Session 3649

A Simple Problem Which Students Can Solve and Check Using an Inexpensive Calculator

Patrick J. Cronin
The Pennsylvania State University
New Kensington Campus

Abstract

This paper proposes a simple engineering structural analysis problem which can be used to introduce lower division engineering or engineering technology students to the fundamentals of the finite element analysis (FEA) method. Step by step the student sets up the matrix equation which represents the system of simultaneous linear equations which is necessary to solve for the unknown displacements at each of the nodes. They then solve this system of equations using a numerical method which is efficient for large numbers of equations. All of this they do with an inexpensive scientific calculator. As the final step they calculate the stress in each structural member.

1. Introduction

Finite element analysis (FEA) software can produce color stress contour plots representing the stress at thousands of points within a machine part with dozens of forces applied. A student studying stress analysis for the first time can beneficially be exposed to the terminology and progression of calculations used to calculate the stresses in the elements of an FEA model. Terms involved in FEA analysis include: nodes, local and global stiffness matrices, local and global coordinate systems, forward reduction, and backward substitution. A model which I have used is a truss composed of three strut elements which form a triangle. Each strut has a simple 4 x 4 local element stiffness matrix [1]. The local x direction of each strut makes a non-zero angle with the global X axis. Therefore, the local stiffness matrix of each strut must be transformed into a global element stiffness matrix. The transformation is done using transformation matrices based upon the angle between the local and global coordinate systems. All of these calculations can be done using an inexpensive scientific calculator. The required data consists of: (1) the number of nodes and elements, (2) the (x,y) coordinates of each node, and (3) node i, node j, area, and elastic modulus of each element. The student calculates: (1) the local and global stiffness matrices of each element, (2) the global stiffness matrix of the structure, (3) the displacement of each node in the X and Y directions using the applied forces and boundary conditions, and (4) the stresses in each element.

Some additional points are the following ones. Stress contours are not shown on truss element FEA models. Often, small squares of red, blue, etc. colors (red representing the highest axial stress) are shown in the middle of each truss element. Another point is that I indicate to the students that the calculations they will perform would also be done by a computer performing an FEA analysis, but a much larger set of finite elements, forces, displacements, and stresses would be involved. One of the educational objectives is to give the students a good understanding of solving an engineering stress analysis problem which must be accomplished by setting up and solving a large set of simultaneous linear equations (usually performed by a computer). The calculations shown in this paper could be used as the basis of programming this type of FEA analysis.

2. Description of the Problem

An elevation view of the structure for which the stress analysis is to be performed is shown in Figure 1. It is a planar truss structure. The joint constraints are shown in Figure 1, as are the forces applied at the joints.

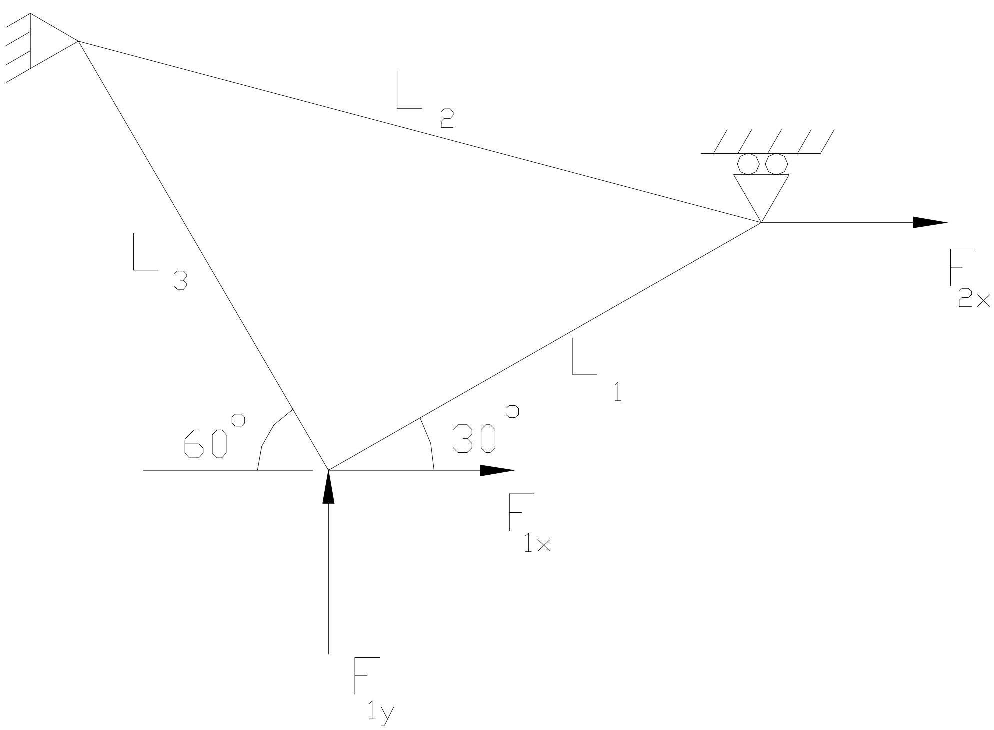

Figure 1. The structure for which a stress analysis is required.

Each of the three truss members is made of steel. The modulus of elasticity value which will be used for the steel is 30 x 10^6 lb/in^2. The data for the three joints and members is given in Table 1.

Table 1. Data for the structure joints and members.

Joint	F_x, lb	F_y, lb
1	1,000	2,000
2	1,000	0
3	0	0

Member	Length, in	Cross-section Area, in^2
1	120	3.0
2	169.7	4.0
3	120	5.0

The geometry of the structure was chosen such that each of the truss members has a different angle of inclination with the horizontal lines.

.

The goal of the exercise is to calculate the tensile or compressive stress in each of the three truss members using an FEA model of the truss with the given applied forces and joint constraints. A scientific calculator is to be used. No stored algorithms such as simultaneous equation solvers are to be used.

3. Solution of the Problem

The finite element model of the truss is shown in Figure 2. The global coordinate system for the structure is also shown in the figure. The node number at which each element begins and ends is shown in Table 2. The truss element itself is depicted in Figure 3. The angle of inclination of the element with the global X axis is angle θ. The value of this angle can be between -180^0 and 180^0.

Table 2. The node at which each element begins and ends.

Element	Node i	Node j
1	1	2
2	2	3
3	1	3

Each of the element stiffness matrices can be written as

$$[K] = \begin{bmatrix} [K'] & -[K'] \\ -[K'] & [K'] \end{bmatrix}$$

where

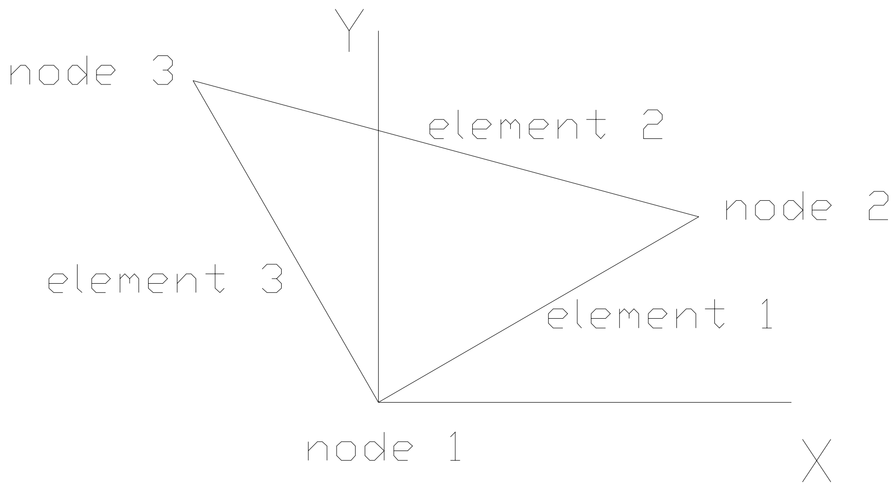

Figure 2. The FEA model of the structure to be analyzed.

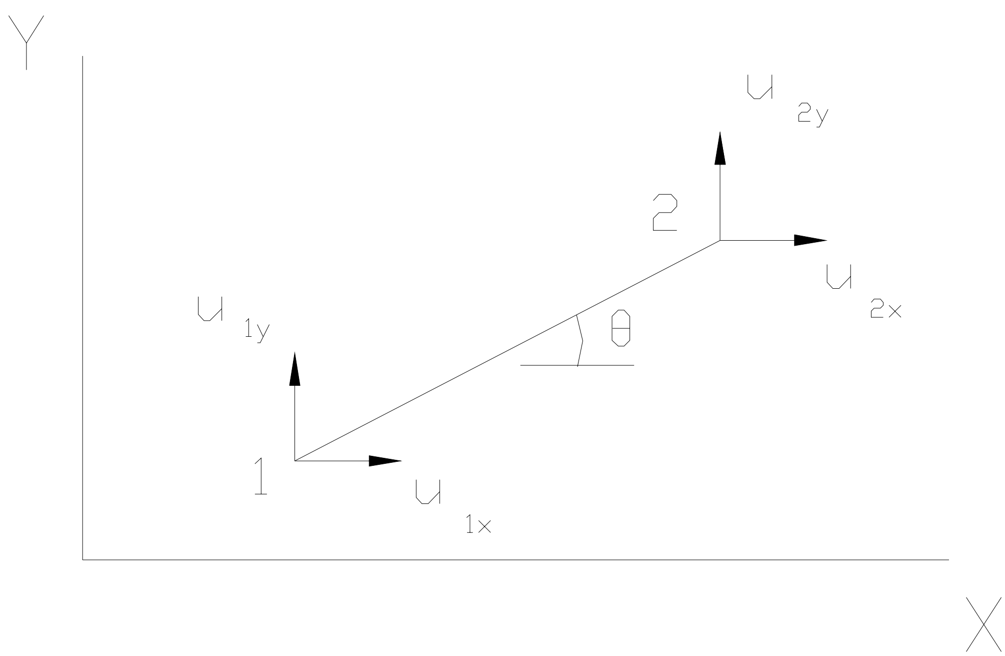

Figure 3. The pin-jointed bar (truss) element.

$$[K'] = \left(\frac{AE}{L}\right)\begin{bmatrix}\lambda^2 & \lambda\mu \\ \lambda\mu & \mu^2\end{bmatrix}$$

and $\lambda = \cos\theta$ and $\mu = \sin\theta$.

For element number 1

$$\theta = 30^o$$

and

$$[K'] = \begin{bmatrix}5.63E5 & 3.25E5 \\ 3.25E5 & 1.88E5\end{bmatrix}$$

The units of the K′ matrix numbers are lb/in.

For element number 2,

$$\theta = 165^o$$

and

$$[K'] = \begin{bmatrix}6.60E5 & -1.77E5 \\ -1.77E5 & 4.74E4\end{bmatrix}$$

For element number 3,

$$\theta = 120^o$$

and

$$[K'] = \begin{bmatrix}3.13E5 & -5.41E5 \\ -5.41E5 & 9.38E5\end{bmatrix}$$

The assembled structure stiffness matrix becomes

$$[K_S] = \begin{bmatrix}[K_{11}] & [K_{12}] \\ [K_{21}] & [K_{22}]\end{bmatrix}$$

where

$$[K_{11}] = \begin{bmatrix} 8.76E5 & -2.16E5 & -5.63E5 \\ -2.16E5 & 1.13E6 & -3.25E5 \\ -5.63E5 & -3.25E5 & 12.23E5 \end{bmatrix}$$

$[K_{12}]$, $[K_{21}]$, and $[K_{22}]$ are similar looking 3 x 3 matrices but will not be given here since they are not needed to solve for the desired displacements at nodes 1 and 2.

The matrix equation for the entire structure is

$$[K_S] \ \{u\} = \{F\}$$

where

$$\{u\} = [u_{1x} \ \ u_{1y} \ \ u_{2x} \ \ 0 \ \ 0 \ \ 0]^T$$

$$\{F\} = [F_{1x} \ \ F_{1y} \ \ F_{2x} \ \ F_{2y} \ \ F_{3x} \ \ F_{3y}]^T$$

Forces F_{1x}, F_{1y}, and F_{2x} are the applied forces of 1,000 lb, 2,000 lb, and 1,000 lb, respectively. Forces F_{2y}, F_{3x}, and F_{3y} are the three unknown reaction forces. Choleski's method [2] can be used to solve for the unknown displacements u_{1x}, u_{1y}, and u_{2x}.

That matrix equation is

$$[K] \ \{u\} = \{F\}$$

where

$$[K] = [K_{11}]$$

$$\{u\} = [u_{1x} \ u_{1y} \ u_{2x}]^T$$

$$\{F\} = [F_{1x} \ F_{1y} \ F_{2x}]^T$$

Choleski's method is based upon

$$[K] = [L] \ [L]^T$$

where [L] is a lower triangular matrix. The elements of the lower triangular matrix can be calculated as

$$l_{11} = k_{11}^{1/2} = 9.36E2$$

$$l_{21} = k_{21} \ / \ l_{11} = -2.31E2$$

$$l_{22} = (k_{22} - l_{21}^2)^{1/2} = 1.04E3$$

$$l_{31} = k_{31} / l_{11} = -6.01E2$$

$$l_{32} = (k_{32} - l_{21}\, l_{31}) / l_{22} = 4.96E2$$

$$l_{33} = (k_{33} - l_{31}^2 - l_{32}^2)^{1/2} = 7.85E2$$

The equation for forward reduction is

$$Lv = F$$

so

$$\begin{bmatrix} 9.36E2 & 0 & 0 \\ -2.31E2 & 1.04E3 & 0 \\ -6.01E2 & -4.96E2 & 7.85E2 \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} 1000 \\ 2000 \\ 3000 \end{bmatrix}$$

The solution to this matrix equation is relatively easily obtained as

$$v_1 = 1.07, \; v_2 = 2.16, \; v_3 = 3.46$$

The equation for backward substitution is

$$L^T u = v$$

so

$$\begin{bmatrix} 9.36E2 & -2.31E2 & -6.01E2 \\ 0 & 1.04E3 & -4.96E2 \\ 0 & 0 & 7.85E2 \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \end{bmatrix} = \begin{bmatrix} 1.07 \\ 2.16 \\ 3.46 \end{bmatrix}$$

The solution to this matrix equation can be obtained giving

$$u_1 = 5.01E{-3}, \; u_2 = 4.18E{-3}, \; u_3 = 4.41E{-3}$$

Therefore, the displacements at nodes 1 and 2 are

$$u_{1x} = 5.01E\text{-}3 \text{ inches}, \qquad u_{1y} = 4.18E\text{-}3 \text{ inches}$$

$$u_{2x} = 4.41E\text{-}3 \text{ inches}$$

The equation for the axial stress can be found [2] to be

$$\sigma_x = (E/L) \, [\, -\lambda \;\; -\mu \;\; \lambda \;\; \mu \,] \; [u_{1x} \;\; u_{1y} \;\; u_{2x} \;\; u_{2y}]^T$$

for the element between nodes 1 and 2. Therefore, the stress is calculated as

$$\sigma_x = \frac{30E6}{120} \begin{bmatrix} -0.866 & -.5 & .866 & .5 \end{bmatrix} \begin{bmatrix} 5.01\mathrm{E}-3 \\ 4.18\mathrm{E}-3 \\ 4.41\mathrm{E}-3 \\ 0 \end{bmatrix}$$

$$\sigma_x = -652 \text{ lb/in}^2$$

Using the same approach for elements 2 and 3 leads to

$$\sigma_x = 753 \text{ lb/in}^2 \quad \text{(element 2)}$$

$$\sigma_x = -279 \text{ lb/in}^2 \quad \text{(element 3)}$$

More exact stress values could be obtained by using a larger number of significant digits in the calculations.

4. Summary

A simple three element FEA problem has been shown. It is a problem which students can solve without investing an overly burdensome amount of time and effort. It includes FEA procedures similar the the ones which would be performed by a high speed computer solving a large FEA problem. It gives the student an insight into what happens to the large amount of input data which causes it to become the colored contour stress plots which are now the standard FEA stress result presentations.

References

1. Martin, H.C. and Carey, G.F., *Introduction To Finite Element Analysis*, McGraw-Hill,1973.
2. Przemieniecki, J.S., *Theory of Matrix Structural Analysis,* McGraw-Hill, 1968.

PATRICK J. CRONIN
Patrick Cronin is currently an Assistant Professor of engineering at The Pennsylvania State University New Kensington Campus. He holds a B.S. in Engineering Science and an M.S. degree in Civil Engineering, both from the State University of New York at Buffalo. He previously worked as an engineer performing stress and vibration analysis of various types of mechanical equipment.