimal digits used.

From these results, it can be seen that the design application has a value that is close to the truth. Nevertheless, this application still has many shortcomings and little difficulty in using this application, so it needs further development for this application in order to obtain more precise results and more easily used by the user in the future.

5. **Conclusion**

The application that has been made has results that are close to the truth, which can be seen from the comparison of the results of analytical calculations, manual calculations and comparisons to previous research. Where the results of the simulation are close to the results of the manual solution, with the percentage difference not exceeding 1%.

**References**

[1] Hakim R H 2017 Pembuatan Program analisis struktur metode matriks kekakuan untuk frame 2D dan Frame 3D. Bandung: Universitas Pendidikan Indonesia

[2] Isworo H and Ansyah P R 2018 Metode Elemen Hingga. Banjarmasin: Universitas Lambung Mangkurat

[3] Levy M and Salvadori M 1992 Why Buildings Falldown : How Structure Fail. London: W.W. Norton and Company

[4] Logan D 2011 A First Course In The Finite Elemen Method Fourth Edition. New Year : Cengage Learning

[5] Raharjo F A 2020 Mahir Solidworks Simulation CAE Seberapa Amankah. Yogyakarta: Deepublish

[6] Supartono F and Boen T 1980 Analisa Struktur Dengan Metoda Matriks. Jakarta: Universitas Indonesia
[7] Widodo S 2007 Mekanika Kekuatan IV (Metode Matrix Kekakuan). Yogyakarta: Universitas Negri Yogyakarta
[8] Zulfikar 2019 Dasar-Dasar Metode Elemen Hingga Untuk Teknik Mesin. Medan: Universitas Medan Area