INTRODUCTION TO FINITE ELEMENT ANALYSIS IN MET CURRICULUM

Thomas G. Boronkay, P.E.
Associate Professor
Mechanical Engineering Technology
OMI College of Applied Science
University of Cincinnati

ABSTRACT

Advances in microcomputer technology, i.e. faster speeds, more memory and better graphical display devices, opened up the way for using Finite Element Analysis methods on microcomputers.

Several new microcomputer based Finite Element software programs are available on the market, extending the use of this method to both industrial and educational users. The low cost and user friendliness of these programs make them ideally suited for an introductory course.

This paper describes how the Finite Element method is introduced to Mechanical Engineering Technology seniors at the OMI College of Applied Science, using the MSC-PAL2 software.

INTRODUCTION

The Finite Element Analysis method was developed in the late nineteen fifties primarily as a structural analysis tool for the aerospace industry. Its usage was restricted to large main frame computers and dedicated analysts were needed for problem solutions. The advent of Microcomputers and interactive software widened the user base. Medium size companies could afford the computers and software packages. The availability of pre and post processors with their graphical input and output allowed the software to be used by less specialized personnel.

Finite Element Analysis is introduced in most Engineering schools as a senior undergraduate or dual undergraduate/graduate course. These courses dwell very heavily on the mathematical foundations of the finite element method theory of elasticity and variational principles. Most of the asssignments consist of derivation of stiffness and/or mass matrices of various structures and calculations of stresses and deflections using computer programs written by the students. Most Engineering Technology schools do not offer courses in this subject.

This paper describes the approach taken by the Mechanical Engineering Technology department at the OMI College of Applied Science, University of Cincinnati in introducing students to the concept and applications of the Finite Element Analysis.

BACKGROUND

The college is a four year ABET accredited Technology school within the University of Cincinnati offering Associate and Bachelor of Science degrees in Architectural, Construction Science, Electrical and Mechanical Engineering Technologies and Associate degrees in Chemical Engineering Technology and Fire Science.

Currently one of the Technical Elective courses offered to the Mechanical Engineering technology student is the Computer Aided Design course. It is available to juniors or seniors who have taken various Mechanics of Materials, Design, Mathematics and Computer Graphics courses. Finite Element Analysis and its applications are introduced as one of the components of the CAD course.

COURSE CONTENT

The course is comprised of four elements:

a) Introduction to Matrix Structural Analysis
b) Introduction to Modeling in Finite Element Analysis
c) Introduction to Microcomputer based commercial F E. software (MSCPAL2)
d) Projects

a) Introduction to Matrix Structural Analysis

Starting with very basic and familiar concepts such as the spring constant of a helical spring and the equivalent stiffness of springs in series and parallel, the student is gradually introduced to the topic of Stiffness Matrix for continuous systems. Ref 1.

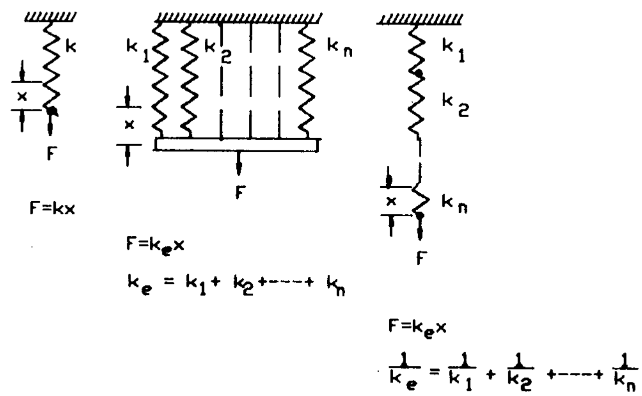
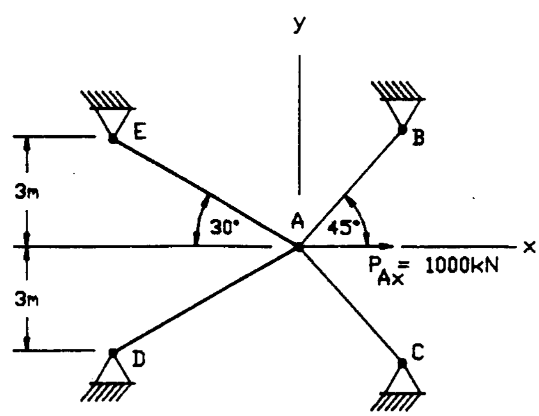


Fig. 1

Local and Global coordinates and Transformation Matrices are introduced and several homework exercises are assigned. Fig. 2 depicts a typical assignment.



Area of members AB and AC = 3000 mm (each)
" " " AE and AD = 5000 mm (each)

a) Determine the force in each member

b) Determine the displacement of A

(Hint: Use symmetry)

Fig. 2

b) Introduction to Modeling

Ref. 3 & 4 are used to introduce the basics of modeling. Various types of elements and their applications are discussed. The effect of the number of elements used in a model is illustrated by the simple frame model shown in fig. 4 & 5. The effect of element size and density is also discussed and illustrated using various examples.

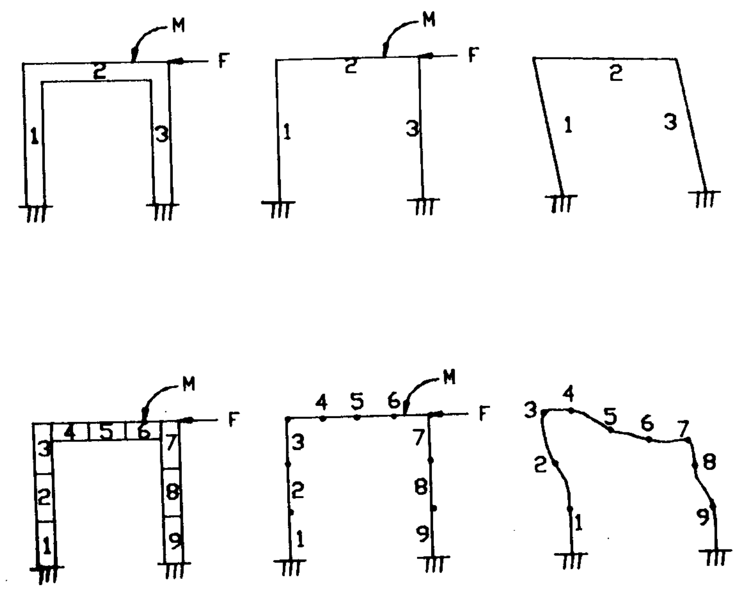


Fig. 3

c) Introduction to Microcomputer based Finite Element Software

The software selected for this introductory course is the MSCPAL2 marketed by the MacNeal-Schwendler Corporation. Although it has limitations in the type and number of elements and the number of nodes and degrees of freedom that may be input, it was selected for the following reasons:

- Easy to learn.

- Good pre and post processing capabilities including automatic node and mesh generation, good graphics displays, stress contours, etc.

- Good example problems in the manuals.

- Inexpensive (several copies of the 2C student version were purchased for less than $500.)

In this portion of the course the emphasis is on learning the overall structure of the PAL2 software. All commands in the Model Definition Solution Definition as well as the Static Analysis and Plotting commands are discussed and explained using examples from Ref 5. Time does not permit the study of Dynamic Analysis.

d) Projects

Three homework projects requiring the use of MSCPAL2 are assigned:

As an illustration of the simplest of elements, i.e. the beam element, the first assignment requires the calculation of the deflection of a cantilever beam loaded by a single vertical force at the free end.

The second problem deals with two dimensional flat plate elements. The problem definition, with the pertinent data, is depicted in Fig. 4. Fig. 5 & 6 show the stress contours of the maximum principal and Von Mises stresses obtained by analyzing the given plate. (only one quarter of the structure had to be modeled due to symmetry.)

a) Calculate the stresses in the plate shown subject to the inplane loading condition shown below.

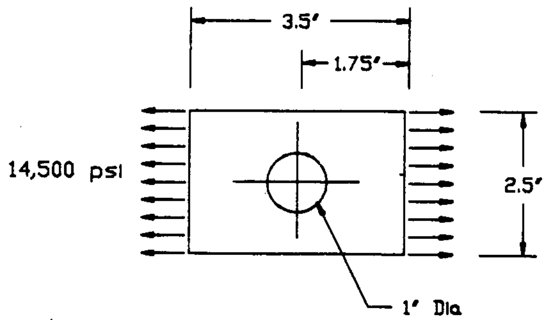


Plate thickness = 0.050"

E = 30×10^6 psi, = 0.3

Typ = 40x10 psi

b) What is the factor of safety?

c) Compare results to manual calculations.

Hand in printouts and graphics showing stresses & deflections and answers to b) and c).

Fig. 4

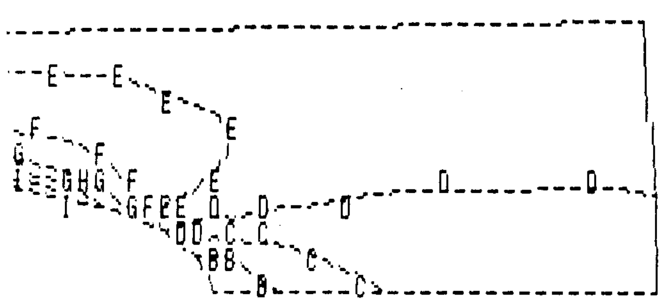


Fig. 5

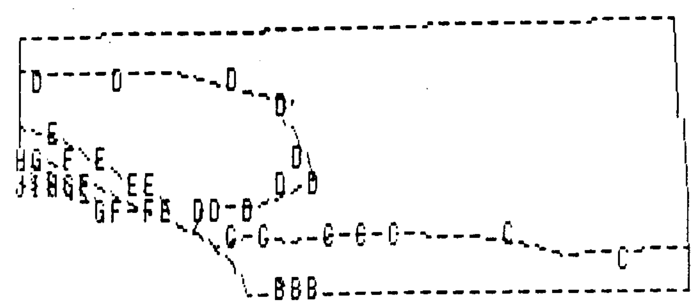


Fig. 6

The third project assignment, described in more detail below, is used to illustrate the analysis of a three dimensional structure.

Problem statement

Determine the safety factor of the cylinder bracket of the 20 ton press shown in the attached figures. The material is plain, low carbon steel plate.

Yield stress = 36000 psi
Ultimate stress = 50000 psi
Poisson ratio = 0.3
Modulus of Elasticity = 30000000 psi
Specific weight = 0.283 lb/cuin
Shear Modulus = 11500000 psi

Note: State your assumptions.
Take advantage of symmetry.

Fig. 7 depicts the press (without the cylinder). Fig. 8 depicts the enlarged detail of the bracket to be analyzed.

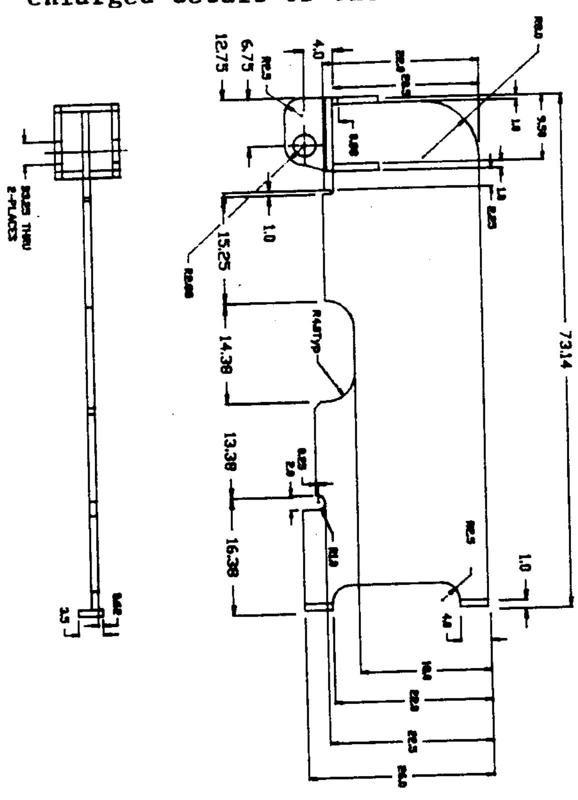


Fig. 7

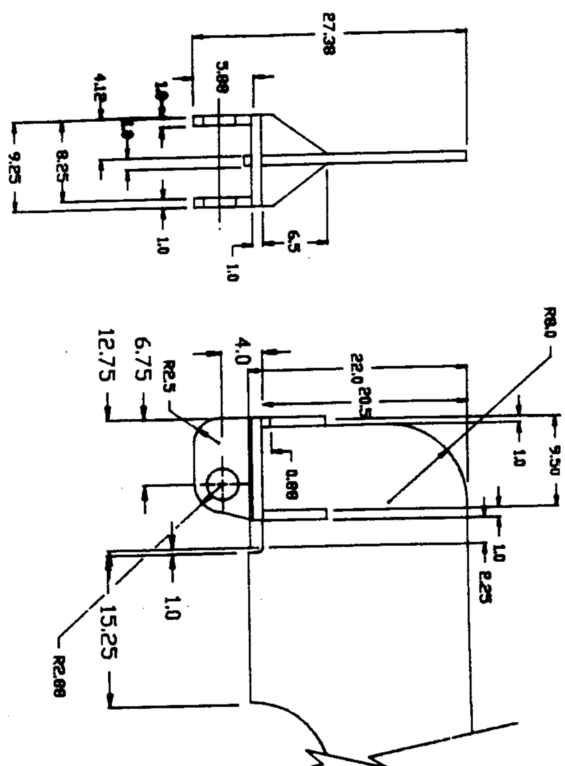


Fig. 8

Because of the relative stiffness of the frame compared to the bracket being analyzed, it was assumed to be perfectly rigid. Hence all degrees of freedom of the nodes at the point of attachment of the bracket to the main frame were set equal to zero.

After the stresses are calculated the students have to use their knowledge of Machine Design for calculating the Safety Factor, based on the failure criterion (Von Mises, Maximum Shear, etc.) applicable to the given material.

CONCLUSION

Our experience shows that introducing the Finite Element Analysis method to four year Engineering Technology students can be achieved by starting with simple and familiar concepts of Mechanics of Materials followed by an introduction to Matrix Structural Analysis. The implementation can best be demonstrated by the use of Microcomputer based commercial F.E. software packages such as MSCPAL2. These packages have the advantage of being interactive, simpler to learn and use, yet sophisticated enough to solve meaningful homework and sample problems.

REFERENCES

1- Matrix structural analysis, William McGuire & Richard Galaher
John Wiley, 1979

2- Analysis and Design of Multiple Input Flexible Link Mechanisms
Thomas Boronkay & Chuh Mei, International Journal of Mechanisms vol.5, 1970

3- Introductory level Finite Element Analysis program, John Hengehold
Senior Thesis, OMI College of Applied Science, University of Cincinnati, 1984

4- Applied Finite Element Analysis, Frank L. Stasa, Holt, Rienhart & Wilson, 1985

5- User manuals, MSCPAL2, MacNeal-Schwendler Corporation.

Taking advantage of symmetry about the Y axis only one half of the bracket had to be modeled. Fig. 9, 10, 11 and 12 dipict the XY, ZX, ZY and pictorial views of the model of the bracket.

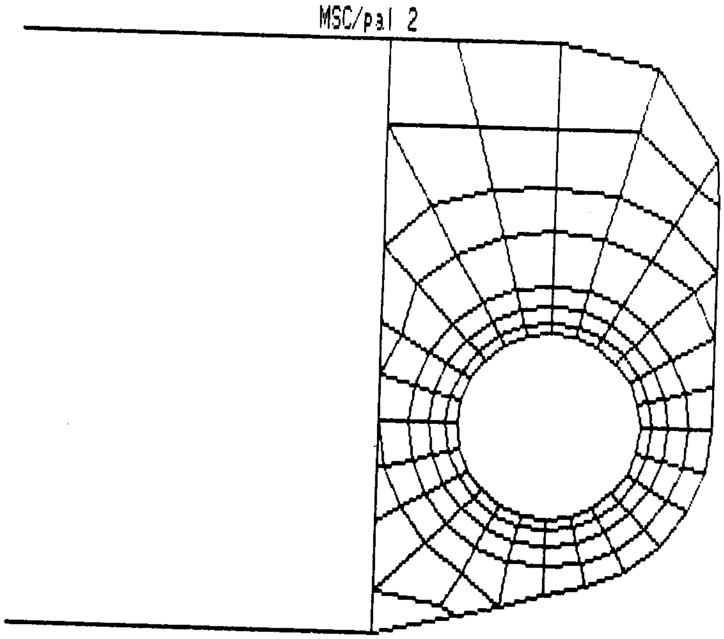


Fig. 9

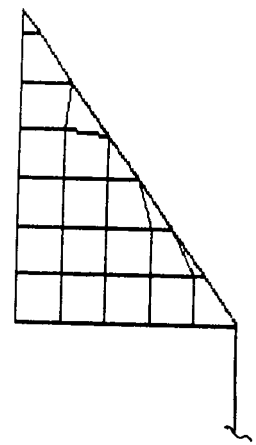


Fig. 10

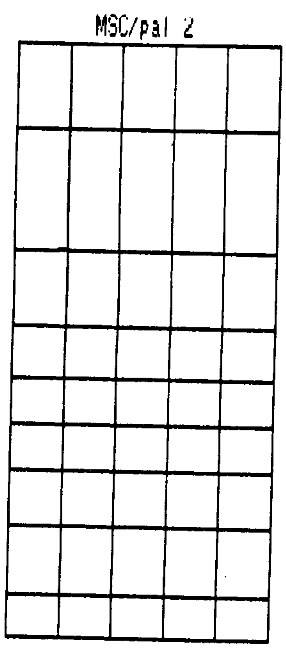


Fig. 11

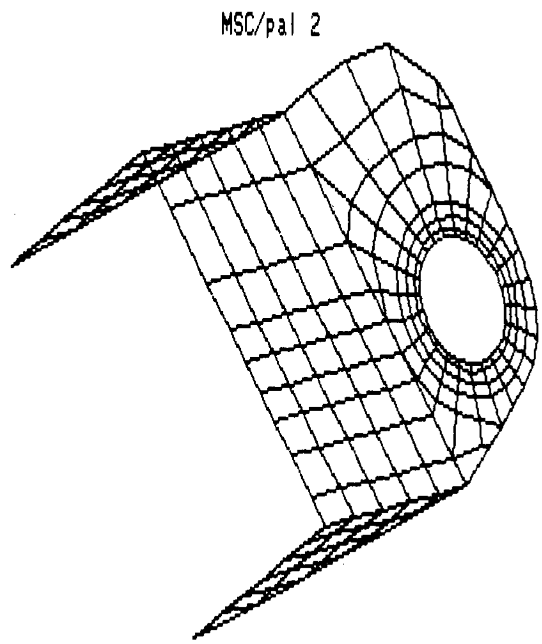


Fig. 12