Teaching Finite Element Analysis as a Solution Method for Truss Problems in Statics

Jiaxin Zhao
Indiana University–Purdue University Fort Wayne

Abstract

Finite Element Analysis (FEA) is a very powerful tool that is used in virtually every area in the field of Mechanical Engineering and many other disciplines. It is beneficial for the mechanical engineering students to have exposure to this tool as early as possible and as frequently as possible in their engineering education. The earliest time comes when they are taught the truss problems in Statics. The FEA can be introduced as a solution method for the truss problems in Statics, without the need to involve additional knowledge of the deformation theory. This paper presents the step by step FEA formulation of the equations to solve for the forces inside the truss members and the reaction forces at the fixed and sliding joints, and compares the FEA formulation with the conventional approach closely. The students will clearly see the methodology and the essential features of FEA. Only truss member internal forces are involved in the elemental formulation and thus there is no need to introduce deformation theory. A MATLAB program is written to implement this FEA solution method. The property of the final assembled coefficient matrix, with the boundary conditions applied, determines the problem at hand to be statically determinate or statically indeterminate. In the former case, the students can easily solve for the joint forces using matrix manipulation just as in the conventional solution method. In the later case, the students are then encouraged to revisit this problem after they learn the deformation theory in the mechanics of materials course. The students are reminded that, with the introduced FEA formulation procedure deep in mind, the only major difference would be in the elemental formulation to apply FEA in mechanics of materials as well as in heat transfer, fluid mechanics, etc. FEA can then be reintroduced in these courses to strengthen the students' understanding of the basic FEA procedure and commercial FEA software can also be involved.

Introduction

Finite Element Analysis (FEA) is a very powerful tool that is used in virtually any area in the field of Mechanical Engineering and many other disciplines. Many institutions have an FEA course as a technical elective in senior level [1, 2]. However, it is beneficial for the mechanical engineering students to have exposure to this tool as frequently as possible in their engineering education [3], and as early as possible. Many educators introduce FEA in lower level mechanical engineering courses, most likely in Mechanics of Materials [4, 5, 6].

FEA can be introduced to students at an even earlier point in the curriculum, i.e. Statics. The conventional deformation based FEA analysis of truss problems can be taught by introducing first the deformation theory [7], which usually appears in the Mechanics of Materials course. However, this extra burden of covering the deformation theory in order to introduce FEA in Statics is not necessary. This paper describes the member force based FEA analysis of plane truss problems that can be introduced to the students as a solution method for the truss problems without involving the additional knowledge of deformation theory.

The Simple Plane Truss Problem - Statically Determinate

A simple plane truss problem is statically determinate. A general layout of the simplest configuration of such a plane truss problem is shown in Figure 1(a) with three truss members, one fixed joint and one sliding joint, which is sliding on the plane of angle $\theta_{sliding}$ to the x axis. This setup is used as an example in deriving the equations for both the conventional solution and the FEA solution. Each joint is numbered consecutively starting from 1. The order assigned to number the joints does not play a factor in the analysis as long as all the joint numbers are consecutive starting from 1. Similarly, each truss member is also numbered consecutively starting from 1 and the number is denoted with a circle around it to distinguish it from the joint number. The conventional assumptions of smooth pin joints and loads applied only at joints are followed. As a result, each truss member is a two-force member as shown in Figure 1(b). As used in this Figure and throughout this paper, the subscript of a variable denotes the joint number while the superscript denotes the truss member number. The sign convention used here is that tension is positive. Thus a positive truss member force F^e means a tensional force in truss member e while a negative F^e means a compressional force. The force acting on the sliding joint $R_{sliding}$ is positive pointing toward the sliding plane as shown in Figure 1(a). The two joints of the truss member e are denoted as i^e and j^e. The angle of the truss member is then defined as the angle from the positive x direction to the direction of $\vec{i^e j^e}$. The forces acting on the truss member e at the i^e and j^e joint locations are then denoted as $\vec{F}^e_{i^e}$ and $\vec{F}^e_{j^e}$, respectively, as shown in Figure 1(b). Consequently, reaction forces acting on the joints by the truss member e are denoted by $-\vec{F}^e_{i^e}$ and $-\vec{F}^e_{j^e}$ as shown in Figure 1(a).

The traditional solution methods are Method of Joints and Method of Sections, as typically taught in the Statics Course. The students will master these two methods first. However, these two methods will become tedious if the number of truss members or joints becomes large. It is very useful to introduce a systematic approach to solve this truss problem.

The Linear Algebra Formulation

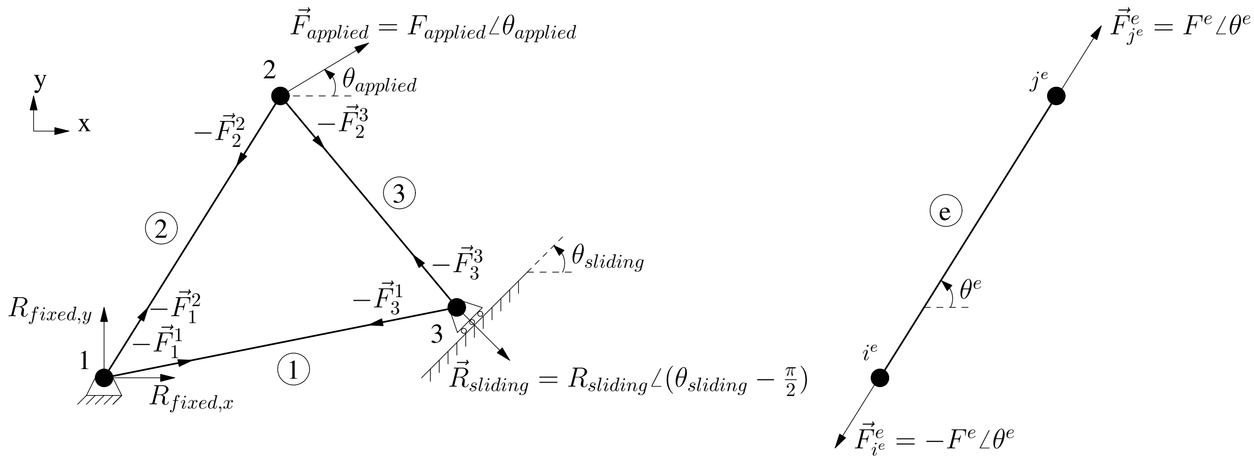
A conventional way of solving this problem systematically is to gather all the equations and solve them using linear algebra. The force balance at all the joints gives:

Joint 1:

$$\begin{aligned} F^1 \cos\theta^1 + F^2 \cos\theta^2 + R_{fixed,x} &= 0 \\ F^1 \sin\theta^1 + F^2 \sin\theta^2 + R_{fixed,y} &= 0 \end{aligned}$$

Joint 2:

$$\begin{aligned} -F^2 \cos\theta^2 + F^3 \cos\theta^3 + F_{applied} \cos\theta_{applied} &= 0 \\ -F^2 \sin\theta^2 + F^3 \sin\theta^3 + F_{applied} \sin\theta_{applied} &= 0 \end{aligned}$$



(a) (b)

Figure 1: (a). A simple truss structure layout and forces on the joints/nodes (b). Notation and forces on a truss member/element

Joint 3:

$$\begin{aligned} -F^3\cos\theta^3 - F^1\cos\theta^1 + R_{sliding}\cos(\theta_{sliding} - \tfrac{\pi}{2}) &= 0 \\ -F^3\sin\theta^3 - F^1\sin\theta^1 + R_{sliding}\sin(\theta_{sliding} - \tfrac{\pi}{2}) &= 0 \end{aligned}$$

We have 6 equations for this problem and also 6 unknowns to solve for: three truss member forces F^1, F^2, F^3, two fixed joint reaction forces $R_{fixed,x}$, $R_{fixed,y}$ and one sliding joint reaction force $R_{sliding}$, which is normal to the sliding plane. Moving the known quantities to the right hand side of the equations and putting the equations into matrix form yields:

$$\underbrace{\begin{bmatrix} \cos\theta^1 & \cos\theta^2 & 0 & 1 & 0 & 0 \\ \sin\theta^1 & \sin\theta^2 & 0 & 0 & 1 & 0 \\ 0 & -\cos\theta^2 & \cos\theta^3 & 0 & 0 & 0 \\ 0 & -\sin\theta^2 & \sin\theta^3 & 0 & 0 & 0 \\ -\cos\theta^1 & 0 & -\cos\theta^3 & 0 & 0 & \cos(\theta_{sliding} - \frac{\pi}{2}) \\ -\sin\theta^1 & 0 & -\sin\theta^3 & 0 & 0 & \sin(\theta_{sliding} - \frac{\pi}{2}) \end{bmatrix}}_{=[K]} \underbrace{\begin{Bmatrix} F^1 \\ F^2 \\ F^3 \\ R_{fixed,x} \\ R_{fixed,y} \\ R_{sliding} \end{Bmatrix}}_{=\{FR\}} = - \underbrace{\begin{Bmatrix} 0 \\ 0 \\ F_{applied}\cos\theta_{applied} \\ F_{applied}\sin\theta_{applied} \\ 0 \\ 0 \end{Bmatrix}}_{=\{FA\}} \tag{1}$$

where as $[K]$ is the coefficient matrix, $\{FR\}$ is the unknown reaction force vector and $\{FA\}$ is the applied force vector from this simple linear algebra formulation. The solution is then simply:

$$\{FR\} = -[K]^{-1}\{FA\} \tag{2}$$

When a simple plane truss problem has more members and joints, two force balance equations are written for each joint, and then all the equations are assembled into the matrix form. Again, as the numbers of members and joints increase, it is troublesome to get this matrix form. This is when, the Finite Element Analysis, as a numerical method, can be used to efficiently and automatically generate this matrix form to solve the problem.

The FEA Formulation

Now, we follow the conventions of FEA to name the truss members as elements and the joints as nodes in this analysis, and the names are interchangeable from here on in this paper. Only the internal forces in the truss members/elements and the reaction forces at the joints/nodes are of concern. For each element, the force inside the element F^e contributes to the load on the joints as:

$$\{F\}^e = \left\{ \begin{array}{c} -F^e_{i_e,x} \\ -F^e_{i_e,y} \\ -F^e_{j_e,x} \\ -F^e_{j_e,y} \end{array} \right\} = \left\{ \begin{array}{c} F^e \cos\theta^e \\ F^e \sin\theta^e \\ -F^e \cos\theta^e \\ -F^e \sin\theta^e \end{array} \right\} = \left\{ \begin{array}{c} \cos\theta^e \\ \sin\theta^e \\ -\cos\theta^e \\ -\sin\theta^e \end{array} \right\} F^e \tag{3}$$

Here $\{F\}^e$ is the elemental force vector that is acting on the i^e and j^e nodes of the element e. Note the first two components are acting on the joint i^e (the i joint of the truss member e) while the last two components are acting on the joint j^e.

On each joint i, we have the force balance of:

$$\sum_e (-\vec{F}^e_{i^e})_{i=i^e} + \sum_e (-\vec{F}^e_{j^e})_{i=j^e} + \vec{R}_{i,fixed} + \vec{R}_{i,sliding} + \vec{F}_{i,applied} = 0 \tag{4}$$

The first two terms only exist if the i node (i^e) or the j node (j^e) of the element e is the current node of interest i, respectively. The last three terms exist if the current node of interest i is a fixed joint, a sliding joint, or a joint with external force(s), respectively. Applying this vector equation on all N_{node} nodes will give us a total of $N_{eq} = 2N_{node}$ independent equations.

To assemble all the equations into matrix form, we first extend the elemental force vector $\{F\}^e$ of element e (on nodes i^e and j^e) in Equation 3 to the global force vector $\{F\}^e_G$ including force components on all the nodes (1,2,3) in the problem. For element 1,

$$\{F\}^1_G = \left\{ \begin{array}{c} \cos\theta^1 \\ \sin\theta^1 \\ 0 \\ 0 \\ -\cos\theta^1 \\ -\sin\theta^1 \end{array} \right\} F^1 \qquad \begin{array}{l} -\ node\ 1,\ x\ component \\ -\ node\ 1,\ y\ component \\ -\ node\ 2,\ x\ component \\ -\ node\ 2,\ y\ component \\ -\ node\ 3,\ x\ component \\ -\ node\ 3,\ y\ component \end{array} \tag{5}$$

Note that the positions corresponding to forces on node 2 are padded with zeros as element 1 is on nodes 1 (i node) and 3 (j node). Similarly, we have for elements 2 and 3:

$$\{F\}^2_G = \left\{ \begin{array}{c} \cos\theta^2 \\ \sin\theta^2 \\ -\cos\theta^2 \\ -\sin\theta^2 \\ 0 \\ 0 \end{array} \right\} F^2 \qquad \{F\}^3_G = \left\{ \begin{array}{c} 0 \\ 0 \\ \cos\theta^3 \\ \sin\theta^3 \\ -\cos\theta^3 \\ -\sin\theta^3 \end{array} \right\} F^3 \tag{6}$$

The global reaction force vector for fixed joint(s) can also be written including the force components on all the nodes as:

$$\{R_{fixed}\}_G = \left\{ \begin{array}{c} R_{fixed,x} \\ R_{fixed,y} \\ 0 \\ 0 \\ 0 \\ 0 \end{array} \right\} = \left\{ \begin{array}{c} 1 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{array} \right\} R_{fixed,x} + \left\{ \begin{array}{c} 0 \\ 1 \\ 0 \\ 0 \\ 0 \\ 0 \end{array} \right\} R_{fixed,y} \tag{7}$$

where as node 1 is the fixed node and the positions corresponding to force on nodes 2 and 3 are padded with zeros. Similarly, the global reaction force vector for sliding joint(s) can be written as:

$$\{R_{sliding}\}_G = \left\{ \begin{array}{c} 0 \\ 0 \\ 0 \\ 0 \\ R_{sliding}\cos(\theta_{sliding} - \frac{\pi}{2}) \\ R_{sliding}\sin(\theta_{sliding} - \frac{\pi}{2}) \end{array} \right\} = \left\{ \begin{array}{c} 0 \\ 0 \\ 0 \\ 0 \\ \cos(\theta_{sliding} - \frac{\pi}{2}) \\ \sin(\theta_{sliding} - \frac{\pi}{2}) \end{array} \right\} R_{sliding} \tag{8}$$

as node 3 is the sliding node.

The global applied force vector can be written as:

$$\{F_{applied}\}_G = \left\{ \begin{array}{c} 0 \\ 0 \\ F_{applied}\cos\theta_{applied} \\ F_{applied}\sin\theta_{applied} \\ 0 \\ 0 \end{array} \right\} = \left\{ \begin{array}{c} 0 \\ 0 \\ \cos\theta_{applied} \\ \sin\theta_{applied} \\ 0 \\ 0 \end{array} \right\} F_{applied} \tag{9}$$

as the external force is applied at node 2.

Applying the above global force vectors to Equation (4), we have:

$$\{F\}_G^1 + \{F\}_G^2 + \{F\}_G^3 + \{R_{fixed}\}_G + \{R_{sliding}\}_G + \{F_{applied}\}_G = 0 \tag{10}$$

or:

$$\left\{ \begin{array}{c} \cos\theta^1 \\ \sin\theta^1 \\ 0 \\ 0 \\ -\cos\theta^1 \\ -\sin\theta^1 \end{array} \right\} F^1 + \left\{ \begin{array}{c} \cos\theta^2 \\ \sin\theta^2 \\ -\cos\theta^2 \\ -\sin\theta^2 \\ 0 \\ 0 \end{array} \right\} F^2 + \left\{ \begin{array}{c} 0 \\ 0 \\ \cos\theta^3 \\ \sin\theta^3 \\ -\cos\theta^3 \\ -\sin\theta^3 \end{array} \right\} F^3 + \left\{ \begin{array}{c} 1 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{array} \right\} R_{fixed,x} + \left\{ \begin{array}{c} 0 \\ 1 \\ 0 \\ 0 \\ 0 \\ 0 \end{array} \right\} R_{fixed,y}$$

$$
+\begin{Bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ \cos(\theta_{sliding} - \frac{\pi}{2}) \\ \sin(\theta_{sliding} - \frac{\pi}{2}) \end{Bmatrix} R_{sliding} + \begin{Bmatrix} 0 \\ 0 \\ \cos\theta_{applied} \\ \sin\theta_{applied} \\ 0 \\ 0 \end{Bmatrix} F_{applied} = 0 \tag{11}
$$

Simplifying and moving the known applied force vector to the right side of the equation, we have:

$$
\underbrace{\begin{bmatrix} \cos\theta^1 & \cos\theta^2 & 0 & 1 & 0 & 0 \\ \sin\theta^1 & \sin\theta^2 & 0 & 0 & 1 & 0 \\ 0 & -\cos\theta^2 & \cos\theta^3 & 0 & 0 & 0 \\ 0 & -\sin\theta^2 & \sin\theta^3 & 0 & 0 & 0 \\ -\cos\theta^1 & 0 & -\cos\theta^3 & 0 & 0 & \cos(\theta_{sliding} - \frac{\pi}{2}) \\ -\sin\theta^1 & 0 & -\sin\theta^3 & 0 & 0 & \sin(\theta_{sliding} - \frac{\pi}{2}) \end{bmatrix}}_{\{F\}^1_G \quad \{F\}^2_G \quad \{F\}^3_G \quad \{R_{fixed}\}_G \quad \{R_{sliding}\}_G} \underbrace{\begin{Bmatrix} F^1 \\ F^2 \\ F^3 \\ R_{fixed,x} \\ R_{fixed,y} \\ R_{sliding} \end{Bmatrix}}_{\{FR\}} = -\underbrace{\begin{Bmatrix} 0 \\ 0 \\ \cos\theta_{applied} \\ \sin\theta_{applied} \\ 0 \\ 0 \end{Bmatrix}}_{\{F_{applied}\}_G} F_{applied} \tag{12}
$$

which is exactly the same as Equation (1) from the direct linear algebra formulation. A close examination of the Equation (12) yields that by arranging the unknown reaction force vector as:

$$
\{FR\} = \begin{Bmatrix} F^1 \\ F^2 \\ F^3 \\ R_{fixed,x} \\ R_{fixed,y} \\ R_{sliding} \end{Bmatrix} \quad \begin{matrix} \left.\begin{matrix} \\ \\ \\ \end{matrix}\right] N_{elem},\ number\ of\ truss\ members/elements \\ \left.\begin{matrix} \\ \\ \end{matrix}\right] 2N_{fixed},\ twice\ the\ number\ of\ joints/nodes\ (x\ and\ y\ components) \\ ]N_{sliding},\ number\ of\ sliding\ joints/nodes \end{matrix} \tag{13}
$$

the coefficient matrix $[K]$ of the final system of equations has the following properties:

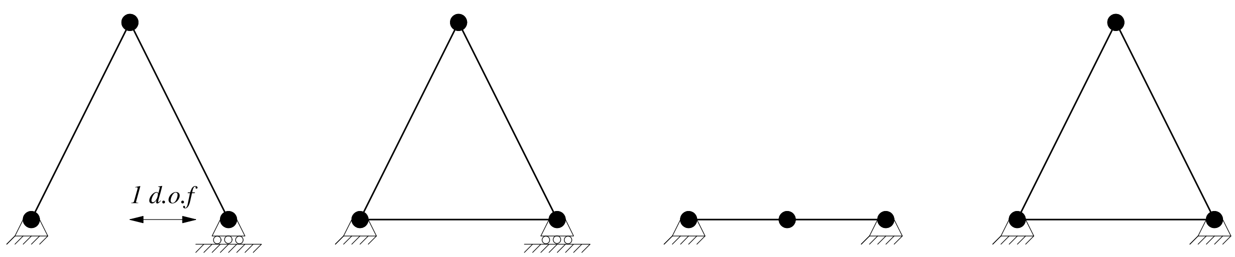
- The first N_{elem} columns correspond to the global force vectors of the N_{elem} elements, without the truss member forces as they are the unknowns.
- The next $2N_{fixed}$ columns correspond to the global force vectors of the N_{fixed} fixed joints, without the x and y reaction force components as they are the unknowns.
- The last $N_{sliding}$ columns correspond to the global force vectors of the sliding joints, without the normal (to the sliding plane) reaction forces as they are the unknowns.

As a result, the coefficient matrix $[K]$ can be determined by simply assembling all the global force vectors. So, the finite element analysis process starts with getting the force vector components due to the elemental forces and put then into the corresponding column positions in the coefficient matrix $[K]$. Then the two different kinds of boundary conditions, namely the fixed and sliding conditions, are applied and the related force components are determined and put in the corresponding column positions in the coefficient matrix $[K]$. Finally the loading information is gathered to determine the global applied force vector $\{FA\}$.

The above FEA formulation can be applied to any truss problem with any number of members or joints. Note that there are $N_{unk} = N_{elem} + 2N_{fixed} + N_{sliding}$ unknowns in the unknown force vector. The number of equations we have is $N_{eq} = 2N_{node}$ due to both the x and the y components of the force balance on each node. N_{eq} and N_{unk} are then the number of rows and the number of columns for the coefficient matrix $[K]$, respectively. Now from the knowledge of linear algebra, we can determine that:

$$
if \begin{cases} N_{eq} & > & N_{unk} & & \\ N_{eq} & = & N_{unk} & \& & |K| \neq 0 \\ N_{eq} & = & N_{unk} & \& & |K| = 0 \\ N_{eq} & < & N_{unk} & & \end{cases} \quad then \begin{cases} No\ Solution & , mechanism\ with\ d.o.f. \\ \{FR\} = -[K]^{-1}\{FA\} & , statically\ determinate \\ No\ Solution & , statically\ indeterminate \\ No\ Solution & , statically\ indeterminate \end{cases}
$$

The final assembled coefficient matrix $[K]$ can then go through the check as described in this equation. To further the students' understanding of the different conditions here, the following figure is shown to them.



$N_{node} = 3$	$N_{node} = 3$	$N_{node} = 3$	$N_{node} = 3$
$N_{elem} = 2$	$N_{elem} = 3$	$N_{elem} = 2$	$N_{elem} = 3$
$N_{fixed} = 1$	$N_{fixed} = 1$	$N_{fixed} = 2$	$N_{fixed} = 2$
$N_{sliding} = 1$	$N_{sliding} = 1$	$N_{sliding} = 0$	$N_{sliding} = 0$
$N_{eq}(6) > N_{unk}(5)$	$N_{eq}(6) = N_{unk}(6)$	$N_{eq}(6) = N_{unk}(6)$	$N_{eq}(6) < N_{unk}(7)$
	$\lvert K\rvert \neq 0$	$\lvert K\rvert = 0$	
1 d.o.f. mechanism	statically determinate	statically indeterminate	statically indeterminate
partial constraint	*proper constraint*	*improper constraint*	*redundant constraint*

Figure 2: Different cases for a general truss problem

A MATLAB program is provided to the students and is attached at the end of this paper. The test case in the provided program is the Example Problem 6.1 in the Statics textbook by Hibbeler [8]. The only part in this program that the students need to modify for different problems is the first part that gathers the FEA model information. The program consists of 100 lines with only 40 lines of true commands. The students are asked to use this program to check their answers to any truss problem they are assigned, whether the problem is solved using the method of joints or the method of sections.

Further Notes About FEA

The students are then introduced to the general FEA procedures, which consists of 7 steps, as related to the current analysis:

1. **Discretization** - We have a naturally discretized system in truss with truss members as elements and joints as nodes.
2. **Interpolation** - The truss member force is used as it is, no need for approximation in this case.
3. **Elemental formulation** - Determine the elemental force vectors for each element as in Equation (3).
4. **Assembly** - Extend the elemental force vectors to global force vectors as in Equations (5) and (6) and put them in corresponding columns of the coefficient matrix $[K]$.
5. **Applying boundary and loading conditions** - Generate the global reaction force vectors as in Equations (7) and (8) and put them in corresponding columns of the coefficient matrix $[K]$. Generate the global applied force vector as in Equation (9).
6. **Solution** - Solve the problem using the matrix manipulation.
7. **Getting other information** - The stress, strain, joint displacement can be determined given the geometry and material properties of the truss members, and of course the deformation theory.

For the statically indeterminate truss problems, the students are reminded that they will revisit truss problems in Mechanics of Materials and those problems can be solved using the deformation theory. After they learned the deformation theory in that course, they will be able to use FEA to solve both the statically determinate and indeterminate truss problems, utilizing the deformation theory in the elemental formulation. They are encouraged to look up the description of such an FEA truss problem formulation in any FEA textbook, if it is not taught in the class. The students are also reminded that, with the introduced FEA formulation procedure deep in mind, FEA can easily be applied to problems in mechanics of materials as well as in heat transfer, fluid mechanics, etc., with the only major difference being in the elemental formulation.

In terms of commercial FEA packages, the students are taught that a typical package consists basically of three parts:

1. **Preprocessor** - Does steps 1 through 5 in the FEA procedure, gathering information about node, element, boundary conditions and loading conditions.
2. **Solver** - Does step 6 in the FEA procedure, the most time consuming stage.
3. **Postprocessor** - Does step 7 in the FEA procedure as well as provide graphic information about the solutions.

Conclusion

In this paper, a basic Finite Element Analysis on statically determinate truss problem is presented, which is well suited to introduce FEA to mechanical engineering students in the Statics course. This introduction is based solely on the topics covered in a general Statics course and does not require the introduction of deformation theory which is typically discussed in the Mechanics of Materials course.

With the exposure to FEA in the Statics course, the students can then be reintroduced to FEA in the subsequent mechanics of materials, heat transfer, fluid mechanics courses as well as many others. Commercial FEA packages can also be introduced in these courses. The students will have a better understanding of FEA in this approach of learning FEA step by step (course by course). This approach is also especially helpful if there is no formal FEA course offered in the curriculum.

This FEA formulation was introduced to the students in the Spring 2004 Statics course. The material was presented in one lecture period immediately after the students were taught the method of joints to solve truss problems. A questionnaire was also handed out to the students for their feedback about the current approach of teaching FEA in Statics. Out of the 11 students responded, all indicated that the material presented here is helpful for them to become familiar with FEA and strengthened their interest in FEA. The FEA program handed out also became a handy tool to check their solutions of truss problems, by simply changing a few lines.

References

[1] Howard, W. E., Musto, J. C., and Prantil, V., "Finite Element Analysis in a Mechanics Course Sequence," Proceedings of the 2001 American Society for Engineering Education Annual Conference & Exposition, Session 2793, 2001.

[2] Lissenden, C. J., Wagle, G. S., and Salamon, N. J., "Applications of Finite Element Analysis for Undergraduates," Proceedings of the 2002 American Society for Engineering Education Annual Conference & Exposition, Session 3568, 2002.

[3] Baker, J. R., Capece, V. R., and Lee, R. J., "Integration of Finite Element Software in Undergraduate Engineering Courses," Proceedings of the 2001 American Society for Engineering Education Annual Conference & Exposition, Session 1520, 2001.

[4] Mueller, D. W., Jr., "Introducing the Finite Element Method to Mechanical Engineering Students Using Matlab," Proceedings of the 2003 American Society for Engineering Education Annual Conference & Exposition, Session 3566, 2003.

[5] Jolley, W. O., Rencis, J. J., and Grandin, H. T., Jr., "A Module for Teaching Fundamentals of Finite Element Theory and Practice Using Elementary Mechanics of Materials," Proceedings of the 2003 American Society for Engineering Education Annual Conference & Exposition, Session 3268, 2003.

[6] Coyle, M. F. and Keel, C. G., "Finite Element Analysis to Second Year Students," Proceedings of the 2001 American Society for Engineering Education Annual Conference & Exposition, Session 2793, 2001.

[7] Pike, M., "Introducing Finite Element Analysis in Statics," Proceedings of the 1997 American Society for Engineering Education Annual Conference & Exposition, Session 2268, 1997.

[8] Hibbeler, R. C., 2004, *Statics, 10th Edition*, Prentice-Hall, Upper Saddle River, NJ.

Biography

JIAXIN ZHAO is an Assistant Professor of Mechanical Engineering at Indiana University-Purdue University Fort Wayne. He received his BS from the University of Science and Technology of China, his MS from the University of Missouri-Rolla, and his PhD from Purdue University-West Lafayette. His research and teaching interests are tribology, machine design, solid mechanics and numerical methods including finite elements and parallel computing.

Appendix - FEA Program of Simple Plane Truss Problem

```
% FEA Program for Plane Truss Problems in Statics
% - Jiaxin Zhao, December, 2003, IPFW

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% Part 1: Input FEA Model Inforcemation %
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% symbol[dimension]: variable
% nnode            : number of nodes (joints)
% nelem            : number of elements (truss members)
% nfixed           : number of fixed joints
% nslide           : number of sliding joints
% nforce           : number of joints with forces applied
%
% xnode[nnode]     : x coordinate of each node
% ynode[nnode]     : y coordinate of each node
% ielem[nelem]     : i node number of each element
% jelem[nelem]     : j node number of each element
% lelem[nelem]     : length of each element
% telem[nelem]     : angle of each element, from x+ to i->j direction, rad
%
% kfixed[nfixed]   : node number of fixed joints
% kslide[nslide]   : node number of sliding joints
% tslide[nslide]   : sliding angle of joints, from x+ to sliding plane, rad
% kforce[nforce]   : node number of joints with forces applied
% fforce[nforce]   : magnitude of the forces applied
% tforce[nforce]   : angle of the forces applied, rad

% input geometry information (Example 6-1, Statics, 10e, R.C.Hibbeler)
nnode=3;
nelem=3;
xnode=[0 0 2];
ynode=[0 2 0];
ielem=[1 1 2];
jelem=[3 2 3];
lelem=sqrt((ynode(jelem)-ynode(ielem)).^2+(xnode(jelem)-xnode(ielem)).^2);
telem=atan2(ynode(jelem)-ynode(ielem), xnode(jelem)-xnode(ielem));

% input boundary conditions
nfixed=1;
kfixed=[1];
nslide=1;          % if nslide=0; then
kslide=[3];        %    kslide=[];
tslide=[0]/180*pi; %    tslide=[];
```

```
% input loading conditions
nforce=1;
kforce=[2];
fforce=[500];
tforce=[0]/180*pi;


%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% Part 2: Build FEA Model and Solve %
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% symbol[dimension]: variable
% nunk              : number of unknowns in the FEA solution
% kg[nunk,nunk]     : coefficient matrix
% xg[nunk]          : unknown vector
% fg[nunk]          : force vector

nunk=nelem+2*nfixed+nslide;
kg=zeros(nunk,nunk);
xg=zeros(nunk,1);
fg=zeros(nunk,1);

% FEA formulations - direct assembly of the elemental force vectors
for e=1:nelem
   kg(2*ielem(e)-1,e)= cos(telem(e)); % x component of i node for element e
   kg(2*ielem(e)  ,e)= sin(telem(e)); % y component of i node for element e
   kg(2*jelem(e)-1,e)=-cos(telem(e)); % x component of j node for element e
   kg(2*jelem(e)  ,e)=-sin(telem(e)); % y component of j node for element e
end

% applying boundary conditions
for i=1:nfixed
   kg(2*kfixed(i)-1,nelem+2*i-1)=1;      % x component of fixed node
   kg(2*kfixed(i)  ,nelem+2*i  )=1;      % y component of fixed node
end
for i=1:nslide
   kg(2*kslide(i)-1,nelem+2*nfixed+i)=cos(tslide(i)-pi/2); % x comp of sli nod
   kg(2*kslide(i)  ,nelem+2*nfixed+i)=sin(tslide(i)-pi/2); % y comp of sli nod
end

% applying loading conditions
for i=1:nforce
   fg(2*kforce(i)-1)=fforce(i)*cos(tforce(i)); % x component of applied force
   fg(2*kforce(i)  )=fforce(i)*sin(tforce(i)); % y component of applied force
end

% solution
xg=-kg\fg

% results for current case: (Example 6-1, Statics, 10e, R.C.Hibbeler}
% xg =
%  500.0000 - F1 , Member 1 in tension
%  500.0000 - F2 , Member 2 in tension
% -707.1068 - F3 , Member 3 in compression
% -500.0000 - R1x, negative x direction
% -500.0000 - R1y, negative y direction
% -500.0000 - R3 , normal to and pointing away from the sliding plane
```