

4-1975

Computer Aided Structural Analysis: The Finite Element Method

Richard T. Ryner
University of Northern Iowa

Let us know how access to this document benefits you

Copyright ©1975 Richard T. Ryner

Follow this and additional works at: <https://scholarworks.uni.edu/grp>

Recommended Citation

Ryner, Richard T., "Computer Aided Structural Analysis: The Finite Element Method" (1975). *Graduate Research Papers*. 3724.

<https://scholarworks.uni.edu/grp/3724>

This Open Access Graduate Research Paper is brought to you for free and open access by the Student Work at UNI ScholarWorks. It has been accepted for inclusion in Graduate Research Papers by an authorized administrator of UNI ScholarWorks. For more information, please contact scholarworks@uni.edu.

Offensive Materials Statement: Materials located in UNI ScholarWorks come from a broad range of sources and time periods. Some of these materials may contain offensive stereotypes, ideas, visuals, or language.

Computer Aided Structural Analysis: The Finite Element Method

DEPARTMENT OF
INDUSTRIAL TECHNOLOGY
University of Northern Iowa
Cedar Falls, Iowa 50614-0178

Wagner Resource Center

COMPUTER AIDED STRUCTURAL ANALYSIS:
THE FINITE ELEMENT METHOD

Approved by

Graduate Committee, Chairman

Date

May 2, 1975

COMPUTER AIDED STRUCTURAL ANALYSIS:
THE FINITE ELEMENT METHOD

RESEARCH PAPER
Presented to the
DEPARTMENT OF INDUSTRIAL TECHNOLOGY
UNIVERSITY OF NORTHERN IOWA

In Partial Fulfillment
of the Requirements for the Degree
MASTER OF ARTS

by
Richard T. Ryner
April 1975

Computer Aided Structural Analysis:

The Finite Element Method

A Research Proposal for Presentation
to the Graduate Committee
of the Department of Industrial Arts
and Technology
University of Northern Iowa

In Partial Fulfillment of the Requirements for
the Non-Thesis Master of Arts Degree

by
Richard T. Ryner

November , 1974

Approved by:

~~Technical Advisor~~

Graduate Committee, Chairman

Nov. 11, 74
Date

Nov. 18, 1974
Date

ACKNOWLEDGMENTS

The author wishes to express sincere appreciation to the following persons for their cooperation and assistance in the development of this paper:

Mr. C. E. Jones
Lockheed Structural Analysis Service
Lockheed Palo Alto Research Laboratory
3251 Hanover Street
Palo Alto, California 94304

Mr. James L. Rogers, Jr.
Aerospace Technologist
NASTRAN Systems Management Office
NASA Langley Research Center
Hampton, Virginia 23665

Mr. Edward C. Martin
Information Services
Computer Software Management and Information Center
Suite 112, Barrow Hall
University of Georgia
Athens, Georgia 30602

Dr. M. T. Wilkinson
Associate Professor
Department of Mechanical Engineering
College of Engineering
Louisiana Tech University
Ruston, Louisiana 71270

Mr. John Selby
Director, Academic Computing Services
University of Northern Iowa
Cedar Falls, Iowa 50613

Mr. Kenneth L. Cox
Programmer-Analyst
Academic Computing Services
University of Northern Iowa
Cedar Falls, Iowa 50613

Dr. W. E. Luck
Director of Industry Programs
Department of Industrial Technology
University of Northern Iowa
Cedar Falls, Iowa 50613

TABLE OF CONTENTS

	Page
LIST OF TABLES	vi
LIST OF FIGURES	vii
 Chapter	
1. OVERVIEW	1
Introduction	1
Statement of the Problem	1
Definition of Terms	3
Limitations	5
Presentation	6
2. WHY USE THE FINITE ELEMENT METHOD IN THE DESIGN PROCESS?	7
The Nature of the Design Process	7
Advantages of the Finite Element Method	9
Capabilities of the Finite Element Method	10
Problems which Have Been Solved Using Finite Element Methods	11
3. WHAT IS REQUIRED OF THE ORGANIZATION TO MAKE USE OF THE FINITE ELEMENT METHODS	23
General Considerations	23
Organizational Size	25
Selecting Hardware and Software	25
Personal Requirements	28
Budget Requirements	30

Chapter	Page
4. WHEN TO GET INVOLVED WITH FINITE ELEMENT METHODS	32
General Considerations	32
Some Presently Available Finite Element Computer Programs	34
5. EFFECTIVE IMPLEMENTATION OF THE INFINITE ELEMENT METHOD INTO THE ENGINEERING FUNCTION	37
6. BASICS OF FINITE ELEMENT THEORY	40
Introduction.	40
Framework Analysis	40
7. USING A FINITE ELEMENT METHOD COMPUTER PROGRAM	53
Introduction.	53
Inputting Data	53
Output Results	56
Conclusion	57
8. A SMALL SAMPLE PROBLEM	58
The Problem	58
Solution by Handbook Formulas	59
Desai/Abel "INTROFEM" Program	62
ICES STRUDL-II Program	63
"INTROFEM" Input: "Simple Beam Problem"	65
"INTROFEM" Output: "Simple Beam Problem"	67
STRUDL-II Input: "Simple Beam Problem"	78
STRUDL-II Output: "Simple Beam Problem"	82
Comments	105
9. CLOSURE	106

Chapter	Page
Summary	106
Conclusion	107
Implications and Recommendations	108
BIBLIOGRAPHY	109
APPENDIXES.	112
A. Description of SAMIS Program	112
B. Description of NASTRAN (Level 15.5)	115
C. Description of SNAP	122
D. Description of ICES	131
E. STRUDL II Elementary User's Manual	168
F. "INTROFEM" User's Information	199
G. "INTROFEM:" Introduction to the Finite Element Method, Example Program for Plane Strain/Plane Stress	213

LIST OF TABLES

Table	Page
1. Engineering Applications of the Finite Element Method	12
2. Minimum Budget Requirements for a Small Organization, \$	31
3. Minimum Budget Requirements for a Large Organization, \$	31

LIST OF FIGURES

Figure	Page
1. The Organization of a Digital Computer	4
2. Structural Member of Baby Ace	13
3. NASTRAN Plot of Orbiter Fuselage	14
4. Orbiter Fuselage under Assembly	14
5. Frame and Finite Element Model	15
6. Comparison of Stress Results	16
7. Chassis Frame Idealization for Dynamic Analysis	16
8. Comparison of Predicted and Measured Mode Shapes of Chassis Frame	17
9. Proposed Body Structure	17
10. Analysis Projects Underway	18
11. Computer Model of Lift Arm and Bucket	19
12. Three-Dimensional Model--Concrete Alternate	20
13. Steel Finite Element (Stick) Model	20
14. CAD Hip Stem.	22
15. Steps in the Analysis Process and Capabilities Required	26
16. Finite Element Interactive Graphics Facility	29
17. Decentralized Analysis Department Approach	38
18. Centralized Division Analysis Approach	39
19. Centralized Corporate Analysis Department Approach	39
20. One-Dimensional Elastic Element	41
21. Two One-Dimensional Elastic Elements	42

Figure

Page

22.	Representation of a Plane Pin-Jointed Framework	44
23.	A Plane Stress Region Divided into Finite Elements	46
24.	Bridge Truss	46
25.	Elastic Elements	49
26.	Finite Element Models of Wheel	50
27.	Representation of "Simple Beam Problem"	58
28.	Finite Element Idealization of "Simple Beam Problem"	61

"It is unworthy of excellent men to lose hours like slaves in the labor of calculation."

Gottfried Wilhelm von Leibniz

Chapter 1

OVERVIEW

Introduction

The technical core of the modern manufacturing organization is being placed under pressure from many directions in today's organizational environment. The traditional pressures of competition from other producers are still with us. But these pressures have been joined by other factors. These include consumer pressures for product improvements, government pressures for greater safety to the operator, pressures for noise reduction, greater reliability, less environmental harm, faster and easier maintenance and repair, less weight, greater fuel economy, smaller usage of expensive raw materials, more efficient use of expensive labor resources, shorter design and development times, etc. All these factors have formerly been hidden in the pressure of competition. But today, they are reaching a level of importance where they must be dealt with in their own contexts. And the result is pressure on the engineering department for better designs in less time for less expenditures of labor and money.

Statement of the Problem

The design process is a complex and tedious one. Within it, the task of design verification appears to require the lion's share of time, effort, and money involved.

The design formulation activity requires creative people, placed in an environment where they are able to formulate ideas which eventually become the manufactured products of an organization. This task takes up comparatively little of the time and effort of the design cycle. Therefore, it would appear that the greatest savings could be made in the design analysis task. Further, if these savings can be made earlier in the design cycle, in the prototype, preprototype, and even the concept phases, they would generate even greater savings in time and money. Such savings could then allow analysis of a greater number of design alternatives, and a more optimum design could be selected for the product at an earlier date.

In one form or another, design evaluation involves some form of modeling. Physical models may be of any size ratio with respect to the final product and may be made of whatever material is most suitable for the desired results. In testing, these models are placed in conditions analogous to the conditions which the product is expected to encounter. The use of such physical models involves a considerable number of problems. They require time and money to build; testing them requires more time and money; they are typically built only after the design process has gone some distance from the concept stage; they have a finite and somewhat unpredictable life; the cost per test can be quite high; and they are not usually very portable and must be stored somewhere.

Many, if not all, of these problems could be avoided if a mathematical model of the structure of a product could be used. Such a mathematical model could be built at an early stage in the design cycle, perhaps even the concept stage. It could have a virtually unlimited life and a low per test cost. Such a model could be duplicated easily and

could be easily portable. And the time required to set up and perform a test could be less.

This paper will focus on one method for creating and analyzing such mathematical models of structures. The method studied is the Finite Element Method, often known as FEM. Finite element theory has been in existence for almost 20 years (Hamann, 1974, p. 30) and is basically an extension of traditional methods of structural analysis of frames and framed structures. With the proliferation of the high-speed digital computer in the 1960's, the finite element method has become a useful tool for the engineer. Only with the use of a high-speed digital computer can the large number of calculations necessary for the use of the finite element method be performed in a reasonable length of time.

Definition of Terms

Calculus of Variation -- The study of the theory of maximum and minimum of definite integrals whose integrands are known functions of one or more independent variables and their derivatives, the problem being to determine the dependent variables so that the integral will be a maximum or a minimum.

Digital Computer -- "...a machine that can be instructed or programmed to carry out a sequence of numerical calculations. For our purposes it will be sufficient to visualize that a computer consists of the following components, interconnected as shown:

- (a) A memory or store in which numbers and instructions can be stored and from which any number can be produced at will.
- (b) An input mechanism for transferring instructions and initial data for a problem from the outside world into the memory.
- (c) An output mechanism for transferring information from the memory on to an output sheet.
- (d) An arithmetic unit for carrying out simple basic arithmetical operations such as addition, subtraction, multiplication, division.
- (e) A control unit that organizes the calculations, i.e., arranges for the input and output of information and the execution of arithmetic operations in the correct sequences as specified by the instructions.

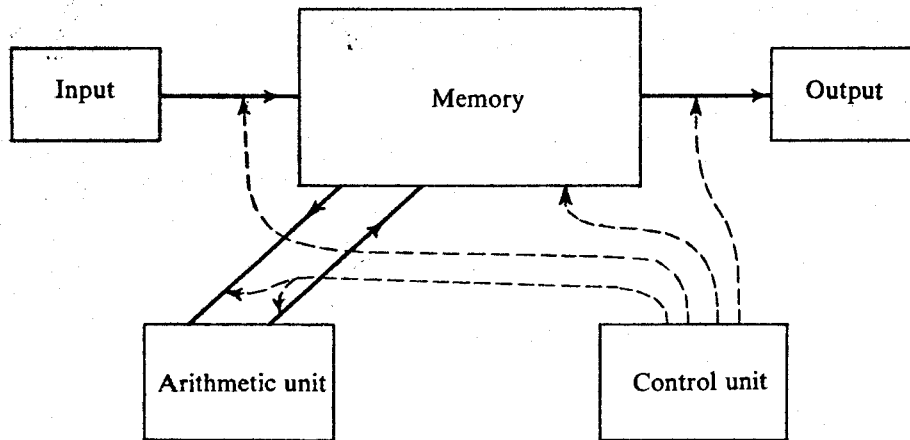


Figure 1

The Organization of a Digital Computer
(Noble, 1969, p. 35)

"In order to perform a calculation, the computer must be provided with a sequence of instructions called a program. When solving a problem, the computer starts by storing the complete program in its memory. It then proceeds to obey the instructions in a sequence determined by the program." (Noble, 1969, p. 35-36).

Dynamics -- A branch of mechanics that deals with forces and their relation primarily to the motion (but sometimes also to the equilibrium) of bodies of matter.

Eigenvalue -- The so called "characteristic value" of a matrix. Eigenvalues are often used in the matrix solution of systems of simultaneous differential equations. This makes them of value in dynamic analysis problems.

Finite Element Method -- "...a process through which a continuum with infinite degrees of freedom can be approximated to by an assemblage of subregions (or elements) each with a specified but now finite number of unknowns. Further, each such element interconnects with others in a way familiar to engineers dealing with discrete structural or electrical assemblies." (Zienkiewicz, 1971, p. vii).

Fortran -- (FORMula TRANslation) A widely used computer program source language. "FORTRAN is based on the language of algebra plus a few rules of grammar and syntax imposed by the nature of the computer. It was developed primarily for engineers and scientists familiar with mathematics." (Stehling, 1972, p. 28).

Matrix -- A matrix is a rectangular array of numbers. The elements of a matrix are denoted by a subscripted variable, where the first subscript indicates the row of the matrix and the second subscript indicates the column. Thus, in matrix notation, a_{ij} indicates the

element of matrix A located at the intersection of row i and column j . A digital computer manipulates matrices through the use of subscripted variables.

Tensor -- An abstract object having a definitely specified system of components in every coordinate system under consideration and such that, under transformations of coordinates, the components of the tensor undergo a transformation of a certain nature.

Statics -- A branch of mechanics dealing with the relations of forces that usually produce equilibrium among material bodies.

Limitations

The primary limitation of this study is in the depth of the presentation of finite element theory presented. According to Desai and Able, a thorough understanding of the theory and applications of the finite element method requires that the student have a working capability in several fundamental areas. "These fields include matrix algebra, solid mechanics, variational methods, and computer skills." (Desai/Abel, 1972, p. 15). Obtaining such a working capability in these areas has been found to be impossible, given the constraints of time.

Though a number of finite element computer programs will be mentioned, primary emphasis will be given to the finite element capabilities of STRUDL II (STRUctural Design Language, version II), which is a part of ICES (Integrated Civil Engineering System), developed by the Massachusetts Institute of Technology. This capability is available to students and faculty of University of Northern Iowa through the UNI Academic Computing Services link with the University of Iowa Regional Computer Center.

Finally, the number of computer runs presented herein will be kept small. The cost of these runs, though well within the capabilities of a manufacturing organization, are considerable in the context of a research paper.

Presentation

According to Carl and Hamann, there are several basic questions which should be considered by the manager contemplating the introduction of the finite element method into the engineering function of his organization before even considering the details of the technique. They are:

1. Why should the finite element method be used at all? (Easily the most basic question.)
2. What are the material, financial, and personnel resources required?
3. When is the best time to get involved with the technique in light of continuous development and advancement?
4. How should the technique be introduced to engineering departments in a cost and time effective manner? (Carl/Hamann, 1974, p. 23).

The first portions of this paper will deal with these questions.

The latter portion of this paper will deal with the mechanics of the finite element method in a non-technical manner. A simple statics problem will be constructed and will be solved using three methods: a simple, strength of materials, handbook formula; a simple finite element method computer program; and the finite element capabilities of STRUDL II. These results will be compared and conclusions drawn.

Chapter 2

WHY USE THE FINITE ELEMENT METHOD IN THE DESIGN PROCESS?

The Nature of the Design Process

Any design process involves two basic activities: the formulation of the design (the "idea" activity), and the verification of the design (the "test" activity).

According to R. L. Davis of Chrysler Corporation, a design cycle is made up of four phases: the Concept Phase, the Preprototype Phase, the Prototype Phase, and the Production Phase. (Davis, 1974, p. 8). During the Concept Phase, an idea is formulated in the designer's mind and a crude model is built to determine the worth of the idea. "Design analysis in the concept stage is concerned with the elimination of structural problems during the evaluation of new features." (Davis, 1974, p. 8). In the Preprototype Phase, the concept is further refined and evaluated by modifying an existing version of the product. So called "soft tooling" is used to manufacture the modified parts and initial structural design verification is carried out. The Prototype Phase often sees the bringing together of many design changes in the production prototype. Endurance and other destructive test methods may be used to verify the overall design. During the Production Phase, the production drawings of the product are released, including any changes which were pointed up as being necessary by the prototype testing. Hard tooling is built (if it was not built during the Prototype Phase), and representative units of the finished product are subjected to final

evaluation. Further design efforts on the product will be limited to correcting deficiencies which turn up in the field and the addition of further features to design.

A different perspective is given by Mr. Keith S. Minard, Manager of Engineering Services for John Deere Waterloo Tractor Works, with respect to designing a farm tractor transmission:

Our general design procedure is to first state the objectives. One or more persons will then make schematic diagrams of the transmission and determine the major features including arrangement and vehicle speeds. Usually one or two arrangements are selected for further study involving scale feasibility layouts and preliminary cost estimates. Subsequently, a prototype design is built to verify function of the chosen concept and the preliminary cost estimates.

The design may then become committed for production, rejected or deferred. Assuming that the proposal is accepted, a full-fledged design effort is initiated and a number of units built for further development and durability testing. During this development program, design changes are made to correct deficiencies in the design prior to release for production. Obviously, a simple design with minimal changes from production units requires little in the way of design/development effort while a complex design may require much time and effort. (Minard, personal letter to R. Ryner, October 24, 1973, p. 1-2).

Implicit in this description is the task of analyzing the proposed designs and selecting the design which will best meet the objectives of the design effort. Obviously, faster and better design analysis methods would result in a better initial choice of the design concept and smaller expenditures in correcting design deficiencies.

Some of the problems faced by the engineer of today have been mentioned in the introductory sections of this paper. An excellent elaboration is given by Carl and Hamann in their paper:

Typically, today's market is characterized by rising labor and material costs and vast shortages of many important resources. Foreign competition is a much more significant

factor than it was a few years ago. The attempt of designers and engineers to reduce structural overdesign and factors of safety has been met with opposing requirements as a result of increased government and industry requirements for improved structural integrity and performance. In short, the engineer is faced with the extremely challenging and conflicting task of building better products with less overdesign, cheaper products with less material, and new products in shorter lead times, while maintaining product performance, improving structural integrity, and using fewer labor resources. Quite frankly, these are difficult if not impossible objectives to accomplish using present "subjective" design techniques. (Carl/Hamann, 1974, p. 23).

Advantages of the Finite Element Method

As has been stated previously, finite element methods are basically used to construct models of some structure or construction with the purpose being the analysis of the subject under certain conditions of loading, stress, temperature, pressure, vibration, impact, flow, etc. According to R. L. Davis (1974, p. 9), finite element methods have several advantages. First, the analysis can begin at an early point of the design cycle. The models constructed using FEM have an infinite life and can be easily changed or modified. Thirdly, the per test cost can be very low when compared with other modeling and analysis techniques. The model may be easily duplicated and is quite portable (it can even be transmitted over telephone lines). And lastly, the model requires little storage space or set-up time for tests.

Hamann (1974, p. 33) lists further advantages of using finite element methods for validating structural designs. Computer programs are operational and available to the engineering community which have broad capabilities for a comparably small cost. The method is generalized:

Given the right type of elements, almost any kind of structure can be analyzed. Complete generality in material properties is permitted. A component can be changed from steel to plastic with a change in few constants. (Hamann, 1974, p. 33).

Finite element modeling is also adaptable to any of the various stages of the design cycle. The models generated by most of the available computer programs may also be "substructured;" that is, divided into substructures which may then be analyzed separately. This allows lower analysis costs and permits more than one analyst to work on the development of a structure. When the finite element model is analyzed under the desired test conditions, the structural information obtained is much more complete than would be available from tests of physical models. And these results are consistent from test to test, since the finite element model is only influenced by the variables controlled by the analyst.

As with anything else, finite element methods have their limitations. A finite element method computer program will not design the structure or component, predict the fatigue life, assure that the structure is feasible to manufacture or construct, or assure packaging feasibility. (Hamann, 1974, p. 33). However, its use has advantages which are too great to ignore in the validation of design alternatives.

Capabilities of the Finite Element Method

The finite element method is not static in its development. Structural engineers working on its continued development are expanding its capabilities almost on a daily basis. As problems or deficiencies are found, methods are found to solve these problems using finite elements. As a result, any list of the method's capabilities must be necessarily incomplete.

Throughout all the applications of finite element methods, its most important capability is the ability to:

...solve problems with complicated geometry. A major limitation of classical mathematics is that only simple geometries

can be practically solved. Arbitrary geometric shapes are the rule, not the exception, in real-life problems. Prior to finite element methods, we were stuck with trying to approximate complex structures with mathematics derived for grossly simplistic shapes. (Smith, 1974, p. 2).

Table 1 presents a list of applications of the finite element method in various engineering fields.

Problems which Have Been Solved Using Finite Element Methods

Before making a sizable investment in implementing finite element methods into the engineering department of his organization, a manager should ask for examples of the types of problems the method is able to deal with in respect to his company and industry. Most managers of manufacturing companies would prefer to spend resources for a capability which permits the greatest return in the shortest amount of time. A capability which requires years of development to adapt it to a particular industry is not especially attractive in today's world of tight profit margins and intense competition. The purpose of this section is to illustrate how the finite element method is being used "on line" in the engineering functions of several industries.

Aerospace. A great deal of the capabilities in the finite element method computer programs was developed by the aerospace companies during the 1960's. This development is being carried on today even in light of cut-backs in government funding for aerospace research and projects. The economies allowed by the use of the method have made its further development and use the preferred alternative to expensive prototype development, analysis, and testing.

Table 1
Engineering Applications of the
Finite Element Method

<i>Fields of Study</i>	<i>Equilibrium Problems</i>	<i>Eigenvalue Problems</i>	<i>Propagation Problems</i>
1 Structural engineering, structural mechanics, and aerospace engineering.	Analysis of beam, plate, and shell structures. Analysis of complex or hybrid structures. Two- and three-dimensional stress analysis. Torsion of prismatic sections.	Stability of structures. Natural frequencies and modes of vibration of structures. Linear viscoelastic damping.	Propagation of stress waves. Dynamic response of structures to aperiodic loads. Coupled thermoelastic and thermoviscoelastic viscoelastic problems.
2 Soil mechanics, foundation engineering, and rock mechanics.	Two- and three-dimensional stress analyses. Construction and excavation problems. Slope stability problems. Soil-structure interaction. Analysis of dams, tunnels, boreholes, culverts, locks, etc. Steady-state seepage of fluids in soils and rocks.	Natural frequencies and modes of vibration of soil-structure combinations.	Transient seepage in soils and rocks. Flow-consolidation in deformable porous media. Propagation of stress-waves through soils and rocks. Dynamic soil-structure interaction.
3 Heat conduction.	Steady-state temperature distribution in solids and fluids.		Transient heat flow in solids and fluids.
4 Hydrodynamics, hydraulic engineering, and water resources.	Solutions for potential flow of fluids. Solutions for viscous flow of fluids. Steady-state seepage in aquifers and porous media. Analysis of hydraulic structures and dams.	Seiche of lakes and harbors (natural periods and modes of oscillation). Sloshing of fluids in rigid and flexible containers.	Salinity and pollution studies of estuaries (diffusion). Sediment transport. Unsteady fluid flow. Wave propagation. Transient seepage in porous media and aquifers.
5 Nuclear engineering.	Analysis of reactor containment structures. Steady state temperature distributions in reactors and reactor structures.		Dynamic analysis of reactor containment structures. Thermoviscoelastic analysis of reactor structures. Unsteady temperature distribution in reactors and reactor structures.

The model illustrated below was constructed using rod and beam elements to model the "Baby Ace" D model homebuilt aircraft. The model was constructed by personnel at Louisiana Technical University in connection with their course in finite element methods for engineering students.

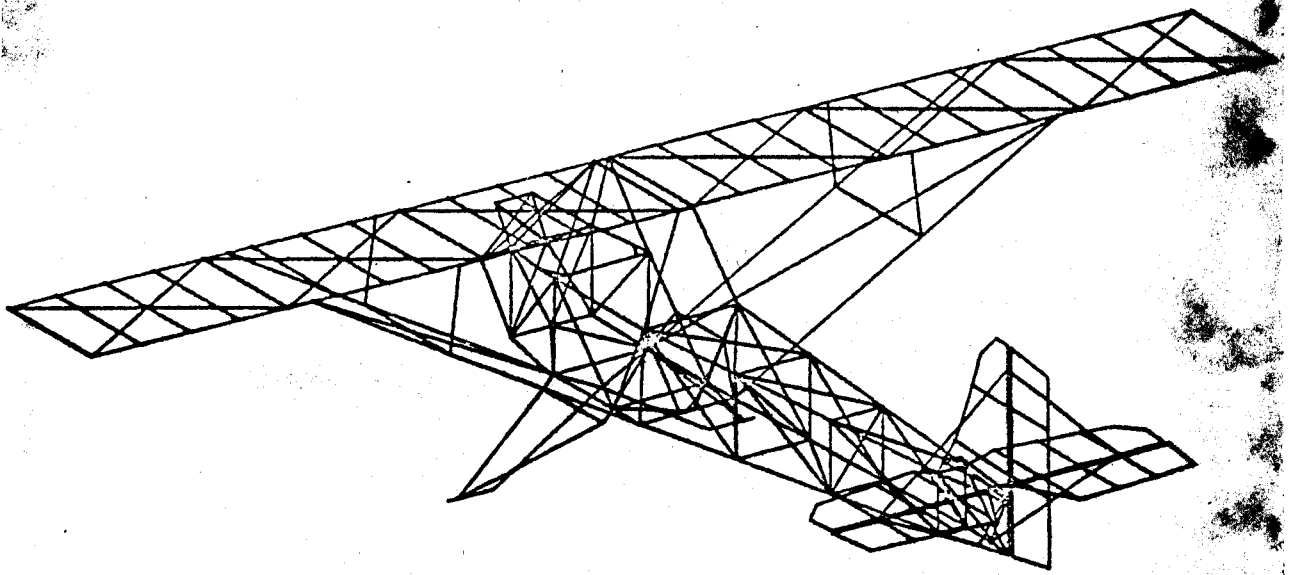


Figure 2

Structural Member of Baby Ace
(Wilkinson/Bruce, 1973, p. 124)

Another illustration of an aerospace usage of the finite element method is shown in Figure 3. This is a plot of the finite element model of the fuselage of the NASA space shuttle orbiter fuselage. This model was constructed by technicians at Grumman Aerospace Corporation and was used to analyze the dynamic characteristics of the design before the prototype stage. Figure 4 shows 1/8th scale model of the fuselage under construction, corresponding to the finite element model.

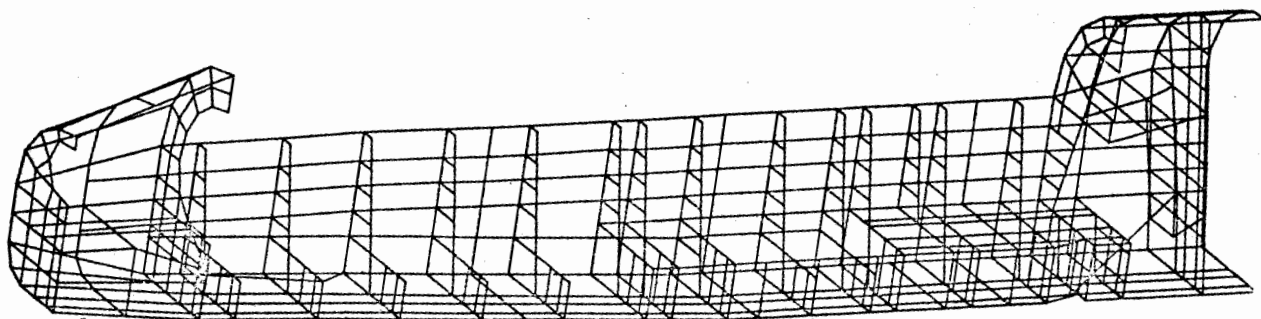


Figure 3
NASTRAN Plot of Orbiter Fuselage
(Bernstein, et al, 1973, p. 213)

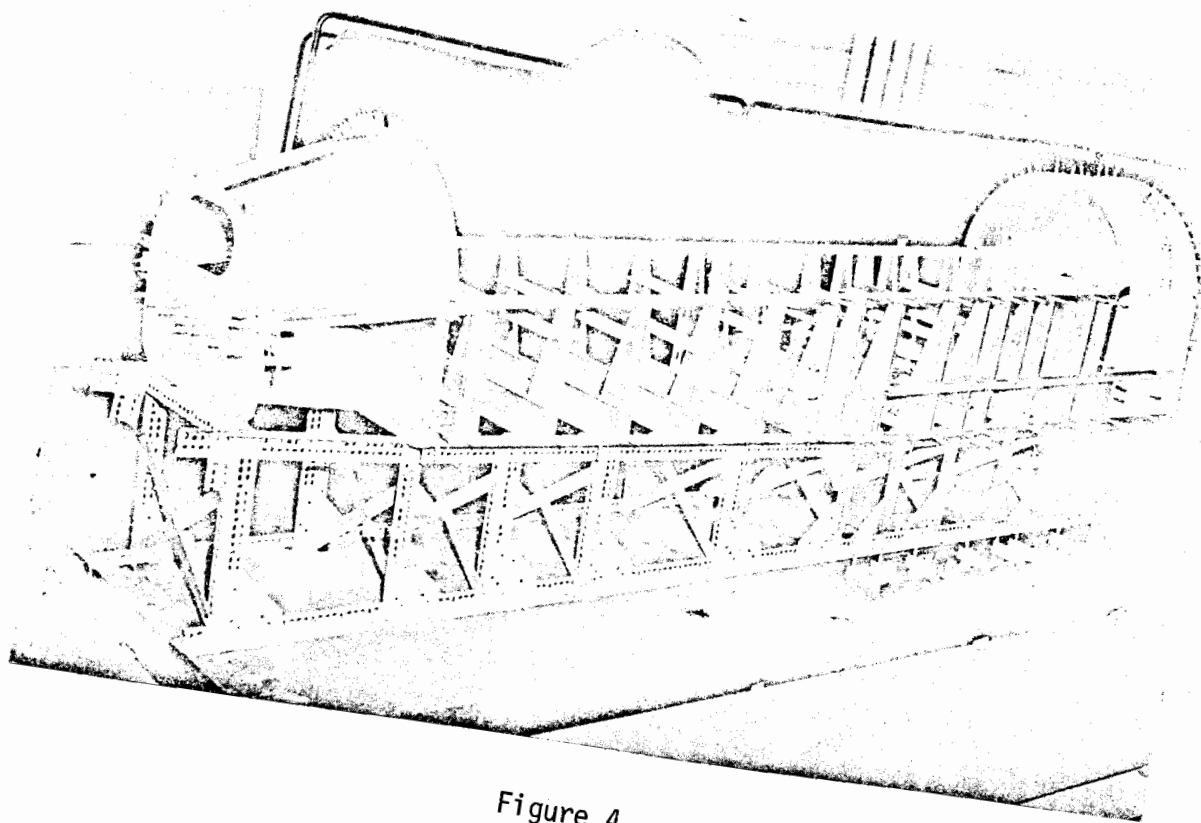


Figure 4
Orbiter Fuselage under Assembly
(Bernstein, et al, 1973, p. 214)

There are many other examples which could be cited in the aerospace field.

Automotive design. The automotive industry is one of the more recent users of finite element technology. With the availability of more "ready to operate" finite element computer programs, this industry promises to be a large user in the near future.

The design of chassis and frames of automobiles has received the attention of engineers using finite element methods. Figure 5 shows a frame and its finite element model. The accompanying table shows the comparison between the finite element stress analysis and a stress analysis using a plastic model. Also included are the results of a finite element analysis of a redesign to allow the frame to withstand a 5 mph rear impact. According to the author, "The general trend of stresses were in very good agreement." (Davis, C. S., 1974, p. 11).

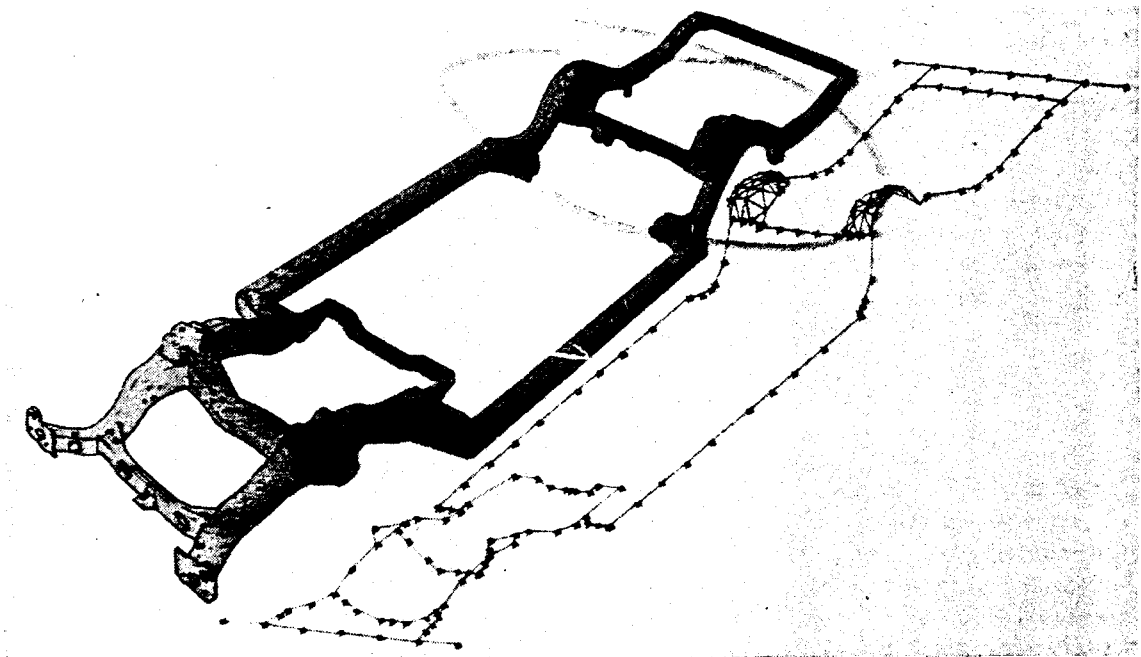


Figure 5

Frame and Finite Element Model
(Davis, C. S., 1974, p. 12)

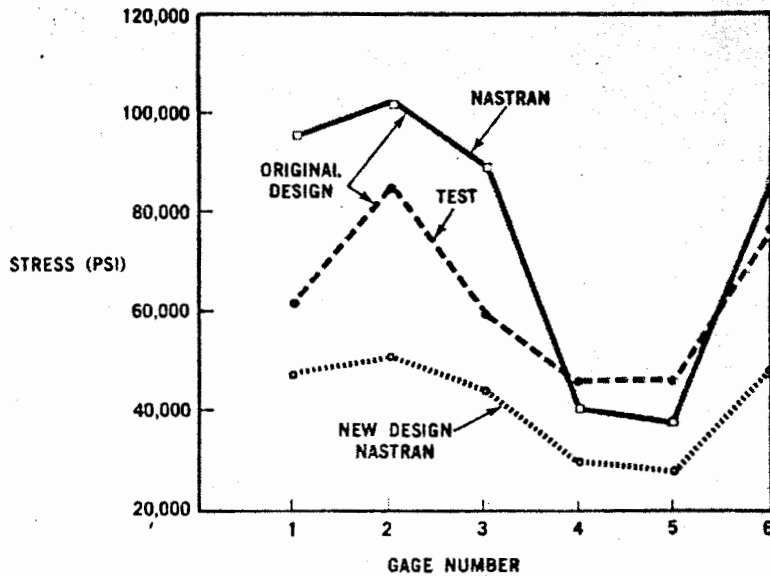


Figure 6

Comparison of Stress Results
(Davis, C. S., 1974, p. 12)

A dynamic analysis of an automobile frame structure is illustrated in Figures 7 and 8. The results of this and other dynamic analyses by finite element techniques have been shown to agree with experimental techniques to within 10%. (Vail, 1974, p. 19).

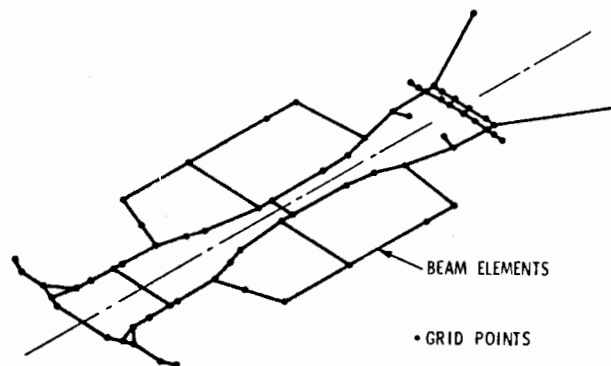


Figure 7

Chassis Frame Idealization for Dynamic Analysis
(Vail, 1974, p. 19)

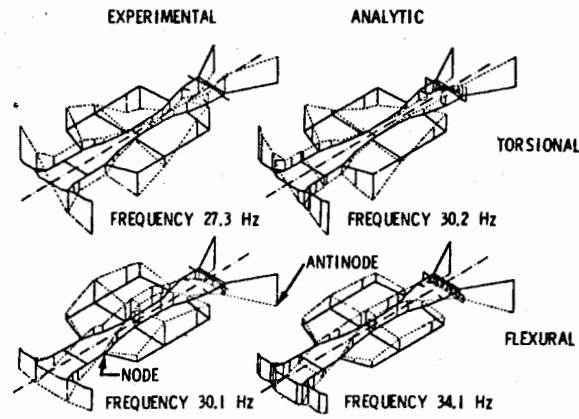


Figure 8

Comparison of Predicted and Measured
Mode Shapes of Chassis Frame
(Vail, 1974, p. 19)

Finally, Figure 9 shows the comparison between representation of an actual vehicle body and the finite element model of the body.

According to Smith (1974, p. 6):

The model shown was used to study front and rear impact at low speeds for bumper design loads and at high speeds for trends during front and rear collisions. Roof crush was studied to develop the window pillar and roof structure for rollover conditions. Torsional and beaming rates were also determined from the model.

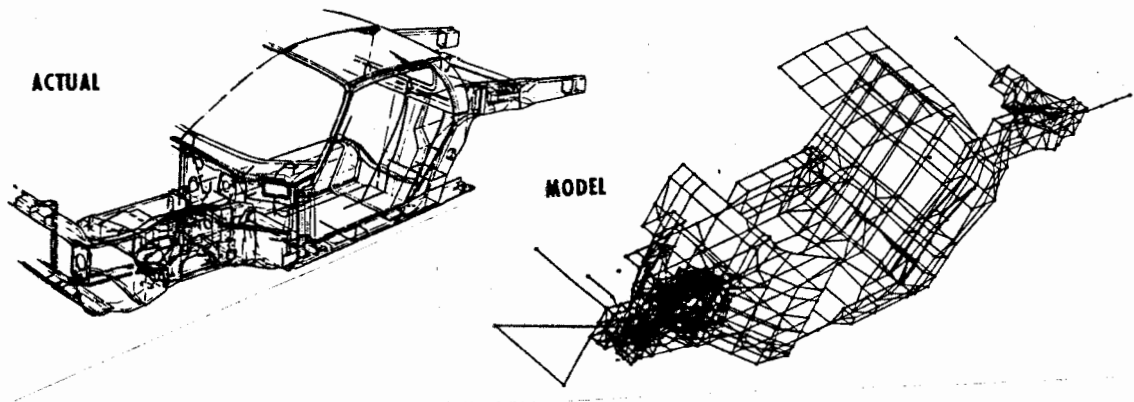


Figure 9

Proposed Body Structure
(Smith, 1974, p. 7)

Additional uses of finite element methods in automotive engineering have included modeling tire construction, fan blades, steering components, wheels bumpers, mufflers, and suspension components.

The following figure shows some of the projects currently using finite element analysis at Ford Motor Company:

COMPONENT OR SYSTEM	CAR DOOR SUBSTRUCTURE, RADIATOR SUPPORT	CAR FRAME, STEERING LINKAGE, SUSPENSION ARMS	TURBINE BLADES	RESEARCH ENGINE
FINITE ELEMENT MODEL	PLATE ELEMENTS	BEAM AND PLATE ELEMENTS	PLATE AND SOLID ELEMENTS	AXI-SYMMETRIC AND SOLID ELEMENTS
ANALYSIS CONDUCTED	STATIC STRESS, DEFLECTION & BUCKLING ANALYSIS, VIBRATION ANALYSIS	STRESS AND BUCKLING ANALYSIS, VIBRATION ANALYSIS	STRESS AND VIBRATION ANALYSIS	HEAT TRANSFER AND STRESS ANALYSIS

Figure 10

Analysis Projects Underway
(Hamann, 1974, p. 33)

In related areas, finite elements have also been used to model components of agricultural and construction equipment. The following illustration shows a finite element model of an industrial loader lift arms and bucket. This model was developed to evaluate the distribution of stresses in the assembly under load.

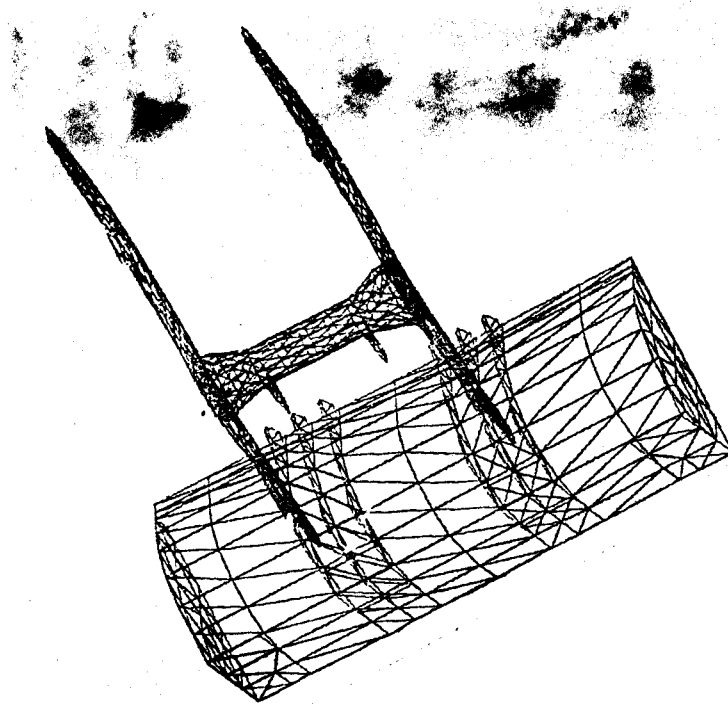


Figure 11

Computer Model of Lift Arm and Bucket
(Sherlock/Kormos, 1973, p. 8)

Civil engineering applications. Finite element methods are enjoying a widening usage in fields relating to civil engineering. They have been used to analyze the design of buildings, concrete structures, bridges, dams, drilling platforms, roadways, and other structures under conditions of load, pressure, impacts, earthquake forces, etc.

Figures 12 and 13 show finite element models of the West Seattle Freeway Bridge over the Duwamish waterway. Several versions of the design for this bridge were analyzed to determine the best design for wind and earthquake resistance. A simple analysis using stick models as well as the three-dimensional finite element model analysis was made.

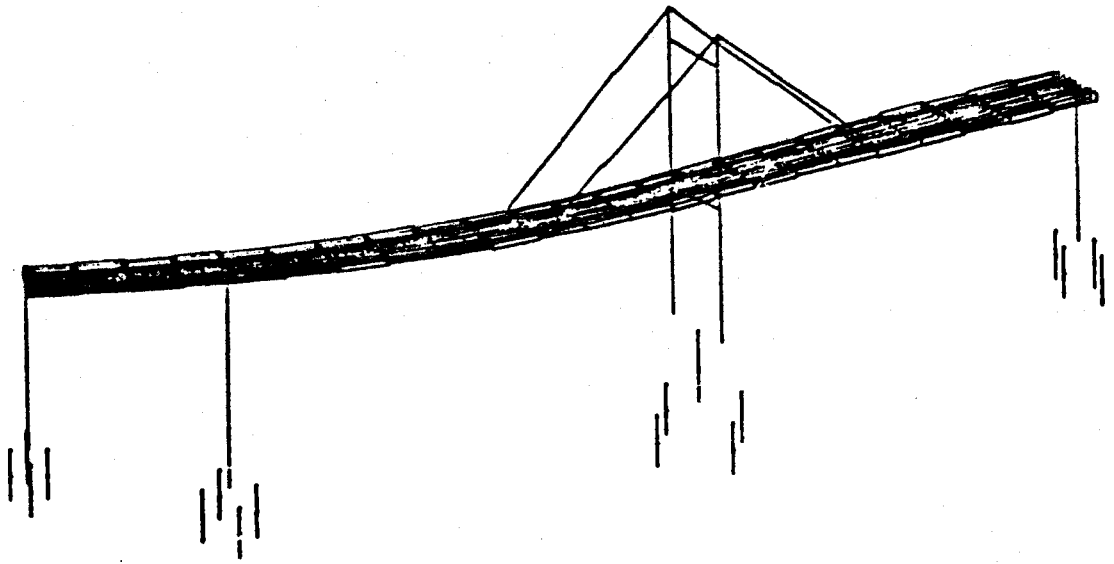


Figure 12

Three-Dimensional Model--Concrete Alternate
(Salus/Jones, NASA IM 2893, p. 157)

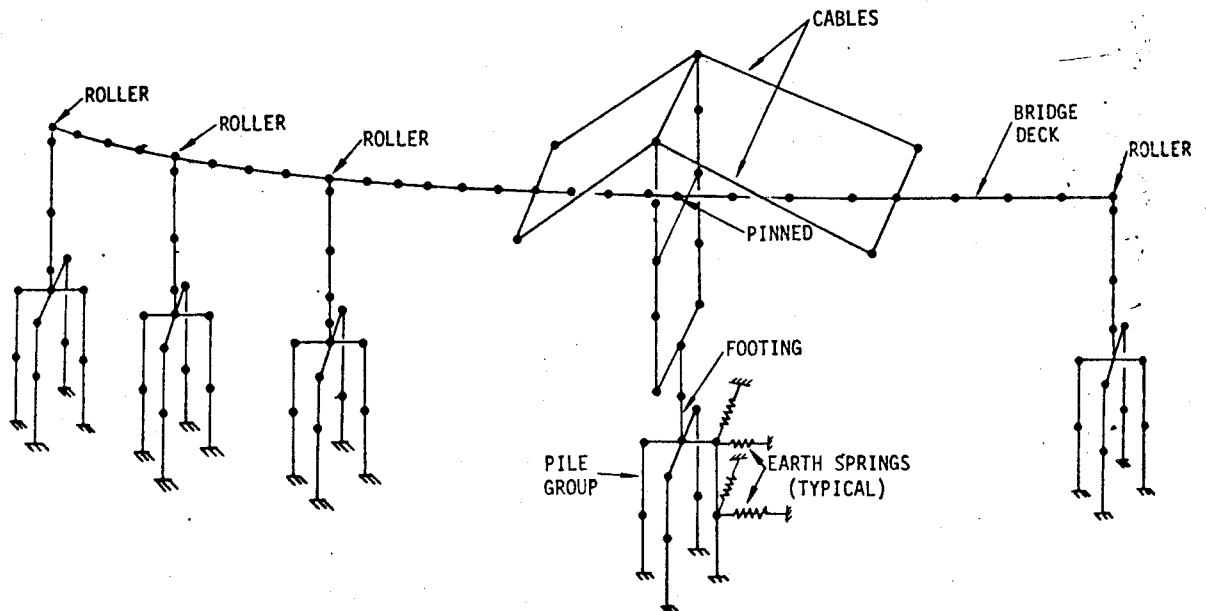


Figure 13

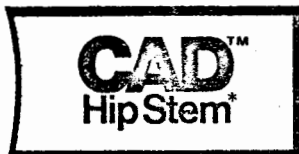
Steel Finite Element (Stick) Model
(Salus/Jones, NASA IM 2893, p. 157)

Soil and rock mechanics applications. Applications of finite element methods in soil and rock mechanics include:

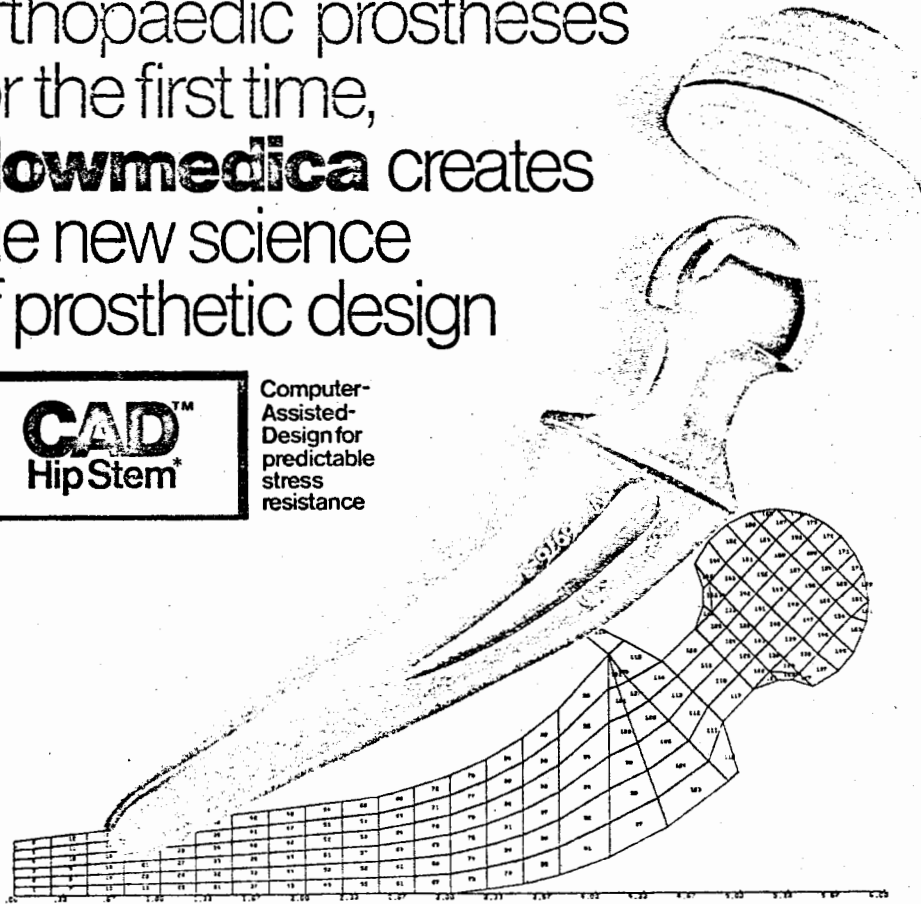
1. analysis of deformation and stability of slopes, excavations, embankments, dams, and riverbanks;
2. determining the bearing capacity and settlement characteristics of footings, foundations, and piles;
3. determining the interactions between soils and structures;
4. analyzing rock structures, such as tunnels, mines, pits, boreholes and cavities; and
5. predicting the characteristics of rock joints, fissures, fractures, and layers.

Other uses. Finite element methods are also being used to predict the characteristics of reinforced and pre-stressed concrete beams and structures. And at least one manufacturer of orthopaedic prostheses has used finite element methods in evaluating the design of implanted prostheses. Figure 14 shows a full hip replacement prosthetic which was analyzed for stress vulnerability using finite element techniques.

Applying space-age
stress analysis to
orthopaedic prostheses
for the first time,
Howmedica creates
the new science
of prosthetic design



Computer-
Assisted-
Design for
predictable
stress
resistance



Structural integrity of prostheses: The biomechanics of prosthetic design are well established, and untold numbers of patients have benefited from relief of pain, increased mobility and greater stability of affected joints. However, human and mechanical factors imposing stress on implanted prostheses are numerous, and at present the anticipated life of an orthopaedic replacement is largely unpredictable.

Stress analysis for the first time: To resolve the problem of stress in orthopaedic prostheses, Howmedica applied space-age stress analysis to total hip prostheses. Using the finite element method under varying conditions, a computer program calculated the degree of stress on each element and then was able to predict precise areas of vulnerability.

* Patent applied for.

CAD™ prostheses: the new science of prosthetic design: Howmedica, in combining computer precision and predictability with proven biomechanical principles, has created the new science of Computer-Assisted Design and a new class of hip prostheses designated CAD™ Hip Stem. CAD™ hip prostheses from Howmedica provide the outstanding features of traditional prostheses, as well as new features to fortify against stress, in accordance with Computer-Assisted Design.

A new measure of structural security: Various prostheses with the letters CAD™ preceding their name are now available only from Howmedica. These are traditional prostheses modified according to Computer-Assisted Design.

Figure 14

CAD Hip Stem
(Howmedica, Inc.)

Chapter 3

WHAT IS REQUIRED OF THE ORGANIZATION TO MAKE USE OF FINITE ELEMENT METHODS

General Considerations

Assuming that the needs of the organization warrant the use of finite element methods, and the capabilities of the method will aid in fulfilling these needs, then an assessment of the expenditures required to make use of finite element technology is necessary. Many other such decisions facing the organization take on the character of the "make or buy" decision. Although it would be possible to develop one's own finite element technology in the organization, and indeed, a great amount of this technology has come about in this manner, the expenditures necessary to follow this course are immense. Coupling this with the fact that a great deal of finite element technology is available at a reasonable cost to the engineering community from established sources, it can be seen that such development expenditures are hardly necessary. Therefore, here, we shall be concerned with the procurement of available technology from outside sources.

In procuring finite element capabilities there appear to be three possible courses of action available to the organization. First, the entire analysis job may be "farmed out" to firms especially equipped and staffed to the entire job, from constructing the finite element models to carrying out and interpreting the results. Such a course of action would be most productive for the smaller company with only an

occasional need for the structural analysis capability. Such analysis firms (such as Structural Dynamics Research Corporation and Lockheed Structural Analysis Service) often work on a predetermined fee basis and can provide an excellent service for the smaller company.

A second alternative involves the employment of one or more trained analysts by the organization and the use of a computer service center. This alternative requires the rental of computer capabilities as well as the payment of a royalty fee to the computer center for the use of the finite element computer program. Such a course of action is probably best for companies that are involved in a moderate amount of analysis and can afford to hire an analyst familiar with finite element methods.

The third course of action is most applicable to large organizations engaged in work requiring a great deal of finite element analysis. This involves the hiring of one or more full time analysts, the purchase of a finite element computer program, and the rental or purchase of sufficient computer hardware to carry out analysis on a full-time basis.

With respect to these last two alternatives, Mr. C. E. Jones of Lockheed Structural Analysis Service has the following comments:

In my opinion, the acquisition of a finite element computer program should be based entirely on the amount of finite element analysis performed by the organization. For moderate amounts of analysis, nationwide computer service centers (CDC, UCC, etc.) provide programs for which the user must pay a royalty fee. An advantage of this is that the user always has available the most up-to-date program versions. At some point, of course, an organization should consider acquiring their own program. Acquisition could be based purely on economics. Finite element programs can be purchased for tremendously varying amounts of money. Several programs like SNAP are in the public domain and can be purchased from NASA for a moderate amount. Proprietary programs like ANSYS and STARDYNE are very expensive and can only be purchased from their developers. One advantage of proprietary program is that the developer or seller usually provides expert consulting services to the user. (Jones, personal letter to R. Ryner, September 20, 1974).

The remainder of this section will be concerned with some of the factors to be considered when implementing an "in house" finite element analysis capability.

Organizational Size

An important factor in procuring and implementing a finite element method capability is the size of the organization we are concerned with.

According to Carl and Hamann (Carl/Hamann, 1974, p. 23, 27), a small organization is one which has:

1. Limited or no in-house engineering computer facilities.
2. Limited or no in-house structural analysis computer codes.
3. Limited or no dedicated research and development functions.
4. Limited or no computer programming support.

On the other hand, the engineer in the large organization will often have access to such resources as:

1. A dedicated in-house computer facility.
2. Internally generated structural analysis codes.
3. An in-house technical programming staff available to write programs.
4. Experienced analysts available for consultation or analysis.

In the case of the larger organization, such capabilities as these may be spread through out the divisions of the organization or may be concentrated in a corporate engineering department, an advanced products department, or a research and development division.

Selecting Hardware and Software

Computer hardware (the computer, input and output devices, and supporting equipment) and computer software (the operating system program, the applications programming, and the input-output conditioning programs) are often interrelated and must be considered together when planning a

finite element method capability. Certain finite element programs are written in such a manner that they cannot be operated on all computers, finite element programs usually require a great deal of computer memory storage, and efficient use of finite element methods usually requires computer graphics and interactive capabilities.

This figure shows the steps in a typical finite element analysis and the hardware/software which may be used in association with each step:

DATA GENERATION	MESH GENERATORS, DIGITIZERS, SECTION PROPERTY PROGRAMS, PRE-PROCESSOR PROGRAMS
DATA VERIFICATION	PLOTTERS, GRAPHICAL DISPLAYS, GEOMETRY CHECKERS, QUALITY CONTROL PROGRAMS
DATA CORRECTION	INTERACTIVE GRAPHICS OR SPECIAL PURPOSE PROGRAMS
ANALYSIS	FINITE ELEMENT PROGRAMS, ACCESSIBILITY TO LARGE COMPUTERS
OUTPUT VISUALIZATION	ANALYST'S EYEBALL, PLOTTERS, GRAPHICAL DISPLAYS
OUTPUT EDITING	SEARCH PROGRAMS, INTERACTIVE EDITING ON GRAPHICAL DISPLAY
OUTPUT MANIPULATION	SUPERPOSITION PROGRAMS, CONTOUR PLOTTERS, INTERACTIVE GRAPHICS

Figure 15.

Steps in the Analysis Process
and Capabilities Required
(Hamann, 1974, p. 35)

According to Carl and Hamann (1974, p. 23-25), the choice of a computer applications package for a small organization should take into account the following factors:

1. the availability of time sharing capabilities to reduce analysis turn-around time;
2. if growth in finite element usage is expected, remote batch capabilities should be available;
3. the availability of graphical output (a must!);
4. the availability of analysis programs from the vendor; and
5. the amount of applications support the vendor can supply to aid in teaching company personnel in the use of the equipment and programs.

In selecting software, Carl and Hamann suggest that finite element programs be procured from a software house. This approach is desirable since:

1. often, a software house can supply a finite element applications library made up of a number of programs aimed at the customer's needs;
2. programs written by software houses are often documented in a "user's language" which is fairly easy for analysis personnel to learn;
3. documentation is usually explicit and clearly written;
4. software houses often supply training in the use and application of their programs; and finally,
5. software houses tend to keep their programs up to date.

(Carl/Hamann, 1975, p. 23-26).

Larger companies will often have computer facilities available to the engineer. However, due to the large memory requirements as well as

the amount of computer time required for finite element problems, the engineer wishing to use finite element analysis may have to go outside of his company for computer facilities. To allow flexibility in the use of in-house as well as external computer facilities, a remote job entry terminal controlled by a minicomputer may be used to route finite element analysis problems to the most efficient facility. If a computer graphics capability is incorporated into the terminal, an excellent system can be had. (Figure 16). (Carl/Hamann, 1974, p. 27-28).

The large organization has the same alternatives open as the small organization in the choice of software for finite element analysis. Again the approach of using a software house is probably the most desirable. The large organization may also choose to implement a finite element code which is in the public domain. Such codes are available from some universities and government agencies at a minimal cost. However, there is usually minimal applications support for these programs.

There is another aspect to the software question for the larger company. This is

...the software required for data generation, data verification, computer graphics, and visualization of output. This is an area where the large company has a distinct advantage over the small company--that is, it can afford to develop input/output software tailor-made for the particular type of design problems inherent in the company's business (for example, automotive engineering). (Carl/Hamann, 1974, p. 28).

Personal Requirements

Larger organizations may employ a number of analysts to employ the finite element method. A smaller organization will probably employ either an analyst-engineer, a full time structural analyst, or a full time structural analyst and a full or part time technical analyst. In

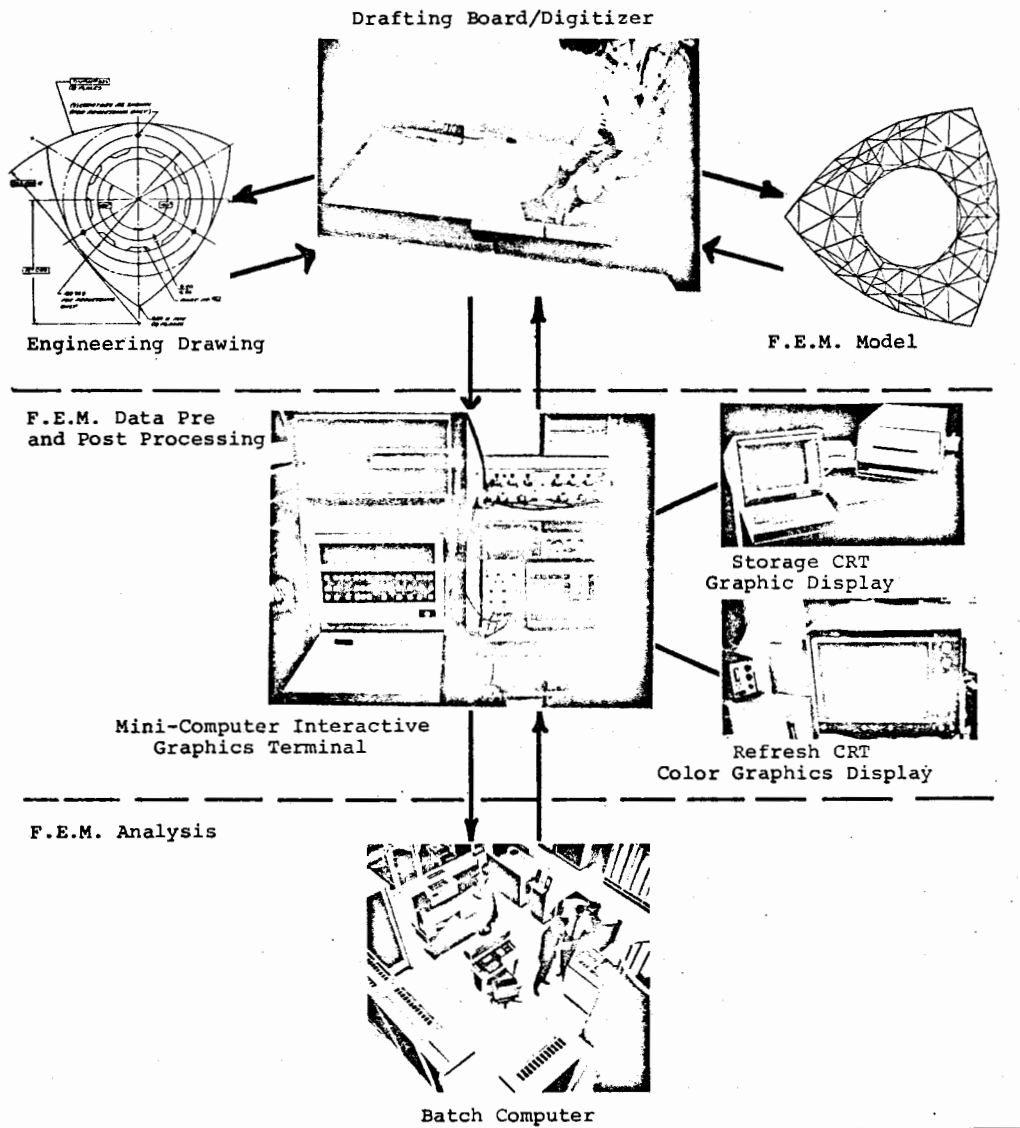


Figure 16

Finite Element Interactive Graphics Facility
(Carl/Hamann, 1974, p. 29)

any case, the job of analyst will require a number of duties besides the obvious job of performing finite element analyses. These include:

1. Search for applications within his company, define approaches, and perform analyses.
2. Keep up-to-date on latest techniques through reading and conferences.
3. Conduct regular educational sessions on FEM within his company.
4. Maintain information on hardware and software vendors.
5. Continually upgrade hardware and software facilities when economically justified.
6. Act as central source of knowledge for finite element techniques within the company. (Carl/Hamann, 1974, p. 27).

Where will these personnel come from? According to Carl and Hamann (1974, p. 26), likely sources are:

1. Colleges and universities (aerospace, mechanical, and civil engineers).
2. Job agencies specializing in geographic areas where the aerospace industry is big (Los Angeles, Seattle, San Francisco).
3. Research and development organization (NASA centers, Stanford Research Institute, etc.).
4. Automotive industry (Detroit).
5. Consultants (from firms that perform advanced design and analysis functions).

Budget Requirements

Finally, how much will this all cost?

The following tables give some idea as to the budget requirements for implementing finite element technology into the engineering function of the small and large organization. Note that the figures given were compiled in the early part of 1974, and the costs have probably increased since that time.

Table 2
Minimum Budget Requirements for a Small Organization, \$

<u>Resource</u>	<u>1st Year*</u>	<u>2nd Year**</u>	<u>3rd Year†</u>
Hardware	3,000	5,000	6,000
Software usage	12,000	30,000	50,000
Labor††	<u>16,000</u>	<u>25,000</u>	<u>40,000</u>
Total	31,000	60,000	96,000

*Low-speed terminal (30 character/s), one analyst.
 **Plotter, one draftsman added.
 †One analyst added.
 ††Does not include labor overhead.

(Carl/Hamann, 1974, p. 27)

Table 3
Minimum Budget Requirements for a Large Organization, \$

<u>Resource</u>	<u>1st Year*</u>	<u>2nd Year**</u>	<u>3rd Year†</u>
Hardware	20,000	25,000	30,000
Software usage			
Development	100,000	100,000	100,000
Applications	80,000	200,000	320,000
Labor			
Development	300,000	300,000	300,000
Applications	<u>125,000</u>	<u>250,000</u>	<u>500,000</u>
Total	625,000	875,000	1,250,000

*RJE terminal, card reader, printer, five analysts, ten development engineers.
 **Plotter, card punch, five analysts added.
 †CRT graphic display, ten analysts added.

(Carl/Hamann, 1974, p. 31)

Chapter 4

WHEN TO GET INVOLVED WITH FINITE ELEMENT METHODS

General Considerations

The question of the timing of an introduction of finite element methods and technology into the engineering function of an organization is one which requires some thought. If a decision has been made to implement the method, a major factor will be the capabilities of existing computer programs and the expected advances and additional capabilities to be introduced in the near future.

Other considerations include such factors as the time required to implement a finite element capability and the learning time required to use this capability. The learning curve for many finite element programs is fairly flat, and the implementation and debugging of these programs can range from a few weeks to several months. There is also the time involved in acquiring the appropriate hardware to use finite element techniques.

Projected advances in computer hardware may also be a factor in the decision as to when to go into finite element techniques. Advances over the past decade have permitted the introduction of "mini-computers" which have as much or more computing power of the large computers of the 1960's. The introduction of higher speed memory storage capabilities has improved the speed (and lowered the operating costs) of computers. It is quite possible that the near future may see small, inexpensive

computers with sufficient amounts of high speed memory to utilize finite element programs at a fraction of today's costs for a machine of similar capabilities.

Software improvements can also be expected in the near future. New finite element programs and systems are being developed which incorporate improved element types, more versatile elements, new mathematical techniques which require less computer time and storage area, more utilization of "machine language" to decrease running time, and greater capabilities in areas such as computer graphics, automatic data generation and error checking, interactive operation, etc. Existing programs are also being revised and updated to incorporate these features.

As time goes on, more and more experience in the use of finite element techniques will be built up. More engineering and technical schools will be training their students in the theory and/or use of the finite element method. The use of the method will invade more and more fields as time goes on. All of these will make the job of implementing and using finite element methods of analysis easier and less costly.

So the answer to the "when" question comes down to a question of need. If the organization needs the advantages and capabilities of finite element technology in its engineering function now, there are few reasons to put off its adoption. Finite element technology has considerable capabilities at its present "state of the art." If the need is not so pressing, then the manager's job becomes one of watching, waiting, self-education, and weighing the costs versus benefits. Most assuredly, the advantages for finite element methods will only increase.

Some Presently Available Finite Element Computer Programs

SAMIS. SAMIS (Structural Analysis and Matrix Interpretive System) was developed in the 1960's to solve problems involving matrix arithmetic in structural applications. It is capable of constructing structural matrices from input data containing an idealization of a structure. It then is able to manipulate these matrices according to the rules of linear algebra. In constructing the structural matrices, the program follows the stiffness or displacement method of finite element theory. A number of types of elements are provided by the program's element library.

SAMIS is a "chain" system in that it solves its problems sequentially instead in a simultaneous manner. Thus, computer running time is a limit on the size of problem which the program can handle. Hamann (1974, p. 31) points out a major problem with SAMIS in connection with experiences at Ford Motor Company:

The program had evolved by modification and extension. As a result, elusive and costly "bugs" appeared quite frequently. Engineers attempting to use the program to solve product engineering problems could not tolerate the delays associated with error resolution.

A description of SAMIS is included in the Appendix.

NASTRAN. NASTRAN (NASA STRuctural ANalysis) is a widely used, general purpose program for evaluating the behavior of elastic structures under various loading conditions. It is very versatile and will handle problems of any size (within computer time cost limitations). NASTRAN will solve four general classes of problems: 1. static structural problems; 2. elastic stability problems; 3. dynamic structural problems; and 4. general matrix problems. An experienced analyst can easily modify and expand the program to fit his own needs.

The program includes a considerable capability for graphical input and output. Many other smaller auxiliary programs are available to aid in the preparation of input for NASTRAN and the editing and display of NASTRAN output.

NASTRAN has been under development constantly since its creation. The currently available level is designated as Level 15.5 (level 15, modification 5). A number of major changes and improvements are scheduled to be included in Level 16.0, which is to be released by NASA in 1975.

Further description of NASTRAN is in the Appendix.

SNAP. SNAP (Lockheed Structural Network Analysis Program) is a fairly new finite element program used to analyze complex, highly interconnected finite element models. SNAP is available in three basic configurations:

1. a static analysis version;
2. a dynamic analysis version; and
3. a version which automatically generates fully-stressed design of structures subjected to multiple static loading conditions (FSD).

Numerous post-processor programs are available and future development of more capabilities is planned.

SNAP is a public domain program which uses a considerable number of advanced matrix storage and manipulation techniques. For this reason, the execution costs of the programs tend to be quite low when compared to some more generalized finite element programs.

A description of SNAP is included in the Appendix.

ICES - STRUDL II. ICES (Integrated Civil Engineering System) was developed at the Massachusetts Institute of Technology to be the basis for computer oriented, civil engineering problem solving. It is designed to be an information processing system for science and management,

engineering in general, and civil engineering in particular. ICES is a collection of computer programs, integrated into a large computer program system. The individual programs may be used to solve problems involving such things as coordinate geometry, foundation loadings in buildings, locating roadways, analyzing transportation networks, bridge design, the planning and control of construction or manufacturing projects, optimization techniques, evaluating the stability of slopes and embankments, working with traffic volume data, land usage problems, and analysis of structural frameworks and structures.

STRUDL II. (The STRUctural Design Language, version II) includes capabilities for the analysis of two and three dimensional structures, consisting of truss, frame, and continuous finite elements. A variety of element types are included, and problems concerning areas such as plane stress/plane strain, plate bending, and shell analysis may be solved. STRUDL II input and output may be coordinated with other ICES programs for greater versatility. In addition, capabilities for the analysis of concrete structures, dynamic structural analysis, nonlinear analysis, and substructuring are included. Continuing modification and expansion of ICES is planned.

Further information concerning ICES and a short STRUDL II finite element user's manual are contained in the Appendix.

Chapter 5

EFFECTIVE IMPLEMENTATION OF THE FINITE ELEMENT METHOD INTO THE ENGINEERING FUNCTION

Once an organization has decided to "take the plunge" into the use of finite element methods, the hardware and software have been selected, and personnel are being hired or trained, the question arises as to how the analysis function should be organized and integrated into the engineering function. Major determinants of this organization process will be the size of the organization and the amount of training necessary for the analysts.

The use of finite element methods requires a skilled analyst to achieve an accurate and credible analysis. He must be experienced and have a solid background in structural mechanics. To build up and maintain this degree of experience with finite element techniques, it is probably best that centralized divisional or corporate finite element analysis activities be established.

Smaller organizations utilizing finite element analysis may have only one or two analysts. Larger organizations may have a number large enough to warrant a separate analysis division.

On this subject, Carl and Hamann (1974, p. 28-30) evaluate three organizational strategies:

1. Have product engineers do their own analyses. Typically, in a large organization, this requires that several hundred engineers be trained in the theory and use of finite element techniques.
2. Have a small group of analysts in each division who provide analysis services to division engineers.

3. Have a centralized activity which provides analysis services to all company divisions and engineers. In many cases, the last approach is to be preferred from the standpoint of effective utilization of manpower, minimal training, and the development of technical proficiency. It also offers the financial means to support a terminal-based computer system, computer graphics, and related software development. When properly managed, a centralized activity offers an efficient use of human and financial resources.

Unfortunately, the price paid for centralization may be bureaucracy which is unable to respond quickly to user's needs, in which case the second approach may be desirable. Centralization of analysis capability may also not be feasible if the company's divisions are not centralized physically. Communications then become a significant problem.

These three alternative organizational structures are shown in Figures 17, 18, and 19.

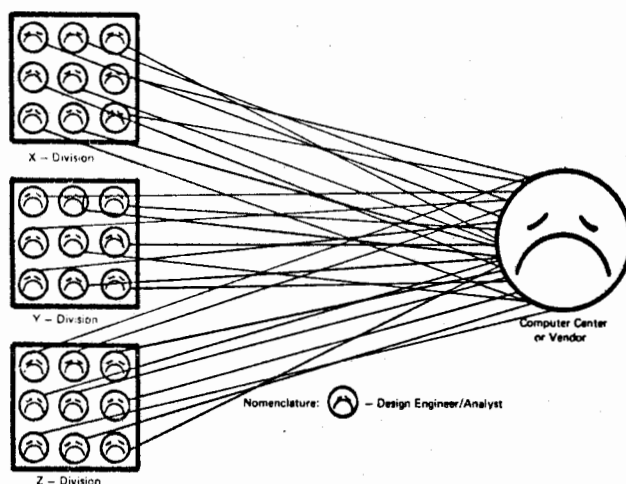


Figure 17

Decentralized Analysis Department Approach
(Carl/Hamann, 1974, p. 30)

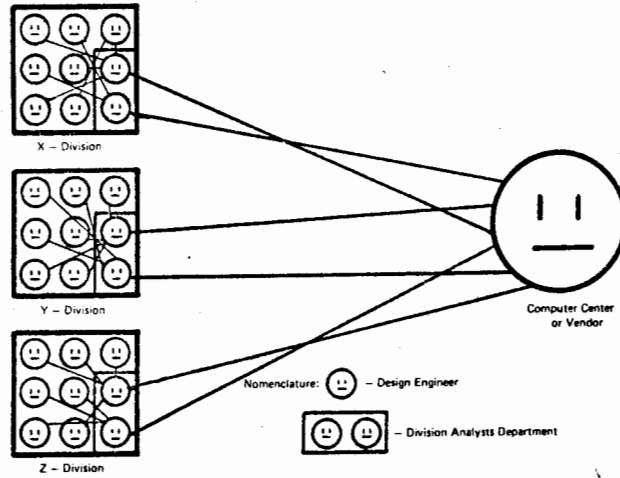


Figure 18

Centralized Division Analysis Department Approach
 (Carl/Hamann, 1974, p. 30)

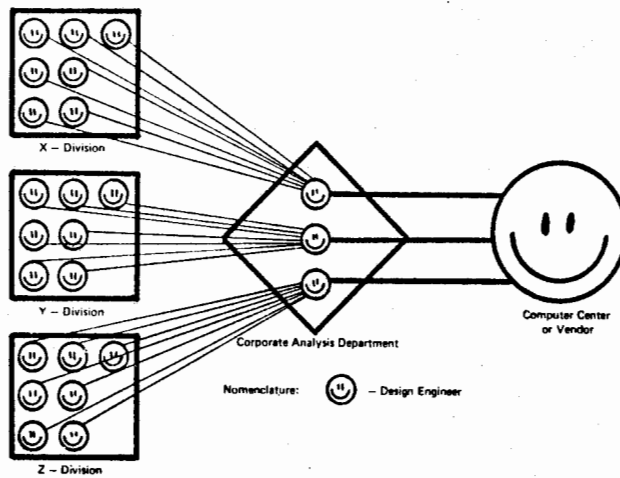


Figure 19

Centralized Corporate Analysis Department Approach
 (Carl/Hamann, 1974, p. 30)

Chapter 6

BASICS OF FINITE ELEMENT THEORY

Introduction

Some understanding of what the finite element method consists is eventually necessary so that one is able to grasp the power of the method. A manager may be able to make a decision on whether or not to adopt the method without a description of "how it works," but the technical mind will usually not be satisfied until it has some feel of what is going on inside the computer programs that utilize the finite element method. The purpose of this chapter is to supply some of these insights.

Framework Analysis

Finite element methods are a "building block approach" (Sherlock/Kormos, 1973, p. 7) to analysis which has evolved from the techniques of analysis of structural frameworks. The concepts are not especially new, but the large amounts of mathematics involved in the finite element formulations required the development of the high speed digital computer to make the use of finite element theory a practical possibility.

Some of the concepts involved in the finite element method can be illustrated by an example involving a one-dimensional rod or bar element made of an elastic material. This element is defined by the position of its endpoints, which we shall call "nodes." The left node of the element is attached with a pin to an immobile surface while the right node is permitted to move in only the horizontal direction.

For reference, a set of coordinate axes can be imposed on the situation. This allows the dimension in which this element exists to be identified as the "X" dimension, since the element is parallel to the "X" axis and perpendicular to the "Y" and "Z" axes. Forces may be applied to the right hand node of the element in either the positive or negative "X" direction, and are applied on the element's axis. This situation is illustrated in Figure 20.

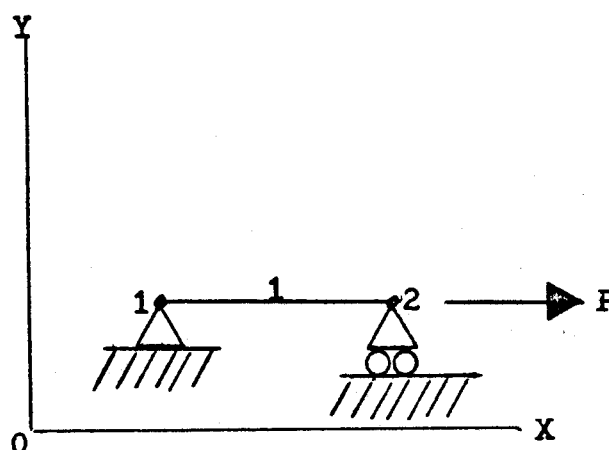


Figure 20

One-Dimensional Elastic Element

Hooke's law states that the extension of a structural member is linearly proportional to the force applied along its length. This can be stated as $e = fP$, where e is the change in length (deformation) of the member, P is the force applied to the member, and f is the flexibility of the member (the extension of the member per unit force) which is determined from the length of the member, the area of the member's cross section, and the modulus of elasticity for the material. The quantity $1/f$ is known as the stiffness coefficient of the member (k). If a

known force (P) were applied to the right hand node of the element in the "X" direction, a deformation would be produced and the right hand node would be displaced in either the positive or negative "X" direction by an amount determined by $e = fP$.

If the desired unknown was the amount of force (P) necessary to produce a given displacement of the right hand node of the above element, it would only be necessary to solve the equations for the force. In this case, $P = e(1/f) = e/f = e \cdot k$.

Now expand this example to include another elastic rod or bar element. It now becomes necessary to number the elements and nodes for clarity. The original element becomes element 1 and the additional element is element 2. The nodes are numbered as shown in Figure 21. Note that element 1 is defined by nodes 1 and 2 while element 2 is defined by nodes 2 and 3.

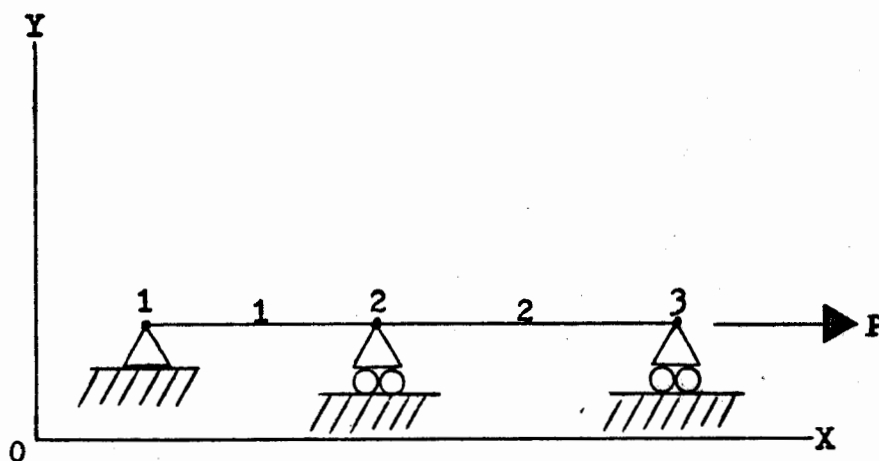


Figure 21

Two One-Dimensional Elastic Elements

Again, node 1 is stationary while nodes 2 and 3 are confined to movement in the positive or negative "X" direction. Since element 1 and

2 need not have the same lengths, cross sections, or even be made of the same material, each element will have its own stiffness coefficient (k). Likewise, each node will have a flexibility coefficient (f) associated with it. If load P_1 is applied in the positive "X" direction at node 2 and load P_2 is applied in the positive "X" direction at node 3, the deflection for node 2 is given as $\delta_2 = f_2(P_1 + P_2)$ and the deflection for node 3 is given as $\delta_3 = f_2(P_1 + P_2) + f_3P_2$.

In the above discussion, a mathematical model has been, in effect, created for the displacements of axially loaded, elastic, one-dimensional elements. If such elements are involved in a structure situated in two or three dimensions, trigonometry can easily be used to resolve forces into their axially applicable components and convert displacements into components on a set of axes.

If the one-dimensional element is found in an assemblage of elements, such as a framework, a number of elements would be found to share end nodes. Figure 22 shows such a framework problem where displacements of the common node are the quantities desired.

In solving such a problem, one would first derive the forces on each element from the external forces applied to the system through the use of trigonometry in a system of simultaneous equations. Next, the resulting deformations for each element would be determined in the same manner as in the examples discussed earlier. Finally, a set of simultaneous equations would be solved to determine the components of the resulting displacement of the common node. In solving these sets of simultaneous equations, it has been found that the use of matrices and matrix algebra allows the problem to be easily expressed and understood. Matrix methods also have the advantage that they are easily programmable

into a digital computer and a problem of practically any size can be accommodated without changing the programming.

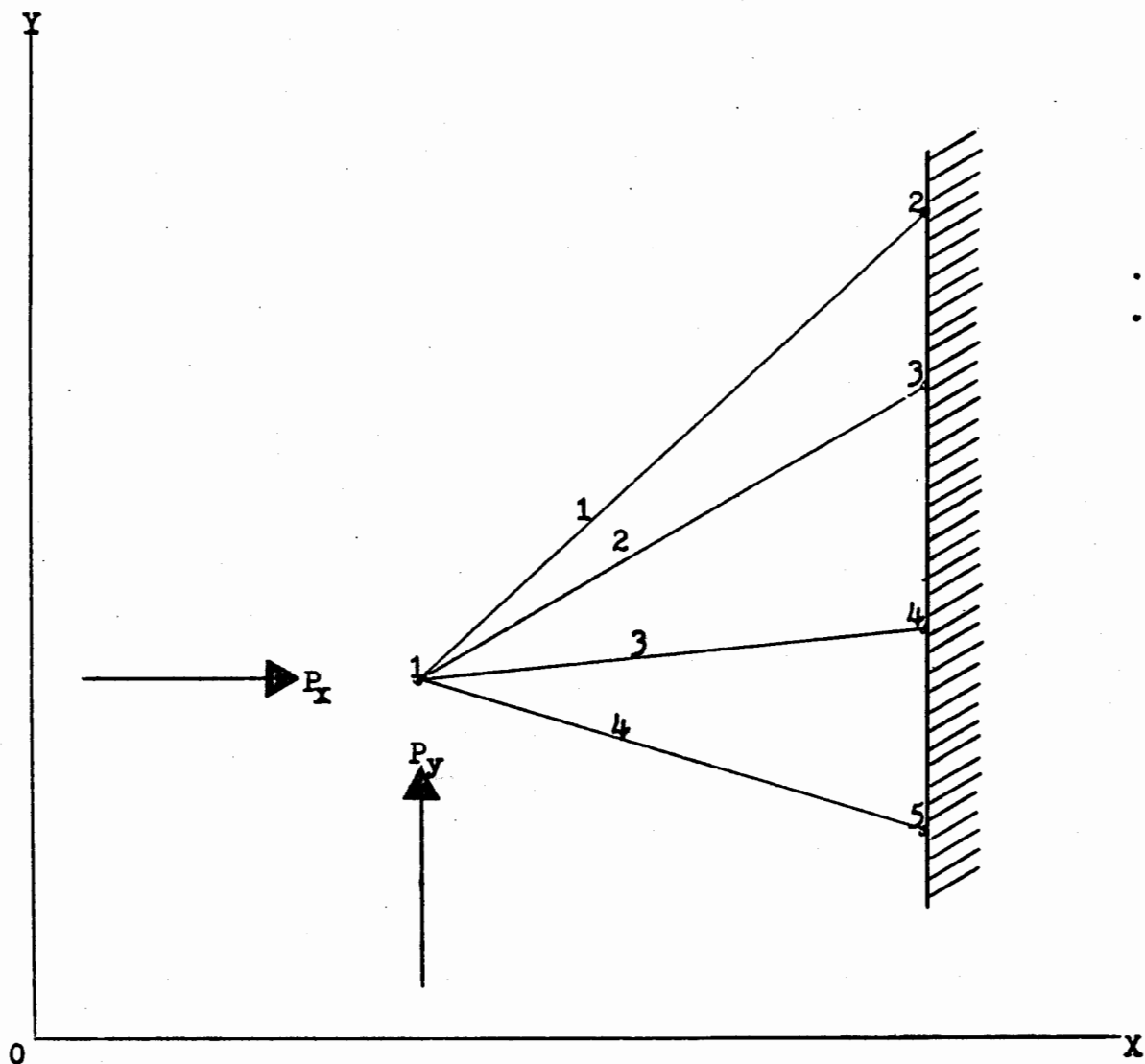


Figure 22

Representation of a Plane
Pin-Jointed Framework

So far in this look at the displacement approach to structural analysis, the emphasis has been on one-dimensional, "rod" or "bar" elements. In this approach, an element made up of an infinite number of

points is modeled in terms of the actions of its end points (nodes) when the element is placed under a load. The techniques of structural analysis made great strides through the use of this technique. However, problems arise when it is desired to model two- or three-dimensional solid objects. The results obtained from attempting to model a solid object by representing it as framework of one-dimensional elements are generally not accurate enough to be useful.

The finite element method has proven itself effective in providing better approximations for problems involving the modeling of two- and three-dimensional continua. The method has had its greatest development in the area of structural problems, but has been found to be applicable in other areas also.

In using the finite element method, a two- or three-dimensional continuum is subdivided into a number of two- or three-dimensional finite elements, the properties of each element are determined, the elements are determined to be connected at a discrete number of nodal points located on their boundaries, the continuum's environmental conditions are applied to the assemblage of elements, the state of the continuum is defined at the nodal points by a set of defining functions, and the results are assembled into a mathematical representation of the original continuum with its environmental conditions applied. Obviously, a number of these points require further explanation.

Figure 23 shows a finite element representation of an area of a continuum under conditions of plane stress. The representation of the triangular elements strongly resembles the representation of a bridge truss in Figure 24. Although the representations are somewhat similar, their purpose is drastically different.

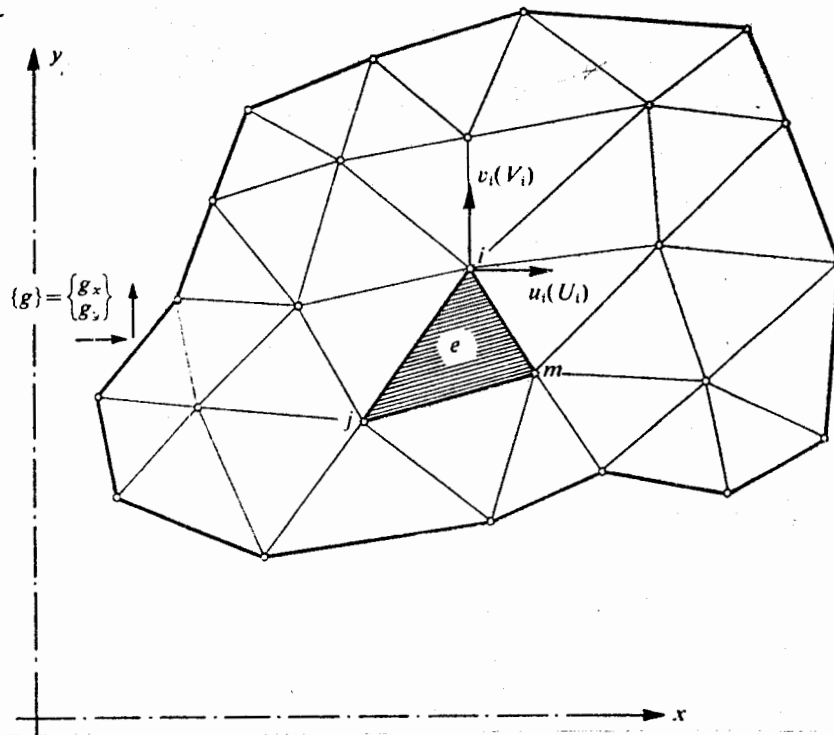


Figure 23

A Plane Stress Region Divided
into Finite Elements
(Zienkiewicz, 1971, p. 18)

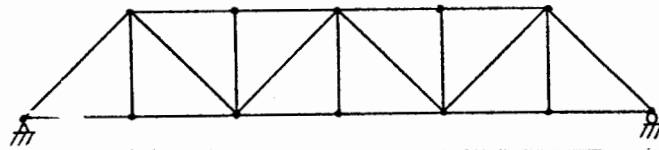


Figure 24

Bridge Truss
(Desai/Abel, 1972, p. 68)

Figure 24 shows an assemblage of one-dimensional rod or beam elements into a truss structure. In this case, the lines in Figure 24 represent the one-dimensional elements themselves. This is not the case with Figure 23.

In the two-dimensional finite element representation of Figure 23, the lines do not represent the elements but serve only to demarcate the

boundaries of the two-dimensional elements. This leads to two possible interpretations of the finite element concept. One interpretation would be that the elements are two- or three-dimensional bodies of their own right. Under this interpretation, each element is essentially a piece of the whole continuum, resulting in the assemblage providing a more natural representation of the properties of the original continuum.

An alternate conceptualization involves considering the elements to be regions of the original continuum:

An alternative but equivalent interpretation of the finite element method derives from the physical continuity of elements in a two- or three-dimensional body. This notion, which is mathematical rather than physical, does not consider the body or structure to be subdivided into separate parts that are reassembled in the analysis procedure. Instead, the continuum is zoned into regions by imaginary lines (two-dimensional bodies) or imaginary planes (three-dimensional bodies) inscribed on the body. Note that no physical separation is envisaged at these lines or planes. Using this concept, we may apply variational procedures to the analysis by assuming a patchwork of solutions or displacement models each of which applies to a single region. (Desai/Abel, 1972, p. 69).

The properties of each element in an assemblage of finite elements are determined by (1) the material in the element, and (2) the shape and mathematical properties of the element type. The properties of the material which bear on the problem usually include the material's elastic modulus (E), Poisson's ratio for the material (ν), the shear modulus of elasticity (G), and the material's density (d). Other properties, such as the material's porosity, thermal conductivity, thermal expansion coefficient, viscosity, etc., are also of importance in certain types of problems.

The mathematical properties of different element types are determined by the mathematician-engineer at the time the element's mathematical

model is developed. The most obvious differences among element types are their shapes and the uses for which each element type is designed.

The shapes of most of the finite elements available are based upon the common engineering shapes, with the triangle being most common. Figure 25 shows some of the elements available for elastic analysis in the NASTRAN program package.

In addition to the geometric shapes, a higher order "isoparametric" type of element has been developed. Isoparametric elements may have curved boundaries and can allow for a distribution of properties (such as stress) within the element. As such, they permit a more accurate representation of a structure while using fewer elements and reducing analysis costs. Figure 26 illustrates a wheel modeled with triangular plate elements and isoparametric elements.

Often times, element formulations are designed for specific types of problems. Some common examples are the specialized elements used for plane stress-plane strain calculations and axisymmetric (solid of revolution) problems. By limiting the uses of such elements to specialized, common conditions, the mathematical formulations can be simplified and the costs of analysis reduced.

Mathematically, geometric elements are modeled in terms of "field" equations. That is, equations are set up which express the desired property (such as stress, strain, porosity, etc.) in terms of a varying field over the element. A minimization process (using the tools of variational calculus) is then carried out and the field property is integrated as being constant over the entire element. This results in a value which may be used in an analogous manner to the flexibility

or stiffness coefficients of the beam elements which were discussed in the earlier part of this chapter.

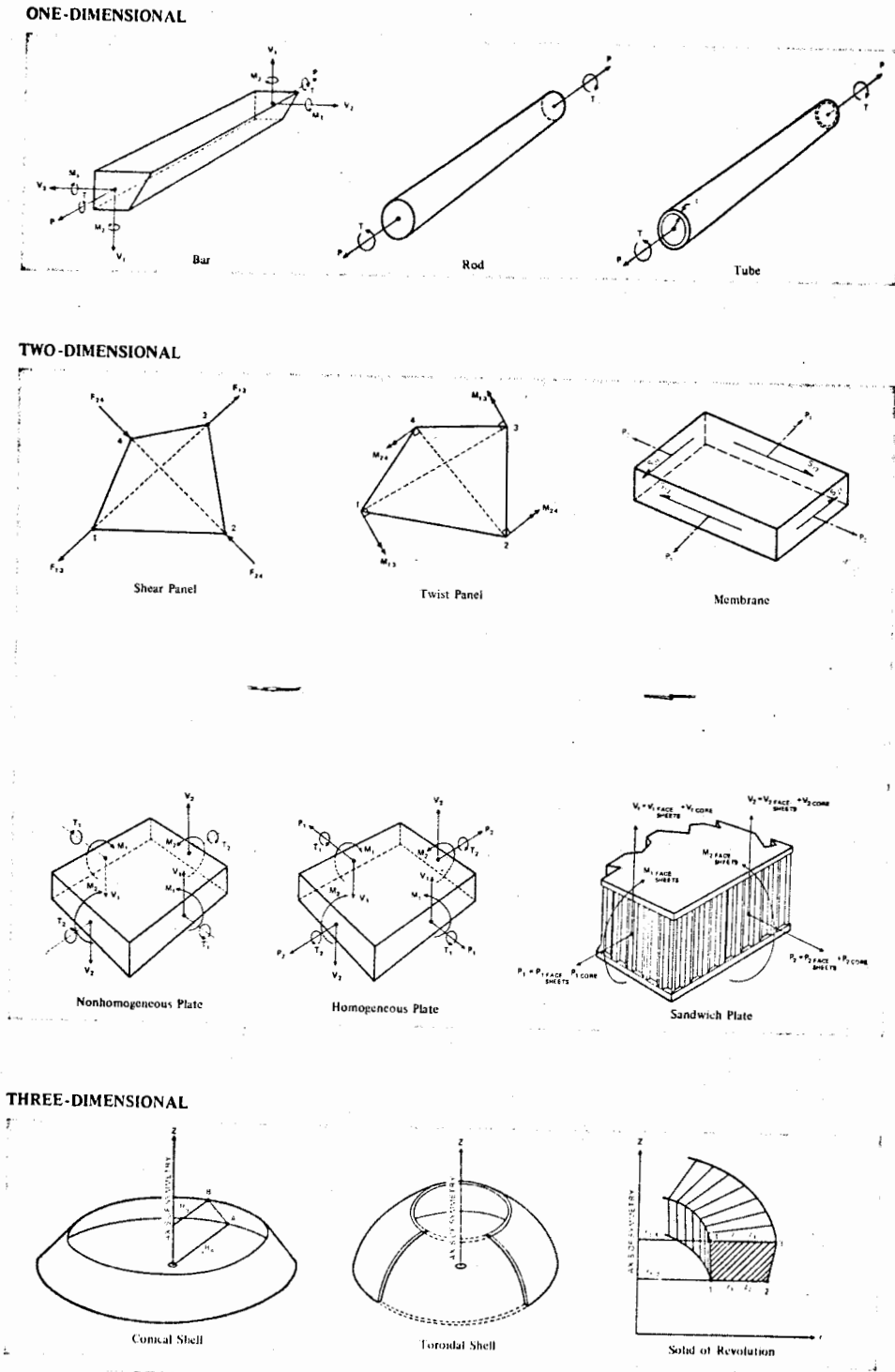


Figure 25

Elastic Elements
(Table 4, NASA SP-260, p. 8-9)

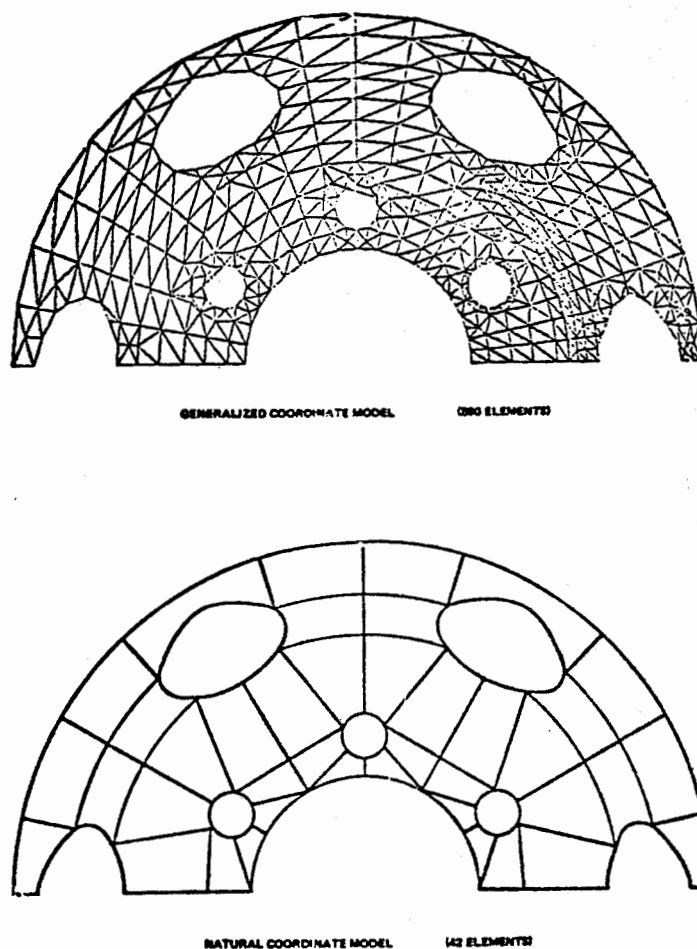


Figure 26

Finite Element Models of Wheel
(Sherlock/Kormos, 1973, p. 10)

When isoparametric elements are formulated, the field properties are not considered to be constant over the element. Instead numerical integration is carried out at a number of points within the element. The number and location of these points are determined by the shape function of the element, which is derived from a "distorted" projection of regular geometric elements.

It should be noted that the usual rectangular coordinate system may not be the optimum for solving the equations associated with finite

elements, especially when these elements are oriented at a certain direction within an assemblage. It appears that the methods which are used to simplify these problems often use tensor analysis to permit simplified coordinate transformations in determining element properties.

A fourth dimension can be added to finite element analysis when the method is used to analyze the dynamic response of a structure or fluid to vibration, pressure, etc. In this type of problem, differential equations are encountered in describing element properties. These problems are solved with applications of linear algebra involving eigenvalue analysis.

In any finite element analysis, the element's nodal points are of importance. The external nodes are usually considered to be located at the intersections of the boundaries of the element. In addition, secondary external nodes may be located along the sides of the element. Finally, it is common to locate internal nodes inside triangular elements.

The importance of the element nodes is two-fold. First, the structure in question is usually defined in terms of the location of the element nodes with respect to a set of rectangular coordinates. And second, the analysis and the results of the analysis are carried out and presented with respect to these nodal points. During structural analysis, external forces on the structure are represented as forces on the element nodes and any resulting displacements are represented as nodal displacements. In other types of problems, the nodes determine the locations in the problem for which solutions will be found.

Throughout the analysis procedure, the properties of the elements and the actions of the element nodes are manipulated in matrices by the rules of matrix algebra. This allows the equations defining the

actions of the nodes to be solved simultaneously, permitting the properties of one element to affect all the other elements in the assemblage while being affected by these other elements at the same time.

This chapter has attempted to list some of the processes involved in a finite element analysis. In this short space, it is impossible to present the entire theory behind the finite element method. In addition, the mathematics involved is extremely complex to the uninitiated person and requires a considerable amount of study before it is possible to become facile with its theory or use.

Chapter 7

USING A FINITE ELEMENT METHOD COMPUTER PROGRAM

Introduction

In using the finite element method to analyze an engineering structure or component, a precise knowledge of the workings of the method is not necessary to prepare data and make use of the results. This is due to the capabilities of the computer programs which are available, packaged so as to insulate the user from the complexities of the mechanisms of the method.

To the user of finite element program packages, a sound knowledge of classical mechanics (strength of materials, metalurgy, statics, dynamics), mathematics (especially trigonometry and analytic geometry), experience in using computers, and experience with the behavior of structures in general, are the most useful types of knowledge to have. In his book, Ural (1973, p. 238) gives the following warning:

I would like to emphasize the need of the engineer to thoroughly understand all relevant theory and behavior of structures before using any of the (finite element) programs... The familiarity with the basic concepts of mathematics, computers, mechanics, materials, and elasticity is necessary to avoid all potentially grave errors.

Inputting Data

The first step in using a finite element computer program involves preparing and defining the problem in a manner which is both

acceptable to the computer program and will result in accurate and useful output data. This step has two major sections:

- a. Idealization of the problem with logical mesh generation.
- b. Preparation of the input information to be used by a particular computer program to obtain the results of the analysis. (Ural, 1973, p. 237).

The first part of this step involves the knowledge and expertise of the experienced engineer. His design must be defined graphically and scale drawings of the components to be analyzed must be prepared. Next, a drawing of the finite element model of each component must be prepared. This involves the generation of the element mesh.

In generating a finite element mesh, three factors must be considered: the type and shape of each element, the sizes of the elements, and the placement and orientation of the elements. Since different elements have different mathematical properties, the element type (and resulting shape) must be selected to closely match the properties of the structure being modeled. In addition, the element's aspect ratio (ratio between an element's longest and shortest dimension) must be taken into consideration, since more accurate results are obtained with elements whose aspect ratios are 1:1. The size of the elements used will largely determine the accuracy of the resulting finite element model and the cost of the analysis, smaller elements providing more accurate results over areas of stress concentrations and variations while increasing the computer time necessary for the analysis. Finally, the placement and orientation of elements in the model will affect the accuracy of the stress results from the analysis as well as determining the amount of work necessary to interpret the analysis results. In cases where regular shapes and uniform stress variations are encountered, the elements mesh can often be generated by a suitable computer program.

Before the computer analysis can be carried out, the conditions to which the modeled component or structure will be subjected must be determined. These conditions are usually considered in the form of loading data. Clark, Barnhart, and Hayes (1974, p. 5) present some of the considerations involved in generating loading data for a finite element model of a backhoe component:

...a combination of past field test data and analytical calculations is used to establish loading conditions which will reproduce maximum operating stresses in the machine. But since the part being computer-modeled is isolated from the machine, overall machine loadings must be transformed into resultant loads acting on the modeled part. These total resultant loads are then solved into components for application to the various connection locations of the model to the machine. These must create an equilibrium condition which is restrained, called a "free-free" model... Our present solution for determining loading on the modeled component is to create a beam or stick model of the complete machine. The (finite element) program is then used to calculate the resultant loads on the component from the selected static input loads to the machine. The beam model is prepared with its major axis as the same axis of the component model. The computer load solutions from the beam model are then directly applicable for the (finite element) model component without any conversion of directions. Additional loads can be applied to the component model at some small additional time requirement and computer cost.

After the finite element model has been defined and the appropriate loading conditions have been determined, the task of preparing the input data for the computer remains. The first consists of converting the model into coordinate nodal points on a set of axes and assigning numbers or letters to these nodal points according to a system designed to minimize the amount of storage required for the problem in the computer memory. Next, the elements are defined in terms of their nodal points, and each element is assigned a number or letter name. The properties of the materials used in the structure

being modeled are specified and the loading conditions are input into the program.

Most of the data preparation work can easily be done by a trained technician familiar with analytic geometry. Since in a large problem this work can be lengthy and time consuming, the data preparation technician must be patient in his work. He must also be accurate, as a single mistake (such as a space left out of an instruction or a misspelled command word) can result in a costly analysis producing useless or inaccurate output.

In converting the finite element model into coordinate points, a digitizer and the appropriate associated software can often be put to good use. By using a digitizer, time can be saved and technician fatigue can be reduced, resulting in fewer input errors.

Output Results

Following the analysis of a structure or machine component by a finite element program, the results of the analysis must be displayed in a readable and useful form. The printout produced by most finite element programs "is a large volume of paper which indicates stress level and stress direction for each element." (Clark/Barnhart/Hayes, 1974, p. 6). In addition, the output may show nodal deflections, reaction forces, natural frequencies, and other information. The problem which arises in interpreting this information is caused by the tabular form of the printed output.

Manual display and interpretation of finite element program output can be very time consuming and tedious. Some finite element programs can supply graphical output of model construction and results

of analysis. Such graphical output can be presented by means of the line printer, a cathode ray tube (CRT) terminal, or by an automatic plotter.

Since line printers are designed to print tabular data, their usefulness as graphical output devices is usually limited to printing simple graphs and straight line representations of structures. CRT terminals are often used to error check input data or to view intermediate results. Additional equipment is necessary to produce permanent "hard" copy.

Automatic plotters combine the advantages of the line printer and the CRT, producing curved as well as straight lines on paper copies. In addition to producing graphic representations of finite element models, plotters are commonly used to display stresses in these models. Two methods of displaying this information include: (1) plotting contours of stress levels over all or part of the area of the system; and (2) drawing the principal stress vectors for each element in the appropriate directions. (Zienkiewicz, 1971, p. 471).

Conclusion

Each finite element program differs in its construction and its input requirements and output capabilities. Although the formulation of the finite element method of analysis insures some similarities among program packages, each program has its own irregularities. Experience with a particular program and a thorough study of its user's manual are necessary prerequisites to using a finite element computer program.

Chapter 8

A SMALL SAMPLE PROBLEM

In order to illustrate the use of the finite element method, a simple plane stress problem was formulated. The problem was first solved using a standard handbook formula, then with a simple special purpose finite element computer program, and finally with the ICES STRUDL-II finite element capability.

The Problem

A simple beam deflection problem was selected with the primary unknown quantity desired being the mid-point deflection of the beam with a static load. The beam is shown in Figure 27.

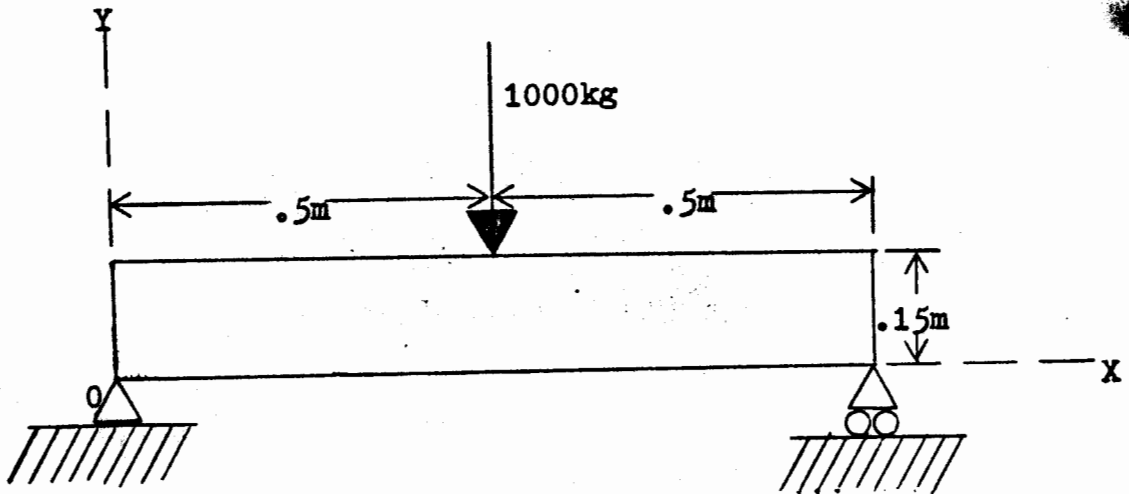


Figure 27

Representation of "Simple Beam Problem"

The beam was represented as being of rectangular cross section, 100 cm (1 m) long, 15 cm wide, and 5 cm thick. Its extreme ends are supported in the vertical direction at the lower corners and its lower left corner is also fixed horizontally.

SAE 950, high strength-low alloy steel was selected as the material for the beam. This steel has the following properties: modulus of elasticity (E) = 2.10921×10^6 kg/cm²; Poisson's ratio (ν) = 0.303; density (d) = 0.0078 kg/cm³.

A static load was assumed to be applied to the center of this beam. This load had a magnitude of 10^3 kg.

Solution by Handbook Formulas

As a check on the solutions to be produced by the finite element computer programs, a calculation for the deflection at the mid-point of the previously described beam was carried out using formulas found in a commonly used engineering handbook. (Oberg/Jones, 1971).

The first step was to calculate the moment of inertia (I) for this rectangular cross section beam. This was found using the following formula:

$$I = \frac{bd^3}{12} \quad \text{where: } \begin{array}{l} b = \text{base dimension of rectangle} \\ d = \text{depth (height) of rectangle} \end{array}$$

$$\text{Substituting: } I = \frac{5 \text{ cm } (15 \text{ cm})^3}{12} = 1406.25 \text{ cm}^4.$$

Next, the distributed load on the beam, caused by the weight of the material in the beam, was calculated. The volume of material in the beam was found to be 7500 cm³, resulting in a total load of 58.5 kg.

The resulting deflection at the center of the constant cross section beam from this distributed load is given by the following formula:

$$D_d = \frac{5 W_d L^3}{384 EI}$$

where: W_d = total distributed load
 L = length of beam between supports
 E = modulus of elasticity
 I = moment of inertia of cross section

$$\text{Substituting: } D = \frac{5 \cdot 58.5 \text{ kg} (100 \text{ cm})^3}{384 \cdot 2.10921 \times 10^6 \text{ kg/cm}^2 \cdot 1406.25 \text{ cm}^4}$$

$$= 0.00025681 \text{ cm}$$

The deflection resulting from the concentrated load at the center of the beam was then calculated from the formula:

$$D_c = \frac{W_c L^3}{48 EI}$$

where: W_c = total concentrated vertical load
 L = length of beam between supports
 E = modulus of elasticity
 I = moment of inertia of cross section

$$\text{Substituting: } D_c = \frac{10^3 \text{ kg} (100 \text{ cm})^3}{48 \cdot 2.10921 \times 10^6 \text{ kg/cm}^2 \cdot 1406.25 \text{ cm}^4}$$

$$= 0.00702387 \text{ cm}$$

The total deflection (D) at the mid-point of the beam was found by adding the deflection resulting from the weight of the beam and the deflection resulting from the concentrated load at the center of the beam: $D = D_d + D_c$. This total deflection was found to be 0.00728068 cm.

The Finite Element Model

The finite element idealization of this problem is shown in Figure 28.

Since both of the finite element computer programs being used were capable of working with rectangular finite elements, the beam was represented as an assemblage of square elements, 5 centimeters on a side and 5 centimeters thick. Sixty of these elements were required to model the beam, and eighty-four nodal points were necessary to define the elements.

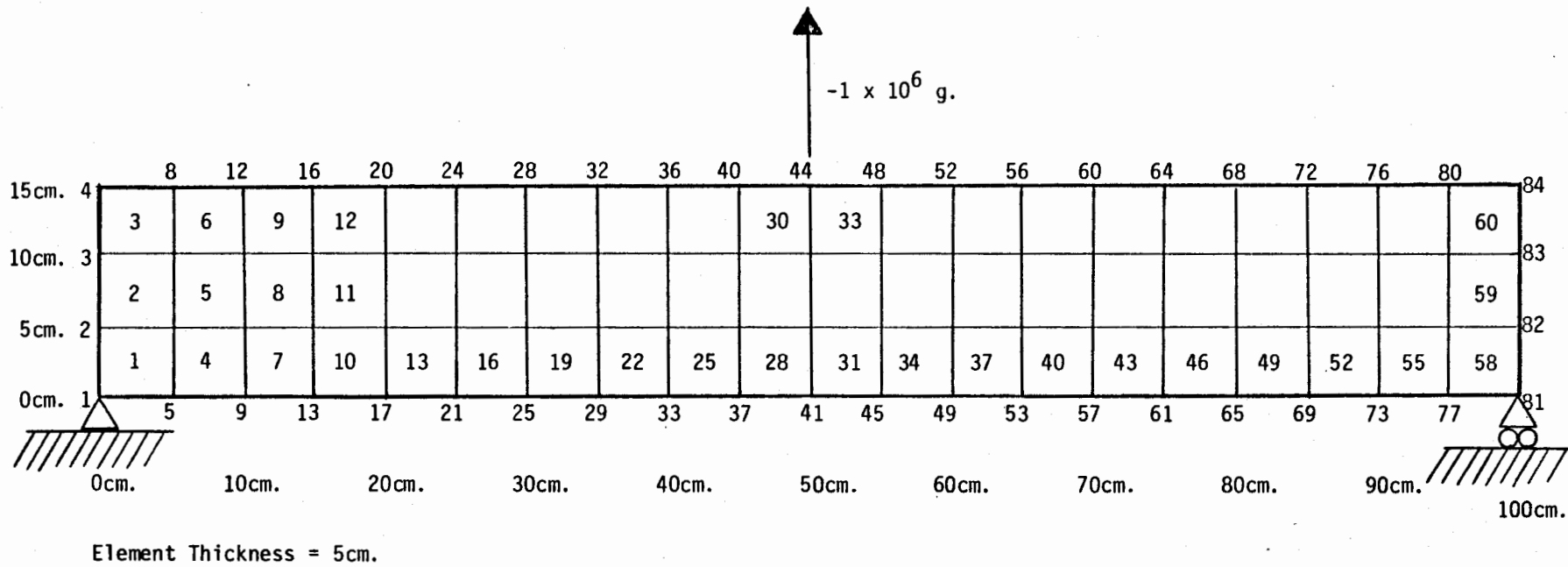


Figure 28

Finite Element Idealization of
"Simple Beam Problem"

The lower left hand node (labeled as node 1 in Figure 28) was considered to be located at the zero point on a rectangular set of X-Y axes, and this node was defined as a support point in both the X and Y directions. Node 81 was defined as another support and was fixed in the Y direction, but allowed to move in the X direction. Node 44 was the load point where the load of 1000 kilograms was applied in the negative Y direction. Node 44 was also used as the point of maximum deflection in interpreting the results.

Desai/Abel "INTROFEM" Program

The previously described problem was first run using the special purpose, plane stress/plane strain computer program which is found in Appendix I of Introduction to the Finite Element Method by Desai and Abel. (Desai/Abel, 1972, p. 439-462). A copy of this program was placed in the University of Northern Iowa's program library at the Regional Computer Center in Iowa City, under the keyword "INTROFEM" (INTROduction to the Finite Element Method). A copy of this program and materials concerning its use are included in the appendix to this paper.

The INTROFEM program will handle plane stress or plane strain problems, using finite elements made up of four constant strain triangle elements (4-CST elements). Therefore, it can use either rectangular or triangular finite elements in the problem idealization. This version of the program has sufficient storage for 120 nodes, 100 elements, 10 material types, and 20 externally applied distributed loads (surface tractions).

A listing of the input to this program (on cards) is included at the end of this chapter. Much of the input data to the program is in

exponential form and is input using a fixed "E" format. Since the input data to this program is manipulated using fixed formats, care must be used in preparing the data and punching it onto cards.

The output from the INTROFEM program is reproduced at the end of this chapter. A summary of the input information is included in the input tables. Output Table 1 shows the displacements of the nodal points of the model while Output Table 2 shows the stresses on the elements, calculated at their centroids. This table includes the components of the plane stresses on the elements and the components of the shear stresses on the elements.

Referring to the nodal displacements for node 44 in Output Table 1, the Y displacement is found to be 0.00726120 cm in the negative Y direction. This is in excellent agreement with the results found by using the handbook formulas.

ICES STRUDL-II Program

The simple example problem was then run using the general purpose finite element capabilities of the Integrated Civil Engineering System (ICES) Structural Design Language-version II (STRUDL-II). This programming system was developed at the Massachusetts Institute of Technology and is available at the Regional Computer Center at Iowa City. A short user's manual covering the plane stress/plain strain finite element capabilities of this system is included in the appendix to this paper.

In addition to plane stress/plane strain problems, STRUDL-II will handle structural problems concerning frameworks, shells, three-dimensional elements, reinforced concrete, and other types of problems. For the example, a plane strain rectangle (PSR) finite element was

selected. STRUDL-II is capable of handling problems of any size, limited by the computer time and cost required to complete the problem.

A list of the input data cards used in this problem is included at the end of this chapter. The data is read into the program using "free field" formats and card punching is not critical. However, spaces must be inserted between each data component and program commands must be inserted before the corresponding data cards.

The output from the STRUDL-II solution is included at the end of this chapter. Since STRUDL-II is a command oriented system, the types and format of the output may be somewhat varied by the user. The output tables are labeled by the program and are self-explanatory.

Referring to the nodal displacements for node 44 in the Resultant Joint Displacements - Free Joints table, the Y displacement is found to be 0.0072172 cm in the negative Y direction. This is in very good agreement with the results found with the handbook formulas and the Desai/Abel "INTROFEM" program.

UNIVERSITY OF IOWA...START JOB 573.....2.22.55 PM 8 APR 75.....HASPLIST...80-80 LISTING.....UNIVERSITY OF IOWA
UNIVERSITY OF IOWA...START JOB 573.....2.22.55 PM 8 APR 75.....HASPLIST...80-80 LISTING.....UNIVERSITY OF IOWA
UNIVERSITY OF IOWA...START JOB 573.....2.22.55 PM 8 APR 75.....HASPLIST...80-80 LISTING.....UNIVERSITY OF IOWA

//HASPLIST JOB (-----,14.....66),*80-80 LISTING*

JOB 573

CARDS REQUIRED FOR 'INTROFEM' 'SIMPLE BEAM PROBLEM' SOLUTION

----- JOB ACCOUNTING CARD -----

//INTROFEM JOB (-----),*RICH RYNER',CLASS=904SGLEVEL=(1,1)

----- JCL (JOB CONTROL LANGUAGE) CARDS: LIBRARY PROGRAM -----

//JOBLIB DD DSN=H2037102.PROGRAMS,UNIT=2314,VOL=SER=IOWA05,DISP=SHR
// EXEC PGM=INTROFEM
//FT01F001 DD UNIT=SYSDA,SPACE=(CYL,(1,1))
//FT06F001 DD SYSOUT=A
//FT05F001 DD *

----- PROBLEM DATA CARDS (FORMATTED) -----

1	SIMPLE BEAM PROBLEM - METRIC UNITS- GRAM, CENTIMEIER				RICH RYNER
84	60	1	0	2	1
2.109E+09	3.030E-01	7.800E+00	5.000E+00		
1	3				
4	0	0.000E+00	1.500E+01		
5	0	5.000E+00			
8	0	5.000E+00	1.500E+01		
9	0	1.000E+01			
12	0	1.000E+01	1.500E+01		
13	0	1.500E+01			
16	0	1.500E+01	1.500E+01		
17	0	2.000E+01			
20	0	2.000E+01	1.500E+01		
21	0	2.500E+01			
24	0	2.500E+01	1.500E+01		
25	0	3.000E+01			
28	0	3.000E+01	1.500E+01		
29	0	3.500E+01			
32	0	3.500E+01	1.500E+01		
33	0	4.000E+01			
36	0	4.000E+01	1.500E+01		
37	0	4.500E+01			
40	0	4.500E+01	1.500E+01		
41	0	5.000E+01			
44	0	5.000E+01	1.500E+01	0.000E+01-1.000E+06	
45	0	5.500E+01			
48	0	5.500E+01	1.500E+01		
49	0	6.000E+01			
52	0	6.000E+01	1.500E+01		
53	0	6.500E+01			
56	0	6.500E+01	1.500E+01		
57	0	7.000E+01			

"INTROFEM" Input:
"Simple Beam Problem"

60	0	7.000E+01	1.500E+01		
61	0	7.500E+01			
64	0	7.500E+01	1.500E+01		
65	0	8.000E+01			
6A	0	8.000E+01	1.500E+01		
69	0	8.500E+01			
72	0	8.500E+01	1.500E+01		
73	0	9.000E+01			
76	0	9.000E+01	1.500E+01		
77	0	9.500E+01			
80	0	9.500E+01	1.500E+01		
81	2	1.000E+02			
84	0	1.000E+02	1.500E+01		
1	1	5	6	2	1
4	5	9	10	6	1
7	9	13	14	10	1
10	13	17	18	14	1
13	17	21	22	18	1
16	21	25	26	22	1
19	25	29	30	26	1
22	29	33	34	30	1
25	33	37	38	34	1
28	37	41	42	38	1
31	41	45	46	42	1
34	45	49	50	46	1
37	49	53	54	50	1
40	53	57	58	54	1
43	57	61	62	58	1
46	61	65	66	62	1
49	65	69	70	66	1
52	69	73	74	70	1
55	73	77	78	74	1
58	77	81	82	78	1
60	79	83	84	80	1

----- BLANK CARD INDICATES END OF INPUT DATA -----

----- JCL END CARD -----

/*

----- END OF 'INTROFEM' INPUT DECK -----

UNIVERSITY OF IOWA....START JOB 725.....12.19.14 PM 13 FEB 75.....INTROFEM....RICH RYNER.....UNIVERSITY OF IOW
UNIVERSITY OF IOWA....START JOB 725.....12.19.14 PM 13 FEB 75.....INTROFEM....RICH RYNER.....UNIVERSITY OF IOW
UNIVERSITY OF IOWA....START JOB 725.....12.19.14 PM 13 FEB 75.....INTROFEM....RICH RYNER.....UNIVERSITY OF IOW

H A S P S Y S T E M L O G

\$ 12.18.54 JOB 725 -- INTROFEM -- BEGINNING EXEC - INIT 5 - CLASS R
N 12.19.14 JOB 725 END EXECUTION.

//INTROFEM JOB (-----,9), 'RICH RYNER', CLASS=R, MSGLEVEL=(1,1) JOB 725
//JOB LIB DD DSN=H2037102.PROGRAMS, UNIT=2314, VOL=SER=IOWA05, DISP=SHR
// EXEC PGM=INTROFEM

//FT01F001 DD UNIT=SYSDA, SPACE=(CYL,(1,1))
//FT06F001 DD SYSOUT=A
//FT05F001 DD *
//SYSIN DD *

GENERATED STATEMENT

***** ALLOCATION FOR INTROFEM. (STEP 001) ***** EXECUTOR V3.0
JOB LIB 130 2314 DISK IOWA05 H2037102.PROGRAMS
FT01F001 132 2314 DISK IOWA07 SYS75044.T121854.RV177.INTROFEM.R0000001
FT06F001 328 1403 PRT SYS75044.T121854.RV177.INTROFEM.R0000002
FT05F001 303 2540 RDR SYS75044.T121854.RV177.INTROFEM.R0000003
SYSIN 304 2540 RDR SYS75044.T121854.RV177.INTROFEM.R0000004

ACCTG -- 5.42 SEC. CPU, 11.37 SEC. WAIT, DAC= 121, HWM= 98K

"INTROFEM" Output:
"Simple Beam Problem"

PROBLEM 1.. SIMPLE BEAM PROBLEM - METRIC UNITS- GRAM, CENTIMETER RICH RYNER

INPUT TABLE 1.. BASIC PARAMETERS

NUMBER OF NODAL POINTS.....	84
NUMBER OF ELEMENTS.....	60
NUMBER OF DIFFERENT MATERIALS	1
NUMBER OF SURFACE LOAD CARDS.....	0
1 = PLANE STRAIN, 2 = PLANE STRESS...	2
BODY FORCES(1 = IN -Y DIREC., 0 = NONE)	1

INPUT TABLE 2.. MATERIAL PROPERTIES

MATERIAL NUMBER	MODULUS OF ELASTICITY	POISSON'S RATIO	MATERIAL DENSITY	MATERIAL THICKNESS
1	0.2109E 10	0.3030E 00	0.7800E 01	0.5000E 01

INPUT TABLE 3.. NODAL POINT DATA

NODAL POINT	TYPE	X	Y	X-DISP. OR LOAD	Y-DISP. OR LOAD
1	3	0.0	0.0	0.0	0.0
2	0	0.0	0.5000E 01	0.0	0.0
3	0	0.0	0.1000E 02	0.0	0.0
4	0	0.0	0.1500E 02	0.0	0.0
5	0	0.5000E 01	0.0	0.0	0.0
6	0	0.5000E 01	0.5000E 01	0.0	0.0
7	0	0.5000E 01	0.1000E 02	0.0	0.0
8	0	0.5000E 01	0.1500E 02	0.0	0.0
9	0	0.1000E 02	0.0	0.0	0.0
10	0	0.1000E 02	0.5000E 01	0.0	0.0
11	0	0.1000E 02	0.1000E 02	0.0	0.0
12	0	0.1000E 02	0.1500E 02	0.0	0.0
13	0	0.1500E 02	0.0	0.0	0.0
14	0	0.1500E 02	0.5000E 01	0.0	0.0
15	0	0.1500E 02	0.1000E 02	0.0	0.0
16	0	0.1500E 02	0.1500E 02	0.0	0.0
17	0	0.2000E 02	0.0	0.0	0.0
18	0	0.2000E 02	0.5000E 01	0.0	0.0
19	0	0.2000E 02	0.1000E 02	0.0	0.0
20	0	0.2000E 02	0.1500E 02	0.0	0.0
21	0	0.2500E 02	0.0	0.0	0.0
22	0	0.2500E 02	0.5000E 01	0.0	0.0
23	0	0.2500E 02	0.1000E 02	0.0	0.0
24	0	0.2500E 02	0.1500E 02	0.0	0.0
25	0	0.3000E 02	0.0	0.0	0.0
26	0	0.3000E 02	0.5000E 01	0.0	0.0
27	0	0.3000E 02	0.1000E 02	0.0	0.0
28	0	0.3000E 02	0.1500E 02	0.0	0.0
29	0	0.3500E 02	0.0	0.0	0.0
30	0	0.3500E 02	0.5000E 01	0.0	0.0
31	0	0.3500E 02	0.1000E 02	0.0	0.0
32	0	0.3500E 02	0.1500E 02	0.0	0.0
33	0	0.4000E 02	0.0	0.0	0.0
34	0	0.4000E 02	0.5000E 01	0.0	0.0
35	0	0.4000E 02	0.1000E 02	0.0	0.0
36	0	0.4000E 02	0.1500E 02	0.0	0.0
37	0	0.4500E 02	0.0	0.0	0.0
38	0	0.4500E 02	0.5000E 01	0.0	0.0
39	0	0.4500E 02	0.1000E 02	0.0	0.0
40	0	0.4500E 02	0.1500E 02	0.0	0.0
41	0	0.5000E 02	0.0	0.0	0.0
42	0	0.5000E 02	0.5000E 01	0.0	0.0
43	0	0.5000E 02	0.1000E 02	0.0	0.0
44	0	0.5000E 02	0.1500E 02	0.0	-0.1000E 07
45	0	0.5500E 02	0.0	0.0	0.0
46	0	0.5500E 02	0.5000E 01	0.0	0.0
47	0	0.5500E 02	0.1000E 02	0.0	0.0
48	0	0.5500E 02	0.1500E 02	0.0	0.0
49	0	0.6000E 02	0.0	0.0	0.0
50	0	0.6000E 02	0.5000E 01	0.0	0.0
51	0	0.6000E 02	0.1000E 02	0.0	0.0
52	0	0.6000E 02	0.1500E 02	0.0	0.0
53	0	0.6500E 02	0.0	0.0	0.0
54	0	0.6500E 02	0.5000E 01	0.0	0.0
55	0	0.6500E 02	0.1000E 02	0.0	0.0
56	0	0.6500E 02	0.1500E 02	0.0	0.0
57	0	0.7000E 02	0.0	0.0	0.0

58	0	0.7000E 02	0.5000E 01	0.0	0.0
59	0	0.7000E 02	0.1000E 02	0.0	0.0
60	0	0.7000E 02	0.1500E 02	0.0	0.0
61	0	0.7500E 02	0.0	0.0	0.0
62	0	0.7500E 02	0.5000E 01	0.0	0.0
63	0	0.7500E 02	0.1000E 02	0.0	0.0
64	0	0.7500E 02	0.1500E 02	0.0	0.0
65	0	0.8000E 02	0.0	0.0	0.0
66	0	0.8000E 02	0.5000E 01	0.0	0.0
67	0	0.8000E 02	0.1000E 02	0.0	0.0
68	0	0.8000E 02	0.1500E 02	0.0	0.0
69	0	0.8500E 02	0.0	0.0	0.0
70	0	0.8500E 02	0.5000E 01	0.0	0.0
71	0	0.8500E 02	0.1000E 02	0.0	0.0
72	0	0.8500E 02	0.1500E 02	0.0	0.0
73	0	0.9000E 02	0.0	0.0	0.0
74	0	0.9000E 02	0.5000E 01	0.0	0.0
75	0	0.9000E 02	0.1000E 02	0.0	0.0
76	0	0.9000E 02	0.1500E 02	0.0	0.0
77	0	0.9500E 02	0.0	0.0	0.0
78	0	0.9500E 02	0.5000E 01	0.0	0.0
79	0	0.9500E 02	0.1000E 02	0.0	0.0
80	0	0.9500E 02	0.1500E 02	0.0	0.0
81	2	0.1000E 03	0.0	0.0	0.0
82	0	0.1000E 03	0.5000E 01	0.0	0.0
83	0	0.1000E 03	0.1000E 02	0.0	0.0
84	0	0.1000E 03	0.1500E 02	0.0	0.0

INPUT TABLE 4.. ELEMENT DATA

ELEMENT	GLOBAL INDICES OF ELEMENT NODES				MATERIAL
	1	2	3	4	
1	1	5	6	2	1
2	2	6	7	3	1
3	3	7	8	4	1
4	5	9	10	6	1
5	6	10	11	7	1
6	7	11	12	8	1
7	9	13	14	10	1
8	10	14	15	11	1
9	11	15	16	12	1
10	13	17	18	14	1
11	14	18	19	15	1
12	15	19	20	16	1
13	17	21	22	18	1
14	18	22	23	19	1
15	19	23	24	20	1
16	21	25	26	22	1
17	22	26	27	23	1
18	23	27	28	24	1
19	25	29	30	26	1
20	26	30	31	27	1
21	27	31	32	28	1
22	29	33	34	30	1
23	30	34	35	31	1
24	31	35	36	32	1
25	33	37	38	34	1
26	34	38	39	35	1
27	35	39	40	36	1
28	37	41	42	38	1
29	38	42	43	39	1
30	39	43	44	40	1
31	41	45	46	42	1
32	42	46	47	43	1
33	43	47	48	44	1
34	45	49	50	46	1
35	46	50	51	47	1
36	47	51	52	48	1
37	49	53	54	50	1
38	50	54	55	51	1
39	51	55	56	52	1
40	53	57	58	54	1
41	54	58	59	55	1
42	55	59	60	56	1
43	57	61	62	58	1
44	58	62	63	59	1
45	59	63	64	60	1
46	61	65	66	62	1
47	62	66	67	63	1
48	63	67	68	64	1
49	65	69	70	66	1
50	66	70	71	67	1
51	67	71	72	68	1
52	69	73	74	70	1
53	70	74	75	71	1
54	71	75	76	72	1
55	73	77	78	74	1
56	74	78	79	75	1
57	75	79	80	76	1

58	77	81	82	78	1
59	78	82	83	79	1
60	79	83	84	80	1

OUTPUT TABLE 1.. NODAL DISPLACEMENTS

NODE	U = X-DISP.	V = Y-DISP.
1	0.0	0.0
2	0.10672756E-02	-0.10458079E-03
3	0.20513460E-02	-0.14894456E-03
4	0.30415705E-02	-0.15710240E-03
5	0.42699045E-04	-0.11561494E-02
6	0.10764620E-02	-0.11568326E-02
7	0.20491881E-02	-0.11622270E-02
8	0.30339253E-02	-0.11612980E-02
9	0.10214932E-03	-0.21779519E-02
10	0.10820562E-02	-0.21898430E-02
11	0.20315887E-02	-0.21850697E-02
12	0.29993153E-02	-0.21723905E-02
13	0.1A221978E-03	-0.31551288E-02
14	0.11048713E-02	-0.31728665E-02
15	0.20037973E-02	-0.31712826E-02
16	0.29255550E-02	-0.31521148E-02
17	0.28914725E-03	-0.40711500E-02
18	0.11399270E-02	-0.40955642E-02
19	0.19678755E-02	-0.40953010E-02
20	0.28192408E-02	-0.40706024E-02
21	0.42569195E-03	-0.49068518E-02
22	0.11855070E-02	-0.49374625E-02
23	0.19223758E-02	-0.49374774E-02
24	0.26823226E-02	-0.49068145E-02
25	0.59230044E-03	-0.56416914E-02
26	0.12410427E-02	-0.56782775E-02
27	0.18668463E-02	-0.56780279E-02
28	0.25155200E-02	-0.56412593E-02
29	0.78927865E-03	-0.62557347E-02
30	0.13061306E-02	-0.62980317E-02
31	0.1A009683E-02	-0.62970817E-02
32	0.23194242E-02	-0.62539764E-02
33	0.10181593E-02	-0.67275502E-02
34	0.13800042E-02	-0.67767911E-02
35	0.17240213E-02	-0.67748874E-02
36	0.20958851E-02	-0.67267902E-02
37	0.12785613E-02	-0.70308372E-02
38	0.14652626E-02	-0.70892125E-02
39	0.16351100E-02	-0.70998147E-02
40	0.1A461959E-02	-0.70429705E-02
41	0.15611264E-02	-0.71368888E-02
42	0.15611304E-02	-0.72010048E-02
43	0.15611367E-02	-0.72335713E-02
44	0.15611460E-02	-0.72612055E-02
45	0.1A436974E-02	-0.70308745E-02
46	0.16569973E-02	-0.70892572E-02
47	0.14871587E-02	-0.70998520E-02
48	0.12761003E-02	-0.70429966E-02
49	0.21041005E-02	-0.67276210E-02
50	0.17422566E-02	-0.67768618E-02
51	0.13982467E-02	-0.67749619E-02
52	0.10264134E-02	-0.67268685E-02
53	0.23329877E-02	-0.62558427E-02
54	0.1A161363E-02	-0.62981397E-02
55	0.13213039E-02	-0.62971972E-02
56	0.80286874E-03	-0.62540993E-02
57	0.25299774E-02	-0.56418292E-02
58	0.1A812309E-02	-0.56784227E-02

59	0.12554261E-02	-0.56781732E-02
60	0.60676550E-03	-0.56414045E-02
61	0.26966007E-02	-0.49070120E-02
62	0.19367756E-02	-0.49376301E-02
63	0.11998978E-02	-0.49376376E-02
64	0.43995352E-03	-0.49069822E-02
65	0.28331669E-02	-0.40713102E-02
66	0.19823625E-02	-0.40957443E-02
67	0.11543960E-02	-0.40954612E-02
68	0.30302233E-03	-0.40707588E-02
69	0.29401106E-02	-0.31552652E-02
70	0.20174270E-02	-0.31730086E-02
71	0.11184728E-02	-0.31714228E-02
72	0.19669812E-03	-0.31522501E-02
73	0.30202048E-02	-0.21780636E-02
74	0.20402528E-02	-0.21899596E-02
75	0.10906793E-02	-0.21851845E-02
76	0.12392816E-03	-0.21725015E-02
77	0.30796938E-02	-0.11562195E-02
78	0.20458561E-02	-0.11569082E-02
79	0.10730776E-02	-0.11622997E-02
80	0.88306639E-04	-0.11613697E-02
81	0.31224550E-02	0.0
82	0.20550436E-02	-0.10459138E-03
83	0.10709199E-02	-0.14896948E-03
84	0.80656479E-04	-0.15713293E-03

TABLE 2. STRESSES AT ELEMENT CENTROIDS

ELEMENT	X	Y	SIGMA(X)	SIGMA(Y)	TAU(X,Y)	SIGMA(1)	SIGMA(2)	ANGLE
1	2.50	2.50	4.6421E 03	-2.0794E 04	-8.6887E 03	7.3268E 03	-2.3478E 04	-1.7170E 01
2	2.50	7.50	-1.8690E 03	-1.1060E 04	-8.7997E 03	3.4629E 03	-1.6392E 04	-3.1212E 01
3	2.50	12.50	-2.7854E 03	-2.3686E 03	-3.4407E 03	8.6998E 02	-6.0240E 03	-4.6734E 01
4	7.50	2.50	1.4220E 04	1.6562E 03	-3.3298E 03	1.5048E 04	8.2828E 02	-1.3963E 01
5	7.50	7.50	-2.8317E 03	-9.8922E 02	-1.0811E 04	8.9401E 03	-1.2761E 04	-4.7435E 01
6	7.50	12.50	-1.1399E 04	-5.8405E 02	-6.6742E 03	2.5984E 03	-1.4581E 04	-6.4507E 01
7	12.50	2.50	2.1807E 04	3.5860E 02	-4.6648E 03	2.2778E 04	-6.1202E 02	-1.1754E 01
8	12.50	7.50	-7.0828E 02	1.1261E 03	-9.7743E 03	1.0026E 04	-9.6083E 03	-4.7681E 01
9	12.50	12.50	-2.1110E 04	3.1997E 02	-6.2682E 03	2.0187E 03	-2.2809E 04	-7.4837E 01
10	17.50	2.50	3.0004E 04	1.9698E 02	-5.2850E 03	3.0913E 04	-7.1235E 02	-9.7627E 00
11	17.50	7.50	-6.9744E 01	3.7268E 02	-9.6998E 03	9.8538E 03	-9.5509E 03	-4.5653E 01
12	17.50	12.50	-2.9943E 04	1.7882E 02	-5.6149E 03	1.1914E 03	-3.0956E 04	-7.9777E 01
13	22.50	2.50	3.8420E 04	3.2031E 01	-5.4214E 03	3.9171E 04	-7.1892E 02	-7.8863E 00
14	22.50	7.50	3.7657E 01	6.8371E 01	-9.6497E 03	9.7027E 03	-9.5967E 03	-4.5046E 01
15	22.50	12.50	-3.8466E 04	2.0309E 01	-5.4286E 03	7.7136E 02	-3.9217E 04	-8.2123E 01
16	27.50	2.50	4.6858E 04	2.6668E 01	-5.4301E 03	4.7480E 04	-5.9470E 02	-6.5280E 00
17	27.50	7.50	1.8033E 01	5.5059E 01	-9.6056E 03	9.6422E 03	-9.5691E 03	-4.5055E 01
18	27.50	12.50	-4.6885E 04	1.4723E 01	-5.3717E 03	6.2209E 02	-4.7493E 04	-8.3549E 01
19	32.50	2.50	5.5306E 04	1.2140E 02	-5.5196E 03	5.5853E 04	-4.2525E 02	-5.6561E 00
20	32.50	7.50	-9.9190E 01	2.2283E 02	-9.5634E 03	9.6266E 03	-9.5029E 03	-4.5482E 01
21	32.50	12.50	-5.5215E 04	1.1536E 02	-5.2313E 03	6.0561E 02	-5.5706E 04	-8.4646E 01
22	37.50	2.50	6.3865E 04	4.5250E 01	-5.8169E 03	6.4390E 04	-4.8060E 02	-5.1656E 00
23	37.50	7.50	-5.1285E 02	4.4662E 02	-9.5259E 03	9.5049E 03	-9.5711E 03	-4.6442E 01
24	37.50	12.50	-6.3362E 04	3.5961E 01	-4.8797E 03	4.0934E 02	-6.3735E 04	-8.5625E 01
25	42.50	2.50	7.2697E 04	-6.6916E 02	-5.4354E 03	7.3097E 04	-1.0696E 03	-4.2141E 00
26	42.50	7.50	-1.4602E 03	-2.2770E 03	-9.9938E 03	8.1335E 03	-1.1871E 04	-4.3830E 01
27	42.50	12.50	-7.1246E 04	5.4472E 02	-4.7065E 03	8.5195E 02	-7.1553E 04	-8.6265E 01
28	47.50	2.50	7.9261E 04	-1.8171E 03	-2.5200E 03	7.9339E 04	-1.8953E 03	-1.7786E 00
29	47.50	7.50	2.0466E 03	-8.4847E 03	-6.1259E 03	4.8590E 03	-1.1297E 04	-2.4659E 01
30	47.50	12.50	-8.1317E 04	-1.8478E 04	-1.1402E 04	-1.6473E 04	-2.3322E 04	-8.0027E 01
31	52.50	2.50	7.9261E 04	-1.8183E 03	2.5140E 03	7.9339E 04	-1.8962E 03	1.7743E 00
32	52.50	7.50	2.0459E 03	-8.4835E 03	6.1210E 03	4.8549E 03	-1.1292E 04	2.4651E 01
33	52.50	12.50	-8.1316E 04	-1.8475E 04	1.1401E 04	-1.6471E 04	-8.3321E 04	8.0028E 01
34	57.50	2.50	7.2697E 04	-6.7012E 02	5.4306E 03	7.3097E 04	-1.0699E 03	4.2105E 00
35	57.50	7.50	-1.4602E 03	-2.2765E 03	9.9898E 03	8.1298E 03	-1.1866E 04	4.3830E 01
36	57.50	12.50	-7.1245E 04	5.4697E 02	4.7043E 03	8.5391E 02	-7.1552E 04	8.6267E 01
37	62.50	2.50	6.3868E 04	4.7230E 01	5.8111E 03	6.4393E 04	-4.7757E 02	5.1604E 00
38	62.50	7.50	-5.1123E 02	4.4466E 02	9.5206E 03	9.4994E 03	-9.5659E 03	4.6437E 01
39	62.50	12.50	-6.3363E 04	3.3133E 01	4.8770E 03	4.0613E 02	-6.3736E 04	8.5627E 01
40	67.50	2.50	5.5310E 04	1.2101E 02	5.5141E 03	5.5855E 04	-4.2453E 02	5.6502E 00
41	67.50	7.50	-9.7896E 01	2.2224E 02	9.5579E 03	9.6214E 03	-9.4971E 03	4.5480E 01
42	67.50	12.50	-5.5218E 04	1.1273E 02	5.2302E 03	6.0277E 02	-5.5708E 04	8.4648E 01
43	72.50	2.50	4.6863E 04	2.4008E 01	5.4257E 03	4.7483E 04	-5.9629E 02	6.5221E 00
44	72.50	7.50	2.1257E 01	5.8000E 01	9.6018E 03	9.6415E 03	-9.5622E 03	4.5055E 01
45	72.50	12.50	-4.6888E 04	1.2582E 01	5.3698E 03	6.1952E 02	-4.7495E 04	8.3551E 01
46	77.50	2.50	3.8426E 04	3.1883E 01	5.4191E 03	3.9176E 04	-7.1834E 02	7.8820E 00
47	77.50	7.50	3.9095E 01	7.0279E 01	9.6482E 03	9.7029E 03	-9.5935E 03	4.5046E 01

ELFMENT	X	Y	SIGMA(X)	SIGMA(Y)	TAU(X,Y)	SIGMA(1)	SIGMA(2)	ANGLE
48	77.50	12.50	-3.8470E 04	1.8703E 01	5.4291E 03	7.6987E 02	-3.9221E 04	8.2123E 01
49	82.50	2.50	3.0009E 04	1.9712E 02	5.2842E 03	3.0918E 04	-7.1181E 02	9.7598E 00
50	82.50	7.50	-6.7959E 01	3.7371E 02	9.6996E 03	9.8550E 03	-9.5493E 03	4.5652E 01
51	82.50	12.50	-2.9945E 04	1.8019E 02	5.6164E 03	1.1932E 03	-3.0958E 04	7.9776E 01
52	87.50	2.50	2.1814E 04	3.5832E 02	4.6630E 03	2.2784E 04	-6.1126E 02	1.1746E 01
53	87.50	7.50	-7.0600E 02	1.1271E 03	9.7729E 03	1.0026E 04	-9.6053E 03	4.7679E 01
54	87.50	12.50	-2.1111E 04	3.2170E 02	6.2690E 03	2.0206E 03	-2.2810E 04	7.4837E 01
55	92.50	2.50	1.4230E 04	1.6576E 03	3.3270E 03	1.5056E 04	8.3151E 02	1.3945E 01
56	92.50	7.50	-2.8296E 03	-9.8742E 02	1.0811E 04	8.9416E 03	-1.2759E 04	4.7435E 01
57	92.50	12.50	-1.1402E 04	-5.8413E 02	6.6762E 03	2.5993E 03	-1.4585E 04	6.4507E 01
58	97.50	2.50	4.6559E 03	-2.0793E 04	8.6831E 03	7.3362E 03	-2.3473E 04	1.7155E 01
59	97.50	7.50	-1.8695E 03	-1.1063E 04	8.8006E 03	3.4626E 03	-1.6395E 04	3.1211E 01
60	97.50	12.50	-2.7866E 03	-2.3699E 03	3.4421E 03	8.7017E 02	-6.0267E 03	4.6732E 01

UNIVERSITY OF IOWA...START JOB 577.....2.23.37 PM A APR 75.....HASPLIST...80-80 LISTING.....UNIVERSITY OF IOWA
 UNIVERSITY OF IOWA...START JOB 577.....2.23.37 PM A APR 75.....HASPLIST...80-80 LISTING.....UNIVERSITY OF IOWA
 UNIVERSITY OF IOWA...START JOB 577.....2.23.37 PM A APR 75.....HASPLIST...80-80 LISTING.....UNIVERSITY OF IOWA

//HASPLIST JOB (-----,14,....66)*80-80 LISTING* JCR 577

CARDS REQUIRED FOR STRUDL II 'SIMPLE BEAM PROBLEM' SOLUTION

-----JOB ACCOUNTING CARD -----

//STRUDLII JOB (-----), 'RICH RYNER', MSGLEVEL=(1,1)

----- JCL (JOB CONTROL LANGUAGE) CARDS -----

```
// EXEC PGM=QQICEX2,REGION=350K,PARM=2000
//STEPLIB DD DSN=ICES.$$$$$SYS,DISP=SHR
// DD DSN=ICES,STRUCL.M002,UNIT=2314,VOL=SER=ICES01,DISP=OLD
// DD DSN=ICES,STRUCL.M001,UNIT=2314,VOL=SER=ICES01,DISP=OLD
// DD DSN=ICES,BUGS.STRUCL2,UNIT=2314,VOL=SER=ICES01,DISP=OLD
// DD DSN=ICES,STFRAME,DISP=OLD,VOL=SER=ICES01,UNIT=2314
// DD DSN=ICES,STFINELM,DISP=OLD,VOL=SER=ICES01,UNIT=2314
// DD DSN=ICES,STDYNONL,DISP=OLD,VOL=SER=ICES01,UNIT=2314
// DD DSN=ICES,STOPTSEL,DISP=OLD,VOL=SER=ICES01,UNIT=2314
// DD DSN=ICES,STCONCRT,DISP=OLD,VOL=SER=ICES01,UNIT=2314
//FT06F001 DD SYSOUT=A
//FT06F002 DD SYSOUT=A
//FT07F001 DD SYSOUT=B
//DD1 DD UNIT=SYSDA,SPACE=(TRK,(10,10)),DCB=DSORG=DA
//DD2 DD DSN=ICES,STRUCL.DATA,DISP=OLD,VOL=SER=ICES01,UNIT=2314
//DD3 DD DSN=ICES,SYSDATA,DISP=OLD,VOL=SER=ICES01,UNIT=2314
//DD4 DD UNIT=SYSDA,SPACE=(TRK,(10,10)),DCB=DSORG=DA
//FT05F001 DD *
```

----- PROBLEM DATA CARDS -----

```
STRUDL *JOB 1* *SIMPLE BEAM PROBLEM METRIC UNITS RICH RYNER*
TYPE PLANE STRESS
UNITS CENTIMETERS KILOGRAMS
JOINT COORDINATES
1 0. 0. S
2 0. 5.
3 0. 10.
4 0. 15.
5 5. 0.
6 5. 5.
7 5. 10.
8 5. 15.
9 10. 0.
10 10. 5.
11 10. 10.
12 10. 15.
13 15. 0.
14 15. 5.
15 15. 10.
16 15. 15.
```

STRUDL II Input:
 "Simple Beam Problem"

17 20. 0.
18 20. 5.
19 20. 10.
20 20. 15.
21 25. 0.
22 25. 5.
23 25. 10.
24 25. 15.
25 30. 0.
26 30. 5.
27 30. 10.
28 30. 15.
29 35. 0.
30 35. 5.
31 35. 10.
32 35. 15.
33 40. 0.
34 40. 5.
35 40. 10.
36 40. 15.
37 45. 0.
38 45. 5.
39 45. 10.
40 45. 15.
41 50. 0.
42 50. 5.
43 50. 10.
44 50. 15.
45 55. 0.
46 55. 5.
47 55. 10.
48 55. 15.
49 60. 0.
50 60. 5.
51 60. 10.
52 60. 15.
53 65. 0.
54 65. 5.
55 65. 10.
56 65. 15.
57 70. 0.
58 70. 5.
59 70. 10.
60 70. 15.
61 75. 0.
62 75. 5.
63 75. 10.
64 75. 15.
65 80. 0.
66 80. 5.
67 80. 10.
68 80. 15.
69 85. 0.
70 85. 5.
71 85. 10.
72 85. 15.
73 90. 0.
74 90. 5.
75 90. 10.
76 90. 15.
77 95. 0.
78 95. 5.
79 95. 10.
80 95. 15.
81 100. 0. S

82 100. 5.
 83 100. 10.
 84 100. 15.

ELEMENT INCIDENCEFS

1	1	5	6	2
2	2	6	7	3
3	3	7	8	4
4	5	9	10	6
5	6	10	11	7
6	7	11	12	8
7	9	13	14	10
8	10	14	15	11
9	11	15	16	12
10	13	17	18	14
11	14	18	19	15
12	15	19	20	16
13	17	21	22	18
14	18	22	23	19
15	19	23	24	20
16	21	25	26	22
17	22	26	27	23
18	23	27	28	24
19	25	29	30	26
20	26	30	31	27
21	27	31	32	28
22	29	33	34	30
23	30	34	35	31
24	31	35	36	32
25	33	37	38	34
26	34	38	39	35
27	35	39	40	36
28	37	41	42	38
29	38	42	43	39
30	39	43	44	40
31	41	45	46	42
32	42	46	47	43
33	43	47	48	44
34	45	49	50	46
35	46	50	51	47
36	47	51	52	48
37	49	53	54	50
38	50	54	55	51
39	51	55	56	52
40	53	57	58	54
41	54	58	59	55
42	55	59	60	56
43	57	61	62	58
44	58	62	63	59
45	59	63	64	60
46	61	65	66	62
47	62	66	67	63
48	63	67	68	64
49	65	69	70	66
50	66	70	71	67
51	67	71	72	68
52	69	73	74	70
53	70	74	75	71
54	71	75	76	72
55	73	77	78	74
56	74	78	79	75
57	75	79	80	76
58	77	81	82	78
59	78	82	83	79
60	79	83	84	80

ELEMENT PROPERTIES

1 TO 60 TYPE 'PSP' THICKNESS 5.

CONSTANTS

POISSON 0.303 ALL

E 2109209.1 ALL

G 801499.45 ALL

DENSITY .007A ALL

JOINT RELEASE

81 FORCE X

LOADING 1 'VERTICAL CENTER OF BEAM'

JOINT LOADS

44 FORCE Y -1000.0

PRINT DATA

DUMP TIME

STIFFNESS ANALYSIS

LIST DISPLACEMENTS STRESSES STRAINS ALL

FINISH

----- JCL END CARD -----

/*

----- END OF STRUDL II INPUT DECK -----

```

UNIVERSITY OF IOWA....START JOB 831.....5.16.26 PM 16 FEB 75.....STRU DL2 ....RICH RYNER.....UNIVERSITY OF IOW
UNIVERSITY OF IOWA....START JOB 831.....5.16.26 PM 16 FEB 75.....STRU DL2 ....RICH RYNER.....UNIVERSITY OF IOW
UNIVERSITY OF IOWA....START JOB 831.....5.16.26 PM 16 FEB 75.....STRU DL2 ....RICH RYNER.....UNIVERSITY OF IOW

```

H A S P S Y S T E M L O G

```

$ 17.45.58 JOB 831 -- STRU DL2 -- BEGINNING EXEC - INIT 8 - CLASS F
# 17.49.33 JOB 831 END EXECUTION.

```

```

//STRU DL2 JOB (-----,9,10),RICH RYNER,MSGLEVEL=(1,1),PRTY=0             JOB 831

```

```

// EXEC PGM=000ICEX2,REGION=350K,PARM=2000
//STEPLIB DD DSN=ICES,DDDSYS,DISP=SHR
// DD DSN=ICES,STRU DL,MOD2,UNIT=2314,VOL=SER=ICES01,DISP=OLD
// DD DSN=ICES,STRU DL,MOD1,UNIT=2314,VOL=SER=ICES01,DISP=OLD
// DD DSN=ICES,BUGS,STRU DL2,UNIT=2314,VOL=SER=ICES01,DISP=OLD
// DD DSN=ICES,STFRAME,DISP=OLD,VOL=SER=ICES01,UNIT=2314
// DD DSN=ICES,STFINELM,DISP=OLD,VOL=SER=ICES01,UNIT=2314
// DD DSN=ICES,STDYNMNL,DISP=OLD,VOL=SER=ICES01,UNIT=2314
// DD DSN=ICES,STOPTSEL,DISP=OLD,VOL=SER=ICES01,UNIT=2314
// DD DSN=ICES,STCONCRT,DISP=OLD,VOL=SER=ICES01,UNIT=2314
//FT06F001 DD SYSOUT=A
//FT06F002 DD SYSOUT=A
//FT07F001 DD SYSOUT=B
//DD1 DD UNIT=SYSDA,SPACE=(TRK,(10,10)),DCB=DSORG=DA
//DD2 DD DSN=ICES,STRU DL,DATA,DISP=OLD,VOL=SER=ICES01,UNIT=2314
//DD3 DD DSN=ICES,SYSO DATA,DISP=OLD,VOL=SER=ICES01,UNIT=2314
//DD4 DD UNIT=SYSDA,SPACE=(TRK,(10,10)),DCB=DSORG=DA
//FT05F001 DD *
//

```

```

IEF236I ALLOC. FOR STRU DL2

```

```

IEF237I 230 ALLOCATED TO STEPLIB
IEF237I 230 ALLOCATED TO
IEF237I 230 ALLOCATED TO

```

```

IEF237I 230 ALLOCATED TO

```

```

IEF237I 230 ALLOCATED TO

```

```

IEF237I 230 ALLOCATED TO

```

```

IEF237I 230 ALLOCATED TO

```

```

IEF237I 230 ALLOCATED TO

```

```

IEF237I 230 ALLOCATED TO

```

```

IEF237I 330 ALLOCATED TO FT06F001

```

```

IEF237I 331 ALLOCATED TO FT06F002

```

```

IEF237I 340 ALLOCATED TO FT07F001

```

```

IEF237I 235 ALLOCATED TO DD1

```

```

IEF237I 230 ALLOCATED TO DD2

```

```

IEF237I 230 ALLOCATED TO DD3

```

```

IEF237I 235 ALLOCATED TO DD4

```

```

IEF237I 300 ALLOCATED TO FT05F001

```

```

IEF142I     STEP WAS EXECUTED - COND CODE 4444

```

```

IEF245I     ICES,DDDSYS                                     KEPT
IEF245I     VOL SER NOS= ICES01.                             KEPT
IEF245I     ICES,STRU DL,MOD2                                 KEPT
IEF245I     VOL SER NOS= ICES01.                             KEPT
IEF245I     ICES,STRU DL,MOD1                                 KEPT
IEF245I     VOL SER NOS= ICES01.                             KEPT
IEF245I     ICES,BUGS,STRU DL2                                KEPT
IEF245I     VOL SER NOS= ICES01.                             KEPT
IEF245I     ICES,STFRAME                                        KEPT

```

IEF2A5I VOL SER NOS= ICES01.
IEF2A5I ICES.STFINELM KEPT
IEF2A5I VOL SER NOS= ICES01.
IEF2A5I ICES.STDYNOML KEPT
IEF2A5I VOL SER NOS= ICES01.
IEF2A5I ICES.STOPTSEL KEPT
IEF2A5I VOL SER NOS= ICES01.
IEF2A5I ICES.STCONCRT KEPT
IEF2A5I VOL SER NOS= ICFS01.
IEF2A5I SYS75046.T101206.RV001.STRUOL2.R0000004 DELETED
IEF2A5I VOL SER NOS= IOWA08.
IEF2A5I ICES.STRUOL.DATA KEPT
IEF2A5I VOL SER NOS= ICES01.
IEF2A5I ICES.SYSDATA KEPT
IEF2A5I VOL SER NOS= ICFS01.
IEF2A5I SYS75046.T101206.RV001.STRUOL2.R0000005 DELETED
IEF2A5I VOL SER NOS= IOWA08.

ACCTMG -- 38.94 SEC. CPU, 2 MIN. 33.20 SEC. WAIT, DAC= 2759, HWM= 350K

*
INTEGRATED CIVIL ENGINEERING SYSTEMS *
*
- ICES - *
*
FEB 15, 1975 TIME=17.46.06 *

STRU DL 'JOB 1' *SIMPLE BEAM PROBLEM METRIC UNITS

RICH RYNER*

```
*****  
*  
*          ICES STRU DL-II          *  
*    THE STRUCTURAL DESIGN LANGUAGE  *  
*  
*    CIVIL ENGINEERING SYSTEMS LAHORATORY *  
*    MASSACHUSETTS INSTITUTE OF TECHNOLOGY *  
*    CAMBRIDGE, MASSACHUSETTS          *  
*  
*    V2 M2      JUNF, 1972          *  
*    17:46:27   2/15/75            *  
*  
*****
```

TYPE PLANE STRESS

UNITS CENTIMETERS KILOGRAMS

JOINT COORDINATES

1	0.	0.	S
2	0.	5.	
3	0.	10.	
4	0.	15.	
5	5.	0.	
6	5.	5.	
7	5.	10.	
8	5.	15.	
9	10.	0.	
10	10.	5.	
11	10.	10.	
12	10.	15.	
13	15.	0.	
14	15.	5.	
15	15.	10.	
16	15.	15.	
17	20.	0.	
18	20.	5.	
19	20.	10.	

20 20. 15.
21 25. 0.
22 25. 5.
23 25. 10.
24 25. 15.
25 30. 0.
26 30. 5.
27 30. 10.
28 30. 15.
29 35. 0.
30 35. 5.
31 35. 10.
32 35. 15.
33 40. 0.
34 40. 5.
35 40. 10.
36 40. 15.
37 45. 0.
38 45. 5.
39 45. 10.
40 45. 15.
41 50. 0.
42 50. 5.
43 50. 10.
44 50. 15.
45 55. 0.
46 55. 5.
47 55. 10.
48 55. 15.
49 60. 0.
50 60. 5.

51 60. 10.
52 60. 15.
53 65. 0.
54 65. 5.
55 65. 10.
56 65. 15.
57 70. 0.
58 70. 5.
59 70. 10.
60 70. 15.
61 75. 0.
62 75. 5.
63 75. 10.
64 75. 15.
65 80. 0.
66 80. 5.
67 80. 10.
68 80. 15.
69 85. 0.
70 85. 5.
71 85. 10.
72 85. 15.
73 90. 0.
74 90. 5.
75 90. 10.
76 90. 15.
77 95. 0.
78 95. 5.
79 95. 10.
80 95. 15.
81 100. 0. S

82 100. 5.

83 100. 10.

84 100. 15.

ELEMENT INCIDENCES

1 1 5 6 2

2 2 6 7 3

3 3 7 8 4

4 5 9 10 6

5 6 10 11 7

6 7 11 12 8

7 9 13 14 10

8 10 14 15 11

9 11 15 16 12

10 13 17 18 14

11 14 18 19 15

12 15 19 20 16

13 17 21 22 18

14 18 22 23 19

15 19 23 24 20

16 21 25 26 22

17 22 26 27 23

18 23 27 28 24

19 25 29 30 26

20 26 30 31 27

21 27 31 32 28

22 29 33 34 30

23 30 34 35 31

24 31 35 36 32

25 33 37 38 34

26 34 38 39 35

27 35 39 40 36

28 37 41 42 38
29 38 42 43 39
30 39 43 44 40
31 41 45 46 42
32 42 46 47 43
33 43 47 48 44
34 45 49 50 46
35 46 50 51 47
36 47 51 52 48
37 49 53 54 50
38 50 54 55 51
39 51 55 56 52
40 53 57 58 54
41 54 58 59 55
42 55 59 60 56
43 57 61 62 58
44 58 62 63 59
45 59 63 64 60
46 61 65 66 62
47 62 66 67 63
48 63 67 68 64
49 65 69 70 66
50 66 70 71 67
51 67 71 72 68
52 69 73 74 70
53 70 74 75 71
54 71 75 76 72
55 73 77 78 74
56 74 78 79 75
57 75 79 80 76
58 77 81 82 78

59 78 R2 R3 79

60 79 R3 R4 80

ELEMENT PROPERTIES

1 TO 60 TYPE 'PSR' THICKNESS 5.

CONSTANTS

POISSON 0.303 ALL

E 2109209.1 ALL

G 801499.45 ALL

DENSITY .0078 ALL

JOINT RELEASE

81 FORCE X

LOADING 1 'VERTICAL CENTER OF BEAM'

JOINT LOADS

44 FORCE Y -1000.0

PRINT DATA

 * PROBLEM DATA FROM INTERNAL STORAGE *

JOB ID - JOB 1 JOB TITLE - SIMPLE BEAM PROBLEM METRIC UNITS RICH RYNER

ACTIVE UNITS - LENGTH WEIGHT ANGLE TEMPERATURE TIME
 CM KG RAD DEGF SEC

***** STRUCTURAL DATA *****

ACTIVE STRUCTURE TYPE - PLANE STRESS

ACTIVE COORDINATE AXES X Y

JOINT COORDINATES-----/				STATUS---/	
JOINT	X	Y	Z	CONDITION	
1	0.0	0.0	0.0	SUPPORT	ACTIVE GLOBAL
2	0.0	5.000	0.0		ACTIVE GLOBAL
3	0.0	10.000	0.0		ACTIVE GLOBAL
4	0.0	15.000	0.0		ACTIVE GLOBAL
5	5.000	0.0	0.0		ACTIVE GLOBAL
6	5.000	5.000	0.0		ACTIVE GLOBAL
7	5.000	10.000	0.0		ACTIVE GLOBAL
8	5.000	15.000	0.0		ACTIVE GLOBAL
9	10.000	0.0	0.0		ACTIVE GLOBAL
10	10.000	5.000	0.0		ACTIVE GLOBAL
11	10.000	10.000	0.0		ACTIVE GLOBAL
12	10.000	15.000	0.0		ACTIVE GLOBAL
13	15.000	0.0	0.0		ACTIVE GLOBAL
14	15.000	5.000	0.0		ACTIVE GLOBAL
15	15.000	10.000	0.0		ACTIVE GLOBAL
16	15.000	15.000	0.0		ACTIVE GLOBAL
17	20.000	0.0	0.0		ACTIVE GLOBAL
18	20.000	5.000	0.0		ACTIVE GLOBAL
19	20.000	10.000	0.0		ACTIVE GLOBAL
20	20.000	15.000	0.0		ACTIVE GLOBAL
21	25.000	0.0	0.0		ACTIVE GLOBAL
22	25.000	5.000	0.0		ACTIVE GLOBAL
23	25.000	10.000	0.0		ACTIVE GLOBAL
24	25.000	15.000	0.0		ACTIVE GLOBAL
25	30.000	0.0	0.0		ACTIVE GLOBAL
26	30.000	5.000	0.0		ACTIVE GLOBAL
27	30.000	10.000	0.0		ACTIVE GLOBAL
28	30.000	15.000	0.0		ACTIVE GLOBAL
29	35.000	0.0	0.0		ACTIVE GLOBAL
30	35.000	5.000	0.0		ACTIVE GLOBAL
31	35.000	10.000	0.0		ACTIVE GLOBAL
32	35.000	15.000	0.0		ACTIVE GLOBAL
33	40.000	0.0	0.0		ACTIVE GLOBAL
34	40.000	5.000	0.0		ACTIVE GLOBAL
35	40.000	10.000	0.0		ACTIVE GLOBAL
36	40.000	15.000	0.0		ACTIVE GLOBAL
37	45.000	0.0	0.0		ACTIVE GLOBAL
38	45.000	5.000	0.0		ACTIVE GLOBAL

39	45.000	10.000	0.0	ACTIVE	GLORAL
40	45.000	15.000	0.0	ACTIVE	GLORAL
41	50.000	0.0	0.0	ACTIVE	GLORAL
42	50.000	5.000	0.0	ACTIVE	GLORAL
43	50.000	10.000	0.0	ACTIVE	GLORAL
44	50.000	15.000	0.0	ACTIVE	GLORAL
45	55.000	0.0	0.0	ACTIVE	GLORAL
46	55.000	5.000	0.0	ACTIVE	GLORAL
47	55.000	10.000	0.0	ACTIVE	GLORAL
48	55.000	15.000	0.0	ACTIVE	GLORAL
49	60.000	0.0	0.0	ACTIVE	GLOBAL
50	60.000	5.000	0.0	ACTIVE	GLOBAL
51	60.000	10.000	0.0	ACTIVE	GLOBAL
52	60.000	15.000	0.0	ACTIVE	GLOBAL
53	65.000	0.0	0.0	ACTIVE	GLOBAL
54	65.000	5.000	0.0	ACTIVE	GLOBAL
55	65.000	10.000	0.0	ACTIVE	GLOBAL
56	65.000	15.000	0.0	ACTIVE	GLOBAL
57	70.000	0.0	0.0	ACTIVE	GLOBAL
58	70.000	5.000	0.0	ACTIVE	GLOBAL
59	70.000	10.000	0.0	ACTIVE	GLOBAL
60	70.000	15.000	0.0	ACTIVE	GLOBAL
61	75.000	0.0	0.0	ACTIVE	GLOBAL
62	75.000	5.000	0.0	ACTIVE	GLOBAL
63	75.000	10.000	0.0	ACTIVE	GLOBAL
64	75.000	15.000	0.0	ACTIVE	GLOBAL
65	80.000	0.0	0.0	ACTIVE	GLOBAL
66	80.000	5.000	0.0	ACTIVE	GLOBAL
67	80.000	10.000	0.0	ACTIVE	GLOBAL
68	80.000	15.000	0.0	ACTIVE	GLOBAL
69	85.000	0.0	0.0	ACTIVE	GLOBAL
70	85.000	5.000	0.0	ACTIVE	GLOBAL
71	85.000	10.000	0.0	ACTIVE	GLOBAL
72	85.000	15.000	0.0	ACTIVE	GLOBAL
73	90.000	0.0	0.0	ACTIVE	GLOBAL
74	90.000	5.000	0.0	ACTIVE	GLOBAL
75	90.000	10.000	0.0	ACTIVE	GLOBAL
76	90.000	15.000	0.0	ACTIVE	GLOBAL
77	95.000	0.0	0.0	ACTIVE	GLOBAL
78	95.000	5.000	0.0	ACTIVE	GLOBAL
79	95.000	10.000	0.0	ACTIVE	GLOBAL
80	95.000	15.000	0.0	ACTIVE	GLOBAL
81	100.000	0.0	0.0	SUPPORT	GLOBAL
82	100.000	5.000	0.0	ACTIVE	GLOBAL
83	100.000	10.000	0.0	ACTIVE	GLOBAL
84	100.000	15.000	0.0	ACTIVE	GLOBAL

JOINT RELEASES			/ELASTIC SUPPORT RELEASES/								
JOINT	FORCE	MOMENT	THETA 1	THETA 2	THETA 3	KFX	KEY	KFZ	KMX	KMY	KMZ
A1	X		0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

ELEMENT INCIDENCES						
ELEMENT	NODES					
1	1	5	6	2		ACTIVE
2	2	6	7	3		ACTIVE

3	3	7	8	4	ACTIVE
4	5	9	10	6	ACTIVE
5	6	10	11	7	ACTIVE
6	7	11	12	8	ACTIVE
7	9	13	14	10	ACTIVE
8	10	14	15	11	ACTIVE
9	11	15	16	12	ACTIVE
10	13	17	18	14	ACTIVE
11	14	18	19	15	ACTIVE
12	15	19	20	16	ACTIVE
13	17	21	22	18	ACTIVE
14	18	22	23	19	ACTIVE
15	19	23	24	20	ACTIVE
16	21	25	26	22	ACTIVE
17	22	26	27	23	ACTIVE
18	23	27	28	24	ACTIVE
19	25	29	30	26	ACTIVE
20	26	30	31	27	ACTIVE
21	27	31	32	28	ACTIVE
22	29	33	34	30	ACTIVE
23	30	34	35	31	ACTIVE
24	31	35	36	32	ACTIVE
25	33	37	38	34	ACTIVE
26	34	38	39	35	ACTIVE
27	35	39	40	36	ACTIVE
28	37	41	42	38	ACTIVE
29	38	42	43	39	ACTIVE
30	39	43	44	40	ACTIVE
31	41	45	46	42	ACTIVE
32	42	46	47	43	ACTIVE
33	43	47	48	44	ACTIVE
34	45	49	50	46	ACTIVE
35	46	50	51	47	ACTIVE
36	47	51	52	48	ACTIVE
37	49	53	54	50	ACTIVE
38	50	54	55	51	ACTIVE
39	51	55	56	52	ACTIVE
40	53	57	58	54	ACTIVE
41	54	58	59	55	ACTIVE
42	55	59	60	56	ACTIVE
43	57	61	62	58	ACTIVE
44	58	62	63	59	ACTIVE
45	59	63	64	60	ACTIVE
46	61	65	66	62	ACTIVE
47	62	66	67	63	ACTIVE
48	63	67	68	64	ACTIVE
49	65	69	70	66	ACTIVE
50	66	70	71	67	ACTIVE
51	67	71	72	68	ACTIVE
52	69	73	74	70	ACTIVE
53	70	74	75	71	ACTIVE
54	71	75	76	72	ACTIVE
55	73	77	78	74	ACTIVE
56	74	78	79	75	ACTIVE
57	75	79	80	76	ACTIVE
58	77	81	82	78	ACTIVE
59	78	82	83	79	ACTIVE
60	79	83	84	80	ACTIVE

ELEMENT PROPERTIES-----CURVATURES-----THERMAL EXPANSION COEFFICIENTS-----
 ELEMENT TYPE THICKNESS /-----K1-----K2-----K12-----CAX-----CAY-----CAZ-----CSXY-----CSXZ-----CSYZ

ELEMENT	TYPE	THICKNESS	K1	K2	K12	CAX	CAY	CAZ	CSXY	CSXZ	CSYZ
1	PSR	5.000									
2	PSR	5.000									
3	PSR	5.000									
4	FSR	5.000									
5	PSR	5.000									
6	PSR	5.000									
7	PSR	5.000									
8	PSK	5.000									
9	PSR	5.000									
10	PSP	5.000									
11	FSR	5.000									
12	PSR	5.000									
13	PSK	5.000									
14	PSR	5.000									
15	PSR	5.000									
16	PSP	5.000									
17	PSR	5.000									
18	PSR	5.000									
19	PSR	5.000									
20	FSR	5.000									
21	FSR	5.000									
22	PSR	5.000									
23	PSK	5.000									
24	PSK	5.000									
25	PSR	5.000									
26	PSR	5.000									
27	FSR	5.000									
28	PSP	5.000									
29	FSR	5.000									
30	PSK	5.000									
31	PSK	5.000									
32	PSR	5.000									
33	PSR	5.000									
34	PSP	5.000									
35	PSK	5.000									
36	PSR	5.000									
37	PSR	5.000									
38	PSP	5.000									
39	PSK	5.000									
40	PSR	5.000									
41	PSR	5.000									
42	PSR	5.000									
43	FSR	5.000									
44	PSP	5.000									
45	PSK	5.000									
46	PSR	5.000									
47	PSK	5.000									
48	PSK	5.000									
49	PSR	5.000									
50	PSR	5.000									
51	PSR	5.000									
52	PSK	5.000									
53	PSR	5.000									
54	PSK	5.000									

55	PSR	5.000
56	PSR	5.000
57	PSR	5.000
58	PSR	5.000
59	PSR	5.000
60	PSR	5.000

MEMBER CONSTANTS-----/

CONSTANT	STANDARD	VALUE	DOMAIN	VALUE	MEMBER LIST
----------	----------	-------	--------	-------	-------------

E		0.210921E-07	ALL		
G		0.801499E-06	ALL		
DENSITY		0.779999E-02	ALL		
CTE		0.100000E-01	ALL		
BETA		0.0	ALL		
POISSON		0.303000E-00	ALL		

***** DESIGN DATA *****

USFR DATA SET

PARAMETER DICTIONARY-----/

NAME	TREATMENT	STANDARD	L	W	A	TEMP	TIME
------	-----------	----------	---	---	---	------	------

STRUOL DATA SET

PARAMETER DICTIONARY-----/

NAME	TREATMENT	STANDARD	L	W	A	TEMP	TIME
------	-----------	----------	---	---	---	------	------

FYID	STANDARD	2531.04	-2	1			
PF	STANDARD	1.00					
EBLTUP	STANDARD	1.00					
CODE	REQUIRED						
KY	STANDARD	1.00					
KZ	STANDARD	1.00					
CB	COMPUTE	QOSTCBIC					
LY	COMPUTE	QOSTULEN					
LZ	COMPUTE	QOSTULEN					
CMY	COMPUTE	QOSTCMYC					
CMZ	COMPUTE	QOSTCMZC					
UNICF	COMPUTE	QOSTULEN					
VALUES	STANDARD	1.00					
TRACE	STANDARD	1.00					
PRINTA	STANDARD	1.00					
TRUHAM	STANDARD	STEELWF					

MATERIALS	STANDARD	25.00
SECONDARY	STANDARD	1.00
SDSWAYY	STANDARD	2.00
SDSWAYZ	STANDARD	2.00

USER DATA SET

CONSTRAINT DICTIONARY-----/
 NAME RETRIEVAL

STRUCTURAL DATA SET

CONSTRAINT DICTIONARY-----/
 NAME RETRIEVAL

AX	TABULAR
AY	TABULAR
AZ	TABULAR
IX	TABULAR
IY	TABULAR
IZ	TABULAR
SY	TABULAR
SZ	TABULAR
YD	TABULAR
ZD	TABULAR
FLTK	TABULAR
WBTK	TABULAR
YD/AFL	TABULAR
RY	TABULAR
RZ	TABULAR
COMP	TABULAR
YC	TABULAR
ZC	TABULAR
WEIGHT	TABULAR

***** LOADING DATA *****

LOADING - 1 VERTICAL CENTER OF BEAM STATUS - ACTIVE

MEMBER AND ELEMENT LOADS-----/
 MEMBER/ELEMENT-----

JOINT LOADS-----/

JOINT	STEP	FORCE X	Y	Z	MOMENT X	Y	Z
44		0.0	-1000.000	0.0	0.0	0.0	0.0

JOINT DISPLACEMENTS-----/

JOINT	STEP	DISP. X	Y	Z	ROT. X	Y	Z

JOINT FORCE ASSUMPTIONS-----/

JOINT	THETA	1	2	3	FORCE X	Y	Z	MOMENT X	Y	Z
NO ASSUMPTIONS GIVEN FOR THIS LOADING										

MEMBER FORCE ASSUMPTIONS -----/ -----/

MEMBER COMPONENT DISTANCE VALUE COMPONENT DISTANCE VALUE
NO ASSUMPTIONS GIVEN FOR THIS LOADING

* END OF DATA FROM INTERNAL STORAGE *

DUMP TIME

STIFFNESS ANALYSIS

TIME FOR CONSISTENCY CHECKS FOR 60 MEMBERS	1.18 SECONDS
TIME TO GENERATE 60 ELEMENT STIF. MATRICES	2.65 SECONDS
TIME TO ASSEMBLE THE STIFFNESS MATRIX	7.45 SECONDS
TIME TO PROCESS 84 JOINTS	0.40 SECONDS
TIME TO SOLVE WITH 14 PARTITIONS	5.88 SECONDS
TIME TO PROCESS 84 JOINT DISPLACEMENTS	0.62 SECONDS
TIME TO PROCESS 60 ELEMENT STRESSES	1.92 SECONDS

LIST DISPLACEMENTS STRESSES STRAINS ALL

 RESULTS OF LATEST ANALYSES

PROBLEM - JOB 1 TITLE - SIMPLE BEAM PROBLEM METRIC UNITS RICH RYNER

ACTIVE UNITS CM KG RAD DEGF SEC

ACTIVE STRUCTURE TYPE PLANE STRESS

ACTIVE COORDINATE AXES X Y

LOADING - 1 VERTICAL CENTER OF BEAM

ELEMENT STRESSES

ELEMENT

1	SXX	-0.468722E 01	SYY	-0.203572E 02	SXY	-0.837505E 01
2	SXX	-0.202066E 01	SYY	-0.107996E 02	SXY	-0.843107E 01
3	SXX	-0.266938E 01	SYY	-0.231902E 01	SXY	-0.319168E 01
4	SXX	0.140467E 02	SYY	0.220087E 01	SXY	-0.298308E 01
5	SXX	-0.296774E 01	SYY	-0.794573E 00	SXY	-0.105572E 02
6	SXX	-0.110801E 02	SYY	-0.561785E 00	SXY	-0.645800E 01
7	SXX	0.214090E 02	SYY	0.278031E 00	SXY	-0.451428E 01
8	SXX	-0.648316E 00	SYY	0.124265E 01	SXY	-0.944173E 01
9	SXX	-0.207608E 02	SYY	0.375266E 00	SXY	-0.604309E 01
10	SXX	0.295297E 02	SYY	0.189852E 00	SXY	-0.513516E 01
11	SXX	-0.258943E 01	SYY	0.327446E 00	SXY	-0.947966E 01
12	SXX	-0.295031E 02	SYY	0.185173E 00	SXY	-0.538514E 01
13	SXX	0.379245E 02	SYY	0.121221E 01	SXY	-0.526940E 01
14	SXX	0.550827E 01	SYY	0.397663E 01	SXY	-0.947871E 01
15	SXX	-0.379775E 02	SYY	0.117504E 01	SXY	-0.525321E 01
16	SXX	0.463711E 02	SYY	0.127030E 01	SXY	-0.528154E 01
17	SXX	-0.328310E 01	SYY	0.412567E 01	SXY	-0.947950E 01
18	SXX	-0.464011E 02	SYY	0.121432E 01	SXY	-0.524191E 01
19	SXX	0.548641E 02	SYY	0.138854E 00	SXY	-0.539050E 01
20	SXX	-0.801695E 01	SYY	0.221190E 00	SXY	-0.948129E 01
21	SXX	-0.547797E 02	SYY	0.141156E 00	SXY	-0.513269E 01
22	SXX	0.635187E 02	SYY	0.685230E 01	SXY	-0.576154E 01
23	SXX	-0.518068E 00	SYY	0.596809E 00	SXY	-0.945759E 01
24	SXX	-0.629953E 02	SYY	0.448638E 02	SXY	-0.478712E 01
25	SXX	0.725691E 02	SYY	-0.699042E 00	SXY	-0.537115E 01
26	SXX	-0.163490E 01	SYY	-0.229502E 01	SXY	-0.101565E 02
27	SXX	-0.709279E 02	SYY	0.104352E 01	SXY	-0.447833E 01
28	SXX	0.790893E 02	SYY	-0.184074E 01	SXY	-0.239610E 01
29	SXX	0.221237E 01	SYY	-0.857837E 01	SXY	-0.603645E 01
30	SXX	-0.812955E 02	SYY	-0.188944E 02	SXY	-0.115721E 02
31	SXX	0.790906E 02	SYY	-0.184097E 01	SXY	0.239431E 01
32	SXX	0.221211E 01	SYY	-0.857833E 01	SXY	0.603282E 01
33	SXX	-0.812973E 02	SYY	-0.188941E 02	SXY	0.115702E 02
34	SXX	0.725738E 02	SYY	-0.699128E 00	SXY	0.536874E 01

35	SXX	-0.163505E 01	SYY	-0.229562E 01	SXY	0.101522E 02
36	SXX	-0.709333E 02	SYY	0.104352E 01	SXY	0.447570E 01
37	SXX	0.635267E 02	SYY	0.684596E-01	SXY	0.575953E 01
38	SXX	-0.518265E 00	SYY	0.596999E 00	SXY	0.945385E 01
39	SXX	-0.630039E 02	SYY	0.425193E-02	SXY	0.478485E 01
40	SXX	0.548750E 02	SYY	0.138795E 00	SXY	0.538939E 01
41	SXX	-0.802652E-01	SYY	0.221286E 00	SXY	0.947923E 01
42	SXX	-0.547910E 02	SYY	0.140996E 00	SXY	0.513134E 01
43	SXX	0.463830E 02	SYY	0.128213E-01	SXY	0.528119E 01
44	SXX	0.329684E-01	SYY	0.412983E-01	SXY	0.947889E 01
45	SXX	-0.464128E 02	SYY	0.120862E-01	SXY	0.524151E 01
46	SXX	0.379360E 02	SYY	0.121094E-01	SXY	0.526983E 01
47	SXX	0.550390E-01	SYY	0.396284E-01	SXY	0.947935E 01
48	SXX	-0.379866E 02	SYY	0.119827E-01	SXY	-0.525360E 01
49	SXX	0.295403E 02	SYY	0.189964E 00	SXY	0.513569E 01
50	SXX	-0.252516E-01	SYY	0.327516E 00	SXY	0.948094E 01
51	SXX	-0.295125E 02	SYY	0.185556E 00	SXY	0.538571E 01
52	SXX	0.214122E 02	SYY	0.278211E 00	SXY	0.451505E 01
53	SXX	-0.648016E 00	SYY	0.124324E 01	SXY	0.944371E 01
54	SXX	-0.207666E 02	SYY	0.375399E 00	SXY	0.604444E 01
55	SXX	0.140536E 02	SYY	0.220146E 01	SXY	0.298355E 01
56	SXX	-0.296816E 01	SYY	-0.794322E 00	SXY	0.105602E 02
57	SXX	-0.110845E 02	SYY	-0.561749E 00	SXY	0.645988E 01
58	SXX	0.469214E 01	SYY	-0.203638E 02	SXY	0.837710E 01
59	SXX	-0.202121E 01	SYY	-0.108030E 02	SXY	0.843422E 01
60	SXX	-0.267057E 01	SYY	-0.231937E 01	SXY	0.319271E 01

ELEMENT STRAINS

ELEMENT

1	EXX	0.514698E-05	EYY	-0.103250E-04	EXY	-0.103477E-04
2	EXX	0.593403E-06	EYY	-0.482992E-05	EXY	-0.104169E-04
3	EXX	-0.932447E-06	EYY	-0.716001E-06	EXY	-0.394343E-05
4	EXX	0.634355E-05	EYY	-0.974434E-06	EXY	-0.368571E-05
5	EXX	-0.129290E-05	EYY	0.496175E-07	EXY	-0.130438E-04
6	EXX	-0.517248E-05	EYY	0.132537E-05	EXY	-0.797908E-05
7	EXX	0.101103E-04	EYY	-0.294371E-05	EXY	-0.557755E-05
8	EXX	-0.485888E-06	EYY	0.682288E-06	EXY	-0.116656E-04
9	EXX	-0.989688E-05	EYY	0.316034E-05	EXY	-0.746645E-05
10	EXX	0.139731E-04	EYY	0.415210E-05	EXY	-0.634467E-05
11	EXX	-0.593163E-07	EYY	0.158966E-06	EXY	-0.117125E-04
12	EXX	-0.140144E-04	EYY	0.432609E-05	EXY	-0.665354E-05
13	EXX	0.179787E-04	EYY	-0.544234E-05	EXY	-0.651053E-05
14	EXX	0.204027E-07	EYY	0.109407E-07	EXY	-0.117113E-04
15	EXX	-0.180073E-04	EYY	0.546126E-05	EXY	-0.649053E-05
16	EXX	0.219833E-04	EYY	-0.665546E-05	EXY	-0.652553E-05
17	EXX	0.963882E-08	EYY	0.148439E-07	EXY	-0.117123E-04
18	EXX	-0.220011E-04	EYY	0.667155E-05	EXY	-0.647657E-05
19	EXX	0.259918E-04	EYY	-0.781571E-05	EXY	-0.666015E-05
20	EXX	-0.697846E-07	EYY	0.116386E-06	EXY	-0.117145E-04
21	EXX	-0.259920E-04	EYY	0.793636E-05	EXY	-0.634162E-05
22	EXX	0.301051E-04	EYY	-0.909235E-05	EXY	-0.711859E-05
23	EXX	-0.331357E-06	EYY	0.357378E-06	EXY	-0.116852E-04
24	EXX	-0.298675E-04	EYY	0.905178E-05	EXY	-0.591466E-05
25	EXX	0.345063E-04	EYY	-0.107564E-04	EXY	-0.663625E-05
26	EXX	-0.445318E-06	EYY	-0.853614E-06	EXY	-0.125487E-04
27	EXX	-0.337777E-04	EYY	0.106940E-04	EXY	-0.553314E-05
28	EXX	0.377616E-04	EYY	-0.122344E-04	EXY	-0.296046E-05
29	EXX	0.228125E-05	EYY	-0.438493E-05	EXY	-0.745825E-05

30	EXX	-0.358269E-04	FYY	0.272052E-05	FXY	-0.142978E-04
31	EXX	0.377622E-04	EYY	-0.122346E-04	EXY	0.295826E-05
32	EXX	0.228111E-05	EYY	-0.438487E-05	EXY	0.745477E-05
33	EXX	-0.358297E-04	EYY	0.272008E-05	EXY	0.142954E-04
34	EXX	0.345085E-04	EYY	-0.107571E-04	EXY	0.663327E-05
35	EXX	-0.445420E-06	FYY	-0.853496E-06	FXY	0.125434E-04
36	EXX	-0.337802E-04	EYY	-0.106847E-04	FXY	0.552988E-05
37	EXX	0.301069E-04	EYY	-0.909353E-05	EXY	0.711610E-05
38	EXX	-0.331478E-06	EYY	0.357496E-06	FXY	0.116806E-04
39	EXX	-0.298715E-04	EYY	0.905290E-05	EXY	0.591185E-05
40	EXX	0.259969E-04	FYY	-0.781731E-05	FXY	0.665878E-05
41	EXX	-0.698437E-07	FYY	0.116445E-06	FXY	0.117119E-04
42	EXX	-0.259973E-04	FYY	0.793789E-05	FXY	0.633995E-05
43	EXX	0.219889E-04	FYY	-0.665712E-05	EXY	0.652509E-05
44	EXX	0.969794E-08	FYY	0.148439E-07	EXY	0.117115E-04
45	EXX	-0.220066E-04	EYY	0.667320E-05	EXY	0.647607E-05
46	EXX	0.179842E-04	EYY	-0.544399E-05	EXY	0.651107E-05
47	EXX	0.204018E-07	FYY	0.108816E-07	FXY	0.117121E-04
48	EXX	-0.180126E-04	EYY	0.546298E-05	EXY	0.649101E-05
49	EXX	0.139711E-04	FYY	-0.415358E-05	EXY	0.634532E-05
50	EXX	-0.590217E-07	EYY	0.158907E-06	EXY	0.117140E-04
51	EXX	-0.140169E-04	EYY	0.432762E-05	FXY	0.665423E-05
52	EXX	0.101147E-04	EYY	-0.294495E-05	FXY	0.557850E-05
53	EXX	-0.485830E-06	FYY	0.682525E-06	EXY	0.116680E-04
54	EXX	-0.990059E-05	FYY	0.316152E-05	EXY	0.746812E-05
55	EXX	0.634673E-05	FYY	-0.975146E-06	EXY	0.368628E-05
56	EXX	-0.129313E-05	EYY	0.497957E-07	EXY	0.130475E-04
57	EXX	-0.517458E-05	EYY	0.132602E-05	EXY	0.798141E-05
58	EXX	0.514998E-05	EYY	-0.103288E-04	EXY	0.103502E-04
59	EXX	0.593642E-06	EYY	-0.483149E-05	FXY	0.104208E-04
60	EXX	-0.932958E-06	EYY	-0.715996E-06	EXY	0.394471E-05

LOADING - 1

VERTICAL CENTER OF BEAM

RESULTANT JOINT DISPLACEMENTS - SUPPORTS

JOINT		-----DISPLACEMENT-----		
		X DISP.	Y DISP.	Z DISP.
1	GLOBAL	0.0	0.0	
81	GLOBAL	0.0031007	0.0	

RESULTANT JOINT DISPLACEMENTS - FREE JOINTS

JOINT		-----DISPLACEMENT-----		
		X DISP.	Y DISP.	Z DISP.
2	GLOBAL	0.0010615	-0.0001056	
3	GLOBAL	0.0020369	-0.0001489	
4	GLOBAL	0.0030187	-0.0001567	
5	GLOBAL	0.0000434	-0.0011495	
6	GLOBAL	0.0010696	-0.0011472	
7	GLOBAL	0.0020347	-0.0011521	
8	GLOBAL	0.0030115	-0.0011515	
9	GLOBAL	0.0001023	-0.0021597	
10	GLOBAL	0.0010741	-0.0021718	
11	GLOBAL	0.0020173	-0.0021664	
12	GLOBAL	0.0029772	-0.0021537	
13	GLOBAL	0.0001807	-0.0031288	
14	GLOBAL	0.0010968	-0.0031462	
15	GLOBAL	0.0019897	-0.0031448	
16	GLOBAL	0.0029058	-0.0031259	
17	GLOBAL	0.0002858	-0.0040380	
18	GLOBAL	0.0011314	-0.0040622	
19	GLOBAL	0.0019545	-0.0040620	
20	GLOBAL	0.0028009	-0.0040377	
21	GLOBAL	0.0004205	-0.0048684	
22	GLOBAL	0.0011765	-0.0048987	
23	GLOBAL	0.0019096	-0.0048987	
24	GLOBAL	0.0026657	-0.0048684	
25	GLOBAL	0.0005853	-0.0055992	
26	GLOBAL	0.0012316	-0.0056355	
27	GLOBAL	0.0018547	-0.0056352	
28	GLOBAL	0.0025006	-0.0055988	
29	GLOBAL	0.0007806	-0.0062106	
30	GLOBAL	0.0012962	-0.0062525	
31	GLOBAL	0.0017493	-0.0062516	
32	GLOBAL	0.0023061	-0.0062086	
33	GLOBAL	0.0010084	-0.0066810	
34	GLOBAL	0.0013694	-0.0067300	
35	GLOBAL	0.0017128	-0.0067274	
36	GLOBAL	0.0020839	-0.0066798	
37	GLOBAL	0.0012682	-0.0069632	
38	GLOBAL	0.0014547	-0.0070417	

39	GLOBAL	0.0016231	-0.0070529
40	GLOBAL	0.0018359	-0.0069936
41	GLOBAL	0.0015502	-0.0070887
42	GLOBAL	0.0015503	-0.0071525
43	GLOBAL	0.0015503	-0.0071852
44	GLOBAL	0.0015503	-0.0072172
45	GLOBAL	0.0018323	-0.0069033
46	GLOBAL	0.0016458	-0.0070418
47	GLOBAL	0.0014775	-0.0070530
48	GLOBAL	0.0012648	-0.0069937
49	GLOBAL	0.0020921	-0.0066612
50	GLOBAL	0.0017311	-0.0067302
51	GLOBAL	0.0013878	-0.0067276
52	GLOBAL	0.0010167	-0.0066800
53	GLOBAL	0.0023199	-0.0062109
54	GLOBAL	0.0018043	-0.0062528
55	GLOBAL	0.0013113	-0.0062519
56	GLOBAL	0.0007946	-0.0062089
57	GLOBAL	0.0025152	-0.0055945
58	GLOBAL	0.0018690	-0.0056358
59	GLOBAL	0.0012459	-0.0056356
60	GLOBAL	0.0006000	-0.0055992
61	GLOBAL	0.0026801	-0.0048687
62	GLOBAL	0.0019240	-0.0048990
63	GLOBAL	0.0011910	-0.0048991
64	GLOBAL	0.0004348	-0.0048688
65	GLOBAL	0.0028148	-0.0040384
66	GLOBAL	0.0019691	-0.0040625
67	GLOBAL	0.0011461	-0.0040623
68	GLOBAL	0.0002996	-0.0040380
69	GLOBAL	0.0029199	-0.0031291
70	GLOBAL	0.0020038	-0.0031465
71	GLOBAL	0.0011108	-0.0031451
72	GLOBAL	0.0001947	-0.0031262
73	GLOBAL	0.0029964	-0.0021599
74	GLOBAL	0.0020265	-0.0021720
75	GLOBAL	0.0010832	-0.0021666
76	GLOBAL	0.0001233	-0.0021539
77	GLOBAL	0.0030573	-0.0011496
78	GLOBAL	0.0020310	-0.0011473
79	GLOBAL	0.0010658	-0.0011522
80	GLOBAL	0.0000889	-0.0011516
82	GLOBAL	0.0020391	-0.0001056
83	GLOBAL	0.0010636	-0.0001490
84	GLOBAL	0.0000818	-0.0001568

FINISH

GOOD-BYE

Comments

In comparing the output from the "INTROFEM" and the STRUDL-II solutions, it can be seen that the results compare quite favorably with each other. While there are differences between the displacement and stress values produced by the two programs, these differences are generally less than 10%. One possible reason for these differences is the differing mathematical formulations of the "4-CST" element and the "PSR" element.

The abundance of information produced by a finite element analysis is illustrated by comparing the handbook calculation with the finite element solutions. In order to obtain one piece of information (the mid-point deflection of the beam) with the handbook method, a calculation was necessary. For each additional piece of information, an additional calculation would be necessary. In contrast, the effort and time to perform the finite element analysis remains relatively constant whether one or many resultant values are sought. To match the information available from a finite element analysis using handbook formulas would require much more time and effort than that required to use a finite element program. In larger problems with more complicated geometries, the difference would be even greater, if comparable information could be obtained from handbook formulas at all.

Chapter 9

CLOSURE

Summary

This paper has attempted to approach the finite element method of structural analysis from two viewpoints. First, the viewpoint of the manager who might be considering the outlay of corporate resources in order to obtain a finite element capability in his company's design process. Secondly, the viewpoint of the engineer or technician who will use a finite element capability in analyzing design alternatives under varying conditions of stress, strain, vibration, etc.

To the manager, the finite element method provides a means for modeling designs from the engineering department in a computer instead of the workshop. It allows testing of the design to be carried out in a computer instead of on a test track, in a test room, or in the field. It allows alternative designs to be more accurately evaluated at an earlier stage in their development so that there is a better chance that the best design will be committed to a prototype. In short, using finite element methods of analysis allows a better design to be produced in a shorter time at a lower cost.

Finite element methods are being used in many applications today. They can be integrated into the design process in a number of ways, depending upon the resources available and the need for this advanced analysis capability. There are programs and hardware available now with

impressive capabilities, and the future is certain to bring about more advanced and accurate programs with lower user costs.

To the engineer or technician, finite element methods provide much more information about the behavior of a design under test conditions than could be gathered by conventional methods. They apply tried concepts of materials mechanics to structures and designs which are beyond the capabilities of traditional methods to analyze. And, through the use of digital computers, this analysis is carried out with comparably little effort in a short time period.

The finite element method augments and expands the power of the engineer's experience. A good finite element model depends upon the skill and insight of its modeler. The method cannot replace the engineer's creativity or experience. It can reduce his uncertainty and actually decrease his work load.

Conclusion

It appears, then, that the finite element method provides a means for improving the output of the design process. Computer programs are available today to implement this methodology. These programs are capable of providing accurate answers to formerly unsolvable types of analysis problems.

Although the cost of using finite element methods appears to be fairly high, the cost of not using them can be even higher for a firm in today's competitive world of increasing consumer demands, shrinking margins, and scarce resources. In short, finite element method capabilities are worth investigating for any firm involved in producing products which operate under conditions of load, stress, vibration, temperature changes, etc.

Implications and Recommendations

Finite element methods are a new application of some very well documented concepts of mathematics and mechanics. The literature on the subject is growing rapidly, almost on a daily basis. As more and more users of the methodology come into being, the need for people aware and able to use finite element methods will increase. Instruction in finite element theory and methodology cannot be limited to the graduate level engineer. The use of finite element methods, as well as other methods of computer aided design, will have to be incorporated into the already crowded undergraduate engineering and technical curriculums.

This paper has hardly scratched the surface of the material available on the finite element method. Further work could easily be done on any aspect of this subject. Some of these areas are listed below:

1. Mechanics: Statics, Dynamics, Strength of Materials, Theory of Structures, etc.
2. Mathematics: Matrix Algebra, Tensors, Variational Calculus, etc.
3. Computers: Programming, Languages, Hardware, etc.
4. Finite Elements: Formulations of Elements, Isoparametric Elements, Higher Level Elements, Axisymmetric Elements, etc.
5. Programs: Advanced use of ICES STRUDL-II, use of other finite element programs, new developments in program capabilities, etc.
6. Other aspects of computer aided design.

BIBLIOGRAPHY

- Bernstein, Murray, and others. "NASTRAN Analysis of the 1/8-Scale Space Shuttle Dynamic Model," NASTRAN: User's Experiences. Compendium of papers prepared for the Third NASTRAN Users' Colloquium, NASA Langley Research Center, September 11-12, 1973. Report number NASA TM X-2893. Springfield, Virginia: National Technical Information Service, 1973.
- Butler, Thomas G., and Douglas Michel. NASTRAN: A Summary of the Functions and Capabilities of the NASA Structural Analysis Computer System. Report number NASA SP-260. Washington: Government Printing Office, 1971.
- Carl, E. J., and W. C. Hamann. How Finite Element Methods are Introduced in Large and Small Organizations. New York: Society of Automotive Engineers, #740006, SP-387, 1974.
- Clark, J. N., D. F. Barnhart, and W. F. Hayes. Stress Analysis of Industrial Components with Plastic and Finite Element Models. Warrendale, Pennsylvania: Society of Automotive Engineers, #740706, 1974.
- Conner, J., and G. Will. Computer-Aided Teaching of the Finite Element Displacement Method. Cambridge, Massachusetts: School of Engineering, Massachusetts Institute of Technology, 1969.
- Davis, Charles S. Illustrations of Automotive Finite Element Models-- Statics. New York: Society of Automotive Engineers, #740004, SP-387, 1974.
- Davis, R. L. How Finite Element Methods Improve the Design Cycle. New York: Society of Automotive Engineers, #740003, SP-387. 1974.
- Desai, Chandrakant S., and John F. Abel. Introduction to the Finite Element Method. New York: Van Nostrand Reinhold Company, 1972.
- Hamann, W. C. "Finite Element Methods in Product Design," Mechanical Engineering, February 1974, pp. 30-36.
- Jones, C. F. Personal letter to Richard Ryner, September 20, 1974.
- Kardestuncer, Hayrettin. Elementary Matrix Analysis of Structures. New York: McGraw-Hill, 1974.
- Lockheed Missiles and Space Company. Lockheed Structural Analysis Service (brochure). Palo Alto, California: Lockheed Missiles and Space Company. (Mimeographed.)

- Logcher, Robert D., Jerome J. Connor, Jr., and Mark F. Nelson. ICES-STRUDL-II: The Structural Design Language Engineering User's Manual. Vol. II, Additional Design and Analysis Facilities. Cambridge, Massachusetts: School of Engineering, Massachusetts Institute of Technology, 1972.
- Logcher, Robert D., and others. ICES STRUDL-II: The Structural Design Language Engineering User's Manual. Vol. I, Frame Analysis. Cambridge, Massachusetts: School of Engineering, Massachusetts Institute of Technology, 1968.
- MacNeal, Richard H., ed. The NASTRAN Theoretical Manual (Level 15.5). NASA SP-221. Washington: National Aeronautics and Space Administration, 1972.
- Meek, John L. Matrix Structural Analysis. New York: McGraw-Hill, 1971.
- Minard, Keith S. Personal letter to Richard Ryner, October 24, 1973.
- Noble, Ben. Applied Linear Algebra. Englewood Cliffs: Prentice-Hall, 1969.
- Oberg, Erik, and Franklin Jones. Machinery's Handbook, ed. H. L. Horton, H. L. Horton, P. B. Schubert, and G. Garratt. New York: Industrial Press, 1971.
- Salus, W. L., R. E. Jones, and M. W. Ice. "Dynamic Analysis of a Long Span, Cable-Stayed Freeway Bridge Using NASTRAN," NASTRAN Users' Experiences. Compendium of papers prepared for the Third NASTRAN Users' Colloquium, NASA Langley Research Center, September 11-12, 1973. Report number NASA TM X-2893. Springfield, Virginia: National Technical Information Service, 1973.
- Schumacker, Betsy. An Introduction to ICES. Cambridge, Massachusetts: School of Engineering, Massachusetts Institute of Technology, 1967.
- Sherlock, James E., and Joseph G. Kormos. "Dynamic Analysis of Complex Machinery." Paper presented at the meeting of the Mississippi Valley Section, Society of Automotive Engineers, Dubuque, Iowa, January 25, 1973. Cincinnati, Ohio: SDRC-Systems, Inc., 1973.
- Smith, Gary L. Automotive Use of Finite Element Methods--Introduction and Overview. New York: Society of Automotive Engineers, SP-387, 1974.
- Stehling, Kurt R. Computers and You. New York: New American Library, 1972.
- Thompson, J. E. Future Developments in Structural Analysis. New York: Society of Automotive Engineers, #740008, SP-387, 1974.
- Timoshenko, Stephen P., and James M. Gere. Mechanics of Materials. New York: Van Nostrand Reinhold, 1972.

- Ural, Oktay. Finite Element Method. New York: Intext Educational Publishers, 1973.
- Vail, Curtis F. Illustrations of Automotive Finite Element Models--Dynamics. New York: Society of Automotive Engineers, #740005, SP-387, 1974.
- Wilkinson, Michael T., and Arthur C. Bruce. "Structural Analysis of Light Aircraft Using Nastran," NASTRAN: Users' Experiences. Compendium of papers prepared for the Third NASTRAN Users' Colloquium, NASA Langley Research Center, September 11-12, 1973. Report number NASA TM X-2893. Springfield, Virginia: National Technical Information Service, 1973.
- Zienkiewicz, O. C. The Finite Element Method in Engineering Science. London: McGraw-Hill, 1971.