rt No. FHWA/RD-80/027

W STRUCTURAL SYSTEMS FOR ZERO – MAINTENANCE PAVEMENTS

Vol. 2. Analysis of Anchored Pavements Using ANSYS

August 1980

Final Report

Document is available to the public through the National Technical Information Service, Springfield, Virginia 22161

Prepared for

FEDERAL HIGHWAY ADMINISTRATION
Offices of Research & Development
Structures and Applied Mechanics Division
Washington, D.C. 20590

FOREWORD

This report provides a set of procedures to evaluate the response of an anchored pavement subjected to static vehicle loads, moisture variation in the subgrade, and/or temperature variation through the surface of the pavement. Basically, these procedures consist of the use of two computer programs known as FEMESH and ANSYS. The FEMESH program divides the analytical model into a set of rectangular elements and the ANSYS program evaluates the stresses, strains, and deflections at each of these elements in each material included in the analytical model. The procedures are versatile and capable of solving geometrically complex structures on a geologically complex earth mass.

This report is the second volume of a set of three final reports resulting from a research contract, "New Structural Systems for Zero-Maintenance Pavements," issued to Dames & Moore by the Office of Research and Development of the Federal Highway Administration. The objective of this research study was to identify and assess the potential of new and innovative structural concepts and systems to serve as "Zero-Maintenance" pavements. An interim report, "Unique Concepts and Systems for Zero Maintenance Pavements," FHWA-RD-77-76, provides an updated state-of-the-art and comprehensive review of each of the three major structural components of a pavement system: the subgrade, the base and subbase, and the pavement surface. The other two volumes in this final set are reports FHWA/RD-80/026, Volume 1: Analytical and Experimental Studies of an Anchored Pavement, and FHWA/RD-80/028, Volume 3: Anchored Pavement System Designed for Edens Expressway. Volume 1 was published and distributed previously.

Copies of Volumes 2 and 3 are being distributed jointly by a single transmittal memorandum primarily to research and development audiences.

Richard E. Hay / for
Charles F. Scheffey
Director, Office of Research
Federal Highway Administration

NOTICE

Technical Report Documentation Page

1. Report No.	2. Government Accession No.	3. Recipient's Catalog No.
FHWA/RD-80/027		
4. Title and Subtitle NEW STRUCTURAL SYSTEMS FOR ZERO-MAINTENANCE PAVEMENTS Volume 2: Analysis of Anchored Pavements Using ANSYS		5. Report Date August 1980 6. Performing Organization Code
7. Author(s) S.K. Saxena and S.G. Militsopoulos		8. Performing Organization Report No.
9. Performing Organization Name and Address Dames & Moore 7101 Wisconsin Avenue Washington, D.C. 20014 and Illinois Institute of Technology Civil Engineering Department Chicago, Illinois 60616		10. Work Unit No. (TRAIS) FCP 35E2-042 11. Contract or Grant No. DOT-FH-11-9114
12. Sponsoring Agency Name and Address Offices of Research and Development Federal Highway Administration U.S. Department of Transportation Washington, D.C. 20590		13. Type of Report and Period Covered Final Report 14. Sponsoring Agency Code
15. Supplementary Notes FHWA Contract Manager: Dr. Floyd J. Stanek (HRS-14) Dames & Moore Project Manager: Dr. Mysore S. Nataraja		

16. Abstract New Structural Systems for Zero-Maintenance Pavements. The purpose of this study is to investigate the feasibility of designing and constructing cost-effective "Zero-Maintenance" highways.

Volume 2: Analysis of Anchored Pavements Using ANSYS. This report is a manual which provides a set of procedures to evaluate the response of an anchored pavement subjected to vehicle static loads, moisture variation in the subgrade, and/or temperature variation through the surface of the pavement. These procedures include two computer programs known as FEMESH and ANSYS. The FEMESH program generates rectangular meshes in either a two or three dimensional coordinate system for any prespecified number and spacing of nodes. The ANSYS program evaluates the stresses, strains, and the deflections at all elements in each material included in the analytical model. The program can be used for any number of different materials in any direction. In the analysis of heat transfer, the program provides the distribution of temperature as a function of time at predesignated points. The program is versatile and capable of solving complex geometrical structures supported on a geologically complex earth mass. The behavior of an anchored pavement section is evaluated with sets of computer programmed mechanistic models. The manual was written to minimize reference to other publications.

This volume is the second in a series. The others in the series are: Volume 1: Analytical and Experimental Studies of an Anchored Pavement, and Volume 3: Anchored Pavement System Designed for Edens Expressway. Abstracts of these volumes are included on page ii of this volume.

17. Key Words	18. Distribution Statement
Anchored pavement pavement systems FEMESH and ANSYS	No restrictions. This document is available to the public through the National Technical Information Service, Springfield, Virginia 22161.

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No. of Pages	22. Price
Unclassified	Unclassified	144	

Form DOT F 1700.7 (8-72) Reproduction of completed page authorized

Abstracts of Related Documents

Volume 1: Analytical and Experimental Studies of an Anchored Pavement: A Candidate Zero-Maintenance Pavement

This report documents an investigation of the design feasibility and construction cost-effectiveness of an anchored pavement concept for zero maintenance highways. An analytical model is designed to verify computer program results and to investigate construction methods for a full-scale highway section. The purpose of the analytical study is (1) to present thermal, mechanical, and thermomechanical properties of typical materials in a form easily adaptable to computer programs, and (2) to describe environmental and mechanical properties of a conventional slab and an anchored pavement in both continuous and jointed configurations. The two pavements were subject to heat transfer, thermal stress, and mechanical stress analyses. The anchored pavement offers two distinct advantages over a conventional pavement--deflections are lower and more uniform, and stresses in the soil are lower and distributed more widely by the rigid anchors. Subgrade-related failure is less likely to occur if loads are transmitted deeper within the subgrade. Three-dimensional finite element analysis is considered to be the most efficient technique for examining the significance of environmentally induced stress. The use of the finite element method is anticipated as more advanced analytic techniques are developed.

Volume 3: Anchored Pavement System Designed for Edens Expressway

This report provides an analysis example of an actual pavement and the cost estimate using the anchored system. The actual pavement is the Edens Expressway in Chicago. The report provides the response of the Edens Expressway subjected to mechanical and environmental loads using the anchored pavement concept. The mechanical and thermal properties of materials that could be encountered in future reconstruction of Edens Expressway are presented in a consistent form for computer programming. These properties are viewed as typical design values during investigation of pavement response. The behavior of the anchored pavement under induced temperature loads and weakening of subgrade (by thawing action) is clearly demonstrated. This report will enable application of the anchored pavement concept by any road with heavy traffic. The example problem provides the input parameters of materials and loads for the analysis, the generation of finite element mesh, and the results of the analysis. The computer program ANSYS was used for this study (the manual for the use of the program is presented in Vol. 2 of this series of reports).

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

CHAPTER 1
INTRODUCTION

1.1 OBJECTIVE

The objective of this manual is to provide the pavement analyst with a ready reference of procedures to obtain the response of an anchored pavement subject to vehicle static loads, moisture variation of the subgrade, and temperature variation at the surface of the pavement.

1.2 SCOPE

The analysis procedures presented include two computer program packages known as FEMESH and ANSYS. An anchored pavement section of known geometry is chosen, and its behavior is evaluated by sets of mechanistic models which have been computer programed.

A subbase and a subgrade material system of known properties are also evaluated.

Figure 1.1 shows an interface connection between a finite pavement element and a finite soil element. Those interface elements transmit compression forces, but they don't take any tension forces (that is, disconnect in tension).

The manual is composed of six chapters and three appendices.

1. INTRODUCTION
2. ANALYSIS PROCEDURE
3. OPERATING INSTRUCTIONS FOR FEMESH
4. OPERATING INSTRUCTIONS FOR ANSYS
5. ELEMENT LIBRARY FOR ANSYS
6. CONCLUSION

APPENDIX A - NOTATIONS
APPENDIX B - THEORETICAL BACKGROUND FOR ANALYSIS METHODS
APPENDIX C - THEORETICAL BACKGROUND FOR ELEMENT LIBRARY

Each chapter has been organized to provide the user a procedure in order to collect the necessary data and run the program ANSYS. Chapter 2 and 3 describe the necessary steps for computer familiarization and data collection. Chapter 4, the user's guide for ANSYS, has been written so that it can be used independently by the computer analyst. Chapter 5 outlines the elements recommended to be used in ANSYS.

1.3 RELATED DOCUMENTS

The manual is developed with the intent of minimizing the amount of reference to other materials. However, references 7 and 8 should be consulted if pre-processing or post-processing routines are desired.

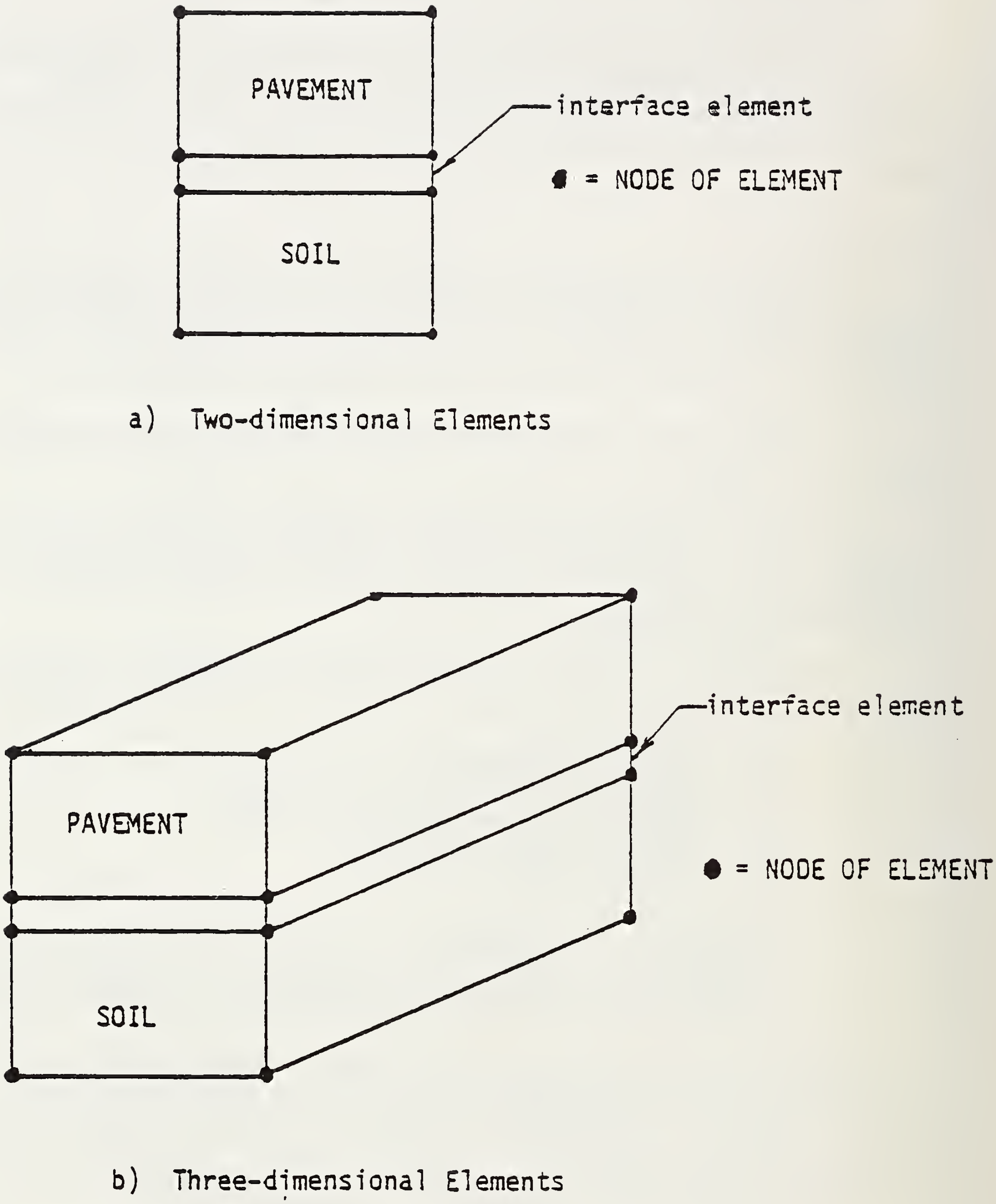


Figure 1.1 Soil-Structure Interface Connection

CHAPTER 2
ANALYSIS PROCEDURE

2.1 COMPUTER CODES USED

The software used to conduct the analytical investigation included two programs - one for mesh generation (elements and nodes) with the name FEMESH and one for the actual analysis called ANSYS.

The former was written as a general mesh generator with several criteria in mind: ease of use, minimization of input data required, and ability to generate any two or three dimensional rectangular mesh of arbitrary number and spacing of nodes in the x, y, and z directions (Fig. 2.1). Format for the output of nodal coordinates and element data is consistent with either ANSYS (Engineering ANalysis SYStem developed by Swanson) or SAP4 (Structural Analysis Program developed at Berkeley).

ANSYS is a proprietary general used, large scale, finite element code with great versatility. Static, heat transfer (steady state and transient), dynamic (modal, forced vibration), electrical, and non-linear (geometric, elastoplastic material, creep) analyses are possible using a large scale element library (a variety of more than 60 elements) comprised of two and three dimensional elements.

2.2 PREPARATION OF COMPUTER INPUT

The computer input consists of the system control cards and the ANSYS data deck as shown in Fig. 2.2. The Cyber 176 System shown in Fig. 2.3 was used in connection with the work at I.I.T.

2.2.1 System Control Cards

The first card of an input file is interpreted as a NOSBE jobcard and must be of the following format:

XXXXX,PARAMETER STRING.CHARGE,USERNAME   COMMENTS

Where XXXXX Job name, must begin with a letter. Other characters may be alphanumeric. Names longer than 5 characters will be truncated to 5. Jobs submitted through INTERCOM have only the first 3 characters preserved.

All of the following parameters are optional, and have default values if not specified. Parameters may be in any order and are separated by commas.

TN N is a decimal value for CP Time Limit in seconds. Default is 10.

CMFL — FL is the maximum field length in octal words required by the job. It is recommended that if the default FL is sufficient to process all steps within a job, the CMFL parameter be omitted from the job card. See -SYSTEM DEFAULT VALUES AND LIMITS- for the default CMFL allocated to each job.

ECFL — FL is the maximum large core field length in octal 1K word blocks required by the job.

PJ — J is the requested priority value and ranges from 1 to 5. Default is 4.

MTN — N is the number of 7 track tape drives reserved by the job.

PEL or NTL — L is the number of 9 track 1600BPI tape drives that will be required by the job.

HDL — L is the number of 9 track 800BPI tape drives that will be required by the job.

GEL — L is the number of 9 track 6250BPI tape drives that will be required by the job.

DYYMM — YY is the dependency string identifier and MM is the dependency count.

The charge number is a six digit (leading zeros must be present) account number, OPTIONALLY followed by a 1 to 3 character suffix for extended accounting. The extended accounting is used for sorting when the Billing detail of run is provided monthly. The username is given as a "Last Name,Initials." The initials are one or two of your choice as specified at account initialization and entered into the System Access Authorization Table. Any difference between the jobcard entry and the table entry will cause the job to abort. Note that blanks are suppressed when scanning the jobcard, so if only one initial is used, it must be followed by a comma if subsequent comments are placed on the jobcard. If it is not, the first character of the comment will be picked up as a second initial, and job abort will occur.

EXAMPLES:

RUNID,T10,P4.264786ABC,MILTON,JE.	Test Run
RUNID,T10,P4.264786,MILTON,JE.	Comment - No Extended Accounting
RUNID,T10,P4.264786,MILTON,J,	If previously set up with one initial

The rest of the control cards to call the 2nd revision of ANSYS are as follows:

```
ATTACH(TAPE22,R2ANSYS)
COPYBR(TAPE22,ANSFT)
COPYBR(INPUT,DATA)
ANSET(DATA)
FILE,TAPE11,BT=C,RT=W,MBL=5120,FO=SQ,SPR=YES,USE.
RFL(XXXXXX)          (XXXXXX MUST BE AT LEAST 170000 OCTAL WDS)
LDSET(PRESET=ZERO,MAP=S/ANSMAP,STAT=TAPE11)
SATISFY,BAMLIB.
TAPE20(DATA)
7/8/9
(ANSYS DATA)
7/8/9
6/7/8/9
```

For big jobs, including three-dimensional elements, it is advised that the 3rd revision/extended core version of ANSYS is used. The control cards for the above revision of ANSYS are as follows:

```
JOBCD,---,ECXXX.   (XXX- NUMBER OF 1000 OCTAL WD ECS BLOCKS REQD)
    '
    '
    '
ATTACH,A,R3ANSYSECS.
LIBRARY,A.
RFL(XXXXXX)        (XXXXXX MUST BE AT LEAST 170000 OCTAL WDS)
FILE,TAPE11,BT=C,RT=W,MBL=5120,FO=SQ,SPR=YES,USE.
ANSYS.
7/8/9
 (ANSYS DATA)
7/8/9
6/7/8/9
```

2.2.2 <u>ANSYS Problem</u>

ANSYS input data is set up in a relatively simple fashion that makes learning the code quick and easy. Sequential sets of cards are lettered "A" through "S", "A" being the title card, "S" being a terminator.

For example, a previous run of FEMESH to generate nodes and elements would supply "F" and "E" cards, respectively. The general purpose of each card group when applied to heat transfer or stress analysis is as follows:

A - Title
B - Accounting and core size
C - Analysis options (control)
D - Element data (types, miscellaneous properties)
E - Elements
F - Nodes
H - Material properties

L - Load control
M - Load control
N - Specified displacements (specified temperatures for thermal analysis)
O - Specified forces (specified heat flow rate for thermal analysis)
P - Specified pressures (specified convection for thermal analysis)
Q - Temperatures (heat generation rate for thermal analysis)
S - Terminator

2.2.3 Login-Logout Procedures for Batch Users

Every BATCH user has a USERNAME and PASSWORD which allows access to the CYBER 176 computer facility.

BATCH USERS

1. Dial access Number for desired baud rate:____________________

2. When connection is established, set data phone to "DATA" and then replace hand set.

3. System will respond with:

```
ITEL CONTROL DATA INTERCOM 4.
DATE  MM/DD/YY
TIME  HH.MM.SS
PLEASE LOGIN
```

4. You type and send:

```
LOGIN,USERNAME,PASSWORD
```

5. System will respond with:

```
Date  LOGGED IN AT  Time
      WITH USED ID
      EQUIP/PORT
```

6. Hit Carriage Return (CR) and system will respond with:

```
LOGIN CREATED  Date
TODAY IS  Date
IMPORTANT SYSTEM INFORMATION MESSAGE....
COMMAND-
```

7. You are now ready to send and receive BATCH commands and messages. In particular, you can now read in card decks and print output from previously run jobs.

8. To submit a BATCH job to the system, place the card deck in the reader, make the reader ready, and type "R". The deck will then be read into the system.

9. After the last card of the deck has been successfully read, the system will once again respond with:

   COMMAND-

10. When a job is ready for printing, ready the printer and type "ON,LP". All jobs waiting in the output queue will then print until the queue is empty. The terminal will then return to COMMAND mode.

11. To disconnect the terminal from the system, type and send:

    LOGOUT

12. Restore dataset to "TALK" position; lift phone to check for dial tone and then replace handset.
    NOTE: This ensures that the phone is properly disconnected.

NOTE: The SUP parameter is optional and can be used on the LOGIN command (Step 4): LOGIN,USERNAME, PASSWORD,SUP. The use of the SUP parameter would result in the elimination of Step 5.

2.2.4 Remote Batch Terminal Commands

All BATCH terminal commands are documented in the INTERCOM V.4 Reference manual.

COMMAND	DESCRIPTION
H,I	Displays your jobs in CYBER 176 input queue
H,E	Displays your jobs in CYBER 176 execution queue
H,O	Displays your jobs in Cyber 176 output queue
FILES	Displays all jobs at your jobsite and the queue they are currently in
WAIT,LP	Suspends job currently printing
GO,LP	Continues a suspended print job
C	Resumes interrupted operation
R	Read cards
EVICT,--	(Last 2 letters of job name) Drop job from input/output queue before printing

E,LP	Kills job while printing
ON,LP	Turns line printer logically ON
REW	Rewinds current output file
BSP,LP,N	Backspace N of output file sectors
RTN,,P	Halts printing and returns job to output queue with priority P
M,msg	Send message to central site
REP,,N	Job is reprinted N additional times
PRIOR,--,----	(Last 2 characters of output file and then priority you want to raise it to).
STATUS,---	Status of job (from your site-ID) whose 1st 3 jobname characters are ---.
STATUS,,--	Status of all jobs from site-ID--.

2.3 ANALYSIS METHODS

This section is intended to give a brief summary of the methods used in the various types of analysis. It is not intended to be a complete theoretical manual or to answer all questions which may arise on the theory behind the ANSYS program. Such detail would expand the already voluminous User's Manual and is better included in a Theoretical Manual. Theoretical details may be obtained by contacting Swanson Analysis Systems, Inc.

Figure 2.4 gives a summary of the ANSYS Analy sis types available and may be used as a guide in selecting which type to use.

2.3.1 Static Analysis

In the matrix displacement method of analysis based upon finite element idealization, the structure being analyzed must be approximated as an assembly of discrete structural elements connected at a finite number of points (called nodal points). If the force-displacement relationship for each of these discrete structural elements is known (the element "stiffness matrix") then the force-displacement relationship for the entire structure can be assembled using standard matrix methods.

The general form of the equilibrium equations for each element is:

$$[K_e]\{U_e\} = \{F_e\} \tag{2.1}$$

where, $[K_e]$ is the element stiffness matrix

$\{U_e\}$ is a vector of the element nodal displacements, and

$\{F_e\}$ is a vector of the element nodal forces.

For the total structure:

$$[K] \{U\} = \{F\} \tag{2.2}$$

where, $[K]$ is the total structure stiffness matrix

$$\sum_{i=1}^{n} [K_e]$$

$\{U\}$ is a vector of all the nodal displacements in the structure

$\{F\}$ is a vector of all the corresponding nodal forces, thermal forces, and pressure forces

$$\sum_{i=1}^{n} \{F_e\}$$

If sufficient boundary conditions are specified on $\{U\}$ to guarantee a unique solution, equation 2.2 can be solved to obtain the nodal point displacements at each node in the structure. From these displacements the forces and stresses within each structural element can be calculated.

For plasticity and creep problems an incremental technique is used. The loading is applied in increments and at each loading level an elastic solution is done, with a correction applied to the next loading step to account for the plasticity and creep occurring during this loading step. In this procedure, the plasticity lags,the loading and the calculated stresses are somewhat higher than the true stresses. The amount of this conservative difference can be reduced by increasing the number of load increments or by running iterations with no increase in loading to refine the solution. Unloading and reversed loading can be handled with no difficulty by this technique. The von Mises yield surface is used, along with the Prandtl-Reuss flow relations. The stress-strain curve upon reversed loading is assumed to be the same shape as the virgin stress-strain curve, but offset to account for the strain due to previous plastic deformation. Kinematic or isotropic hardening rules are also available for the treatment of cyclic plasticity.

The program will handle creep by a similar incremental technique. Both primary and secondary creep equations are available to the user. The user has the option of selecting either a creep formulation which assumes the stresses decay due to the creep (as in thermal stresses), or a formulation in which the stresses are independent of creep (as in primary stresses).

The ANSYS program also includes irradiation induced swelling and creep for use in the analysis of nuclear reactor internals. The swelling is not stress dependent and is treated in a manner similar to thermal strains, while the irradiation creep is a stress and temperature dependent pheonomenon and requires an iterative solution.

For large deflection analysis the geometry is modified at the end of each load increment so that the total loading is applied to the deformed structure at the next load increment. This procedure thus follows the large deflection load-deflection curve.

If the load is applied to the structure in a single step and the rate of convergence to the large deflection is observed, an estimate of the stability of the structure can be made. In particular, if the deflection diverges, the load is above the critical buckling load. This large deflection analysis then becomes a stability check.

The basic equations for the formation of the element equilibrium equations are summarized in Table 2.1. The same definitions used here apply to all other analysis types except the heat transfer analysis.

2.3.2 Heat Transfer

Transient and steady state heat transfer problems can be solved by finite element techniques analogous to those used for structural analyses. In this case the basic equilibrium equation is:

$$[C]\{\dot{T}\} + [K]\{T\} = \{Q\} \qquad (2.3)$$

where, $[K]$ is the thermal conductivity matrix

$\{Q\}$ is the heat flow vector

$\{T\}$ is the vector of the nodal point temperatures

$[C]$ is the specific heat matrix

This equation is identical to the nonlinear dynamic equation except that the mass term does not exist. The solution technique is the same as for the dynamic analysis except that linear and quadratic options are available for this approximation function.

This equation is solved in ANSYS at each time point in the heat transfer transient. Material properties (and convection coefficients) can be a function of temperature. In a steady-state analysis the properties are evaluated at the temperature of the previous iteration. In a transient analysis the properties are evaluated at a temperature extrapolated from the previously calculated temperatures.

The temperature output from the ANSYS heat transfer analysis is in the required form for input to the ANSYS stress analysis, giving an integrated analysis capability.

The basic equations for the formation of the element equilibrium equations are given in Table 2.2.

2.4 LOAD CHARACTERISTICS

Pavements are subject to axle weight distributions produced by the traffic volume. Vehicle speed and load duration are not included in this report. The load input consists only of the static weight of an automobile, and the corresponding pavement response is evaluated.

Static load can be input as nodal forces (See Fig. 2.5) or element surface pressures. Environmental loads, however, cause more damage to the pavement. Moisture variation is handled by varying the modulus of elasticity of the top four feet of the subgrade soil (of course the variation of modulus with moisture content must be known as an input).

Temperature variation in a time domain can be input in a heat transfer model as shown in Fig. 2.6. The resulting temperature distribution can be handled as a thermal load for a static analysis.

2.5 MATERIAL CHARACTERIZATION

All material properties are listed in Table 2.3. Table 2.4 represents the material properties needed for the element library used in this report.

TABLE 2.1 EQUILIBRIUM EQUATIONS (Static Analysis) FOR AN ELEMENT

$$[K_e]\{U_e\} = \{P_e\} + \{Q_e\} + \{R_e\} + \{S_e\}$$

where

$$[K_e] = [TR]^T\,[H]^T \int_V [g]^T[C][g]\;dV\;\;[H]\,[TR]$$

$\{U_e\}$ = Nodal displacement vector (in global coordinates)

$\{P_e\}$ = Applied nodal load vector

$\{Q_e\} = [TR]^T[H]^T \int_V [g]^T\,[C]\{\varepsilon_{Th}\}\,dV$ = Thermal load vector

$\{R_e\} = [TR]^T[H]^T \int_A [e]^T\,\{P\}\,dA$ = Pressure load vector

$\{S_e\} = [M_e]\{A_c\}$ = Body force vector

also

$\{U\} = [TR]\{U_e\} = [H^{-1}]\{b\}$ = Nodal displacement vector in local coordinates

$[TR]$ = Geometrical transformation matrix

$[U]$ = Matrix relating the nodal displacement vector in local coordinates to the displacement function

$\{b\}$ = Vector of the coefficients of the displacement functions

$\{w\} = [e]^T\{b\}$ = Displacement functions

$[e]$ = Matrix of displacement shapes

$\{\varepsilon\} = [g]\{b\}$ = elastic strain vector

$\{\varepsilon_{Th}\}$ = Thermal strain vector

$[g]$ = Matrix relating the elastic strains to the displacement functions

$[C]$ = Elastic material property matrix

$\{\sigma\} = [C](\{\varepsilon\} - \{\varepsilon_{Th}\})$ = stress vector

$\{P\}$ = Distributed load vector

$$[M_e] = [TR]^T\,[H]^T \int_V \rho\,[e][e]^T\;dV\;\;[H][TR]$$

ρ = Density

$\{A_c\}$ = Acceleration vector

TABLE 2.2. EQUILIBRIUM EQUATIONS (Thermal Analysis) FOR AN ELEMENT

$$[\overline{C}_e]\{T_e\} + [\overline{K}_e]\{T_e\} = \{Q_e\}$$

where,

$[\overline{C}_e]$ = $[H]^T \int_V \rho \; C_p \{e\}\{e\}^T \; dV \; [H]$ = Specific heat matrix

$\{\dot{T}_e\}$ = Vector of time derivatives of nodal temperatures

$[\overline{K}_e]$ = $[H]^T \int_V [g]^T [k][g] \; dV \; [H] + [H]^T \int_S h\{e\}\{e\}^T \; dA \; [H]$

= conductivity matrix

$\{T_e\}$ = $[H^{-1}]\{b_t\}$ = Vector of nodal temperatures

$\{Q_e\}$ = $[H]^T \int_V \rho \; q\{e\} \; dV + [H]^T \int_S T_c \; h \; \{e\} \; dA$ = Element heat flow and heat generation vector

also,

$\{b_t\}$ = Coefficients of tempeture functions

Φ_{xT} = $\{e\}^T \{b_t\}$ = Temperature distribution over the element

$\{e\}$ = Vector of temperature distribution shapes

ρ = Density

C_p = Specific heat

$\{\Phi_{xT,i}\}$ = $[g_t]\{b_t\}$ = Vector of thermal gradients

$[g_t]$ = Matrix relating the thermal gradients to the temperature functions

$[k]$ = Conductivity material property matrix

q = Internal heat generation rate per unit mass

T_c = Coolant temperature

h = Convection coefficient

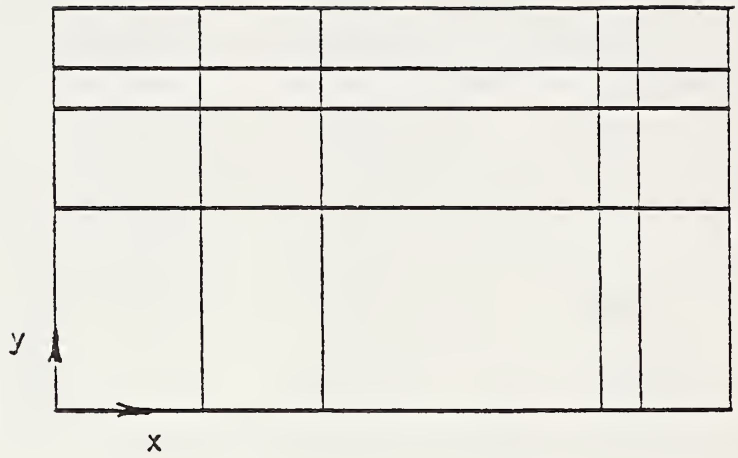
TABLE 2.3 MATERIAL PROPERTY NAMES

Property	Units	Description
EX	Force/Area	Elastic modulus, X direction
EY	Force/Area	Elastic modulus, Y direction
EZ	Force/Area	Elastic modulus, Z direction
ALPX	Strain/Temp	Coefficient of thermal expansion, X direction
ALPY	Strain/Temp	Coefficient of thermal expansion, Y direction
ALPZ	Strain/Temp	Coefficient of thermal expansion, Z direction
NUXY	----	Poisson's ratio (X strain due to Y stress)
NUYZ	----	Poisson's ratio (Y strain due to z stress)
NUXZ	----	Poisson's ratio (X strain due to Z stress)
DENS	Mass/Vol	Mass density
*C	Heat/Mass*Degree	Specific heat
*KXX	$\frac{\text{Heat * Length}}{\text{Time*Area*Degree}}$	Thermal conductivity, X direction
*KYY	$\frac{\text{Heat * Length}}{\text{Time*Area*Degree}}$	Thermal conductivity, Y direction
*KZZ	$\frac{\text{Heat * Length}}{\text{Time*Area*Degree}}$	Thermal conductivity, Z direction
*HF	$\frac{\text{Heat}}{\text{Time*Area*Degree}}$	Convection or film coefficient
*VISC	$\frac{\text{Force * Time}}{\text{Length}^2}$	Viscosity
MU	----	Coefficient of friction
GXY	Force/Area	Shear modulus, X-Y direction
GYZ	Force/Area	Shear modulus, Y-Z direction
GXZ	Force/Area	Shear modulus, X-Z direction
DAMP	----	K matrix multiplier for damping
*OHMS	$\frac{\text{Resistance*Area}}{\text{Length}}$	Electrical resistivity
*EMIS	----	Emissivity

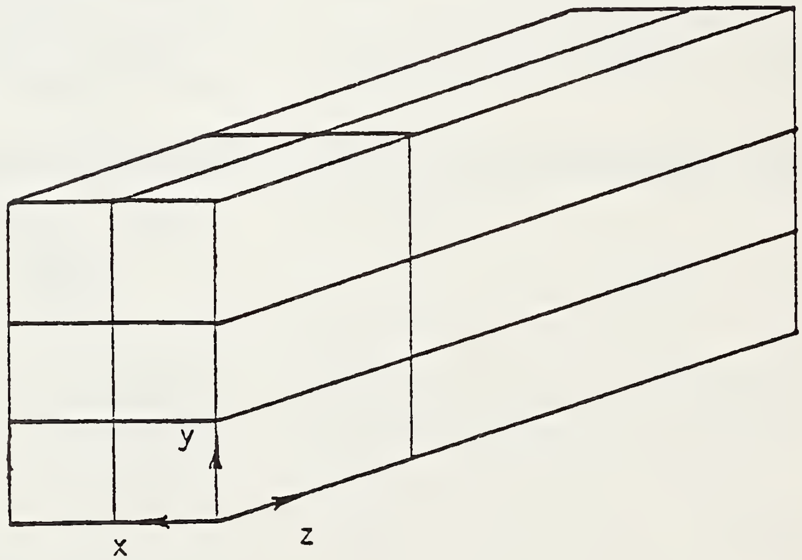
* Used only for the Thermal analysis (K20=-1)

TABLE 2.4 MATERIAL PROPERTIES VERSUS ELEMENT SUBROUTINE

Element Subroutine	Material Property
STIF42 OR STIF45	EX NUXY ALPX DENS
STIF12 OR STIF52	MU
STIF55	C KXX KYY DENS
STIF32	C KXX DENS



a) Two-dimensional Mesh



b) Three-dimensional Mesh

Figure 2.1 Arbitrary Rectangular Mesh Generation

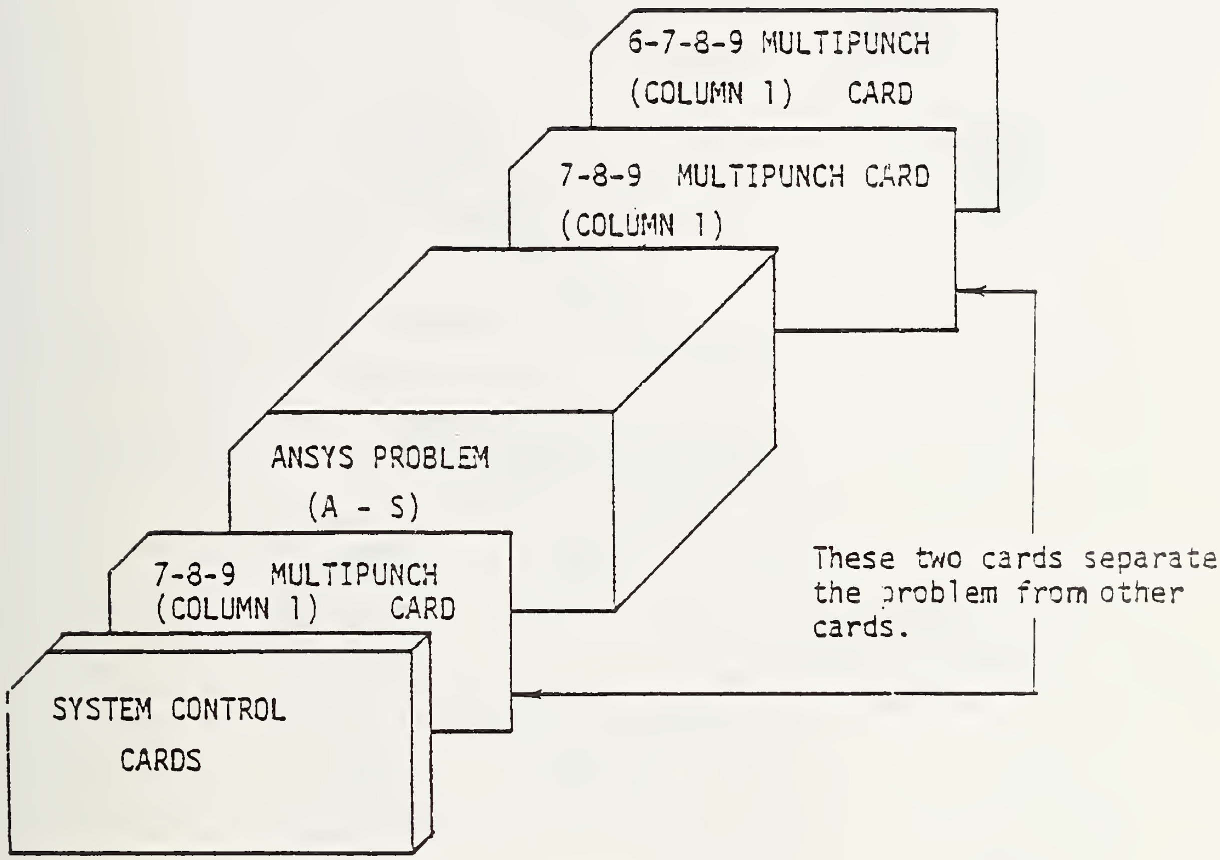


Figure 2.2 ANSYS Setup Deck For CDC Computer

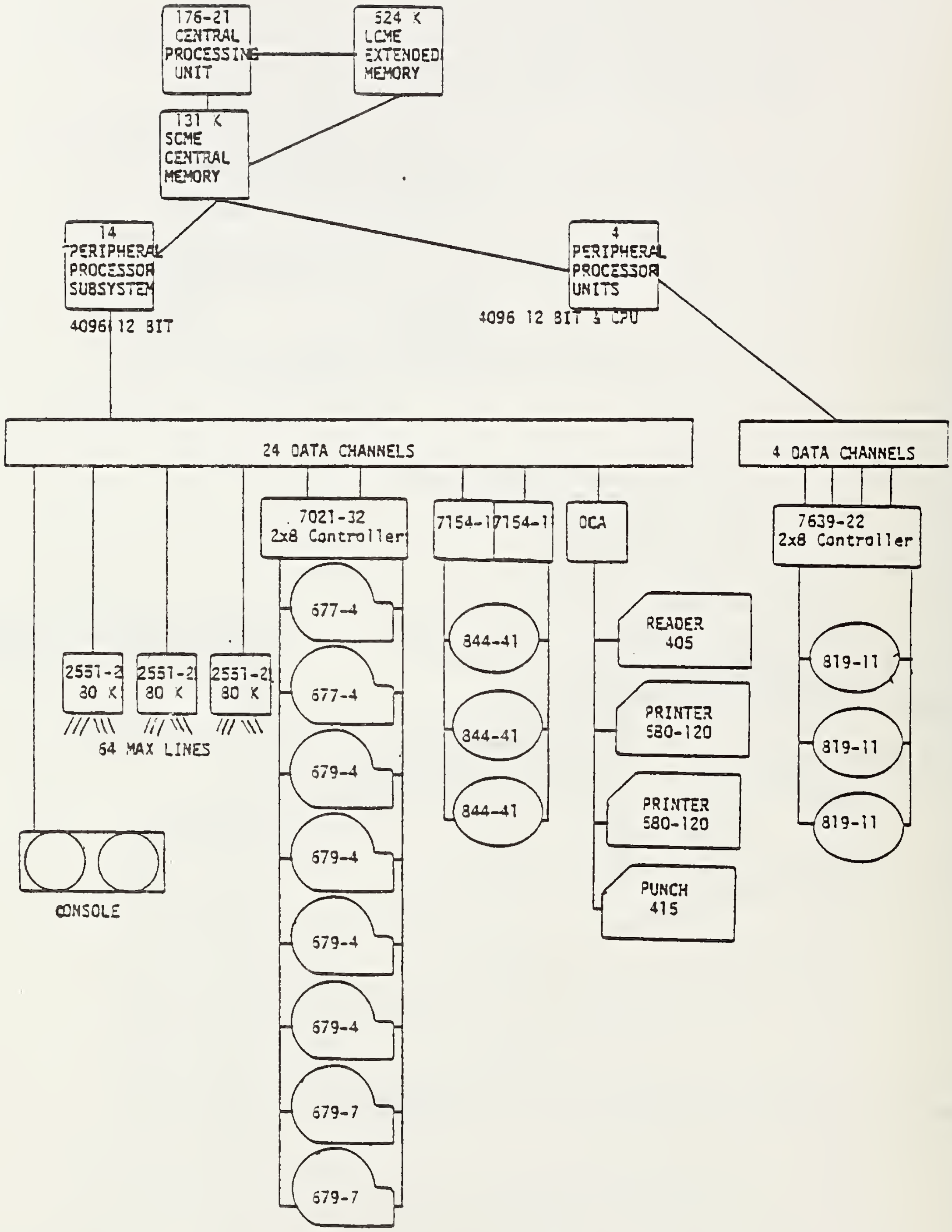


Figure 2.3 Cuber 176 Configuration

TRANSIENT AND
STEADY STATE
THERMAL ANALYSIS
(K20=-1)*

STATIC ANALYSIS,
LINEAR AND
NON-LINEAR
(K20=0)

NON-LINEAR
TRANSIENT DYNAMIC
ANALYSIS
(K20=4)

REDUCED LINEAR
DYNAMIC TRANSIENT
ANALYSIS
K20=5)

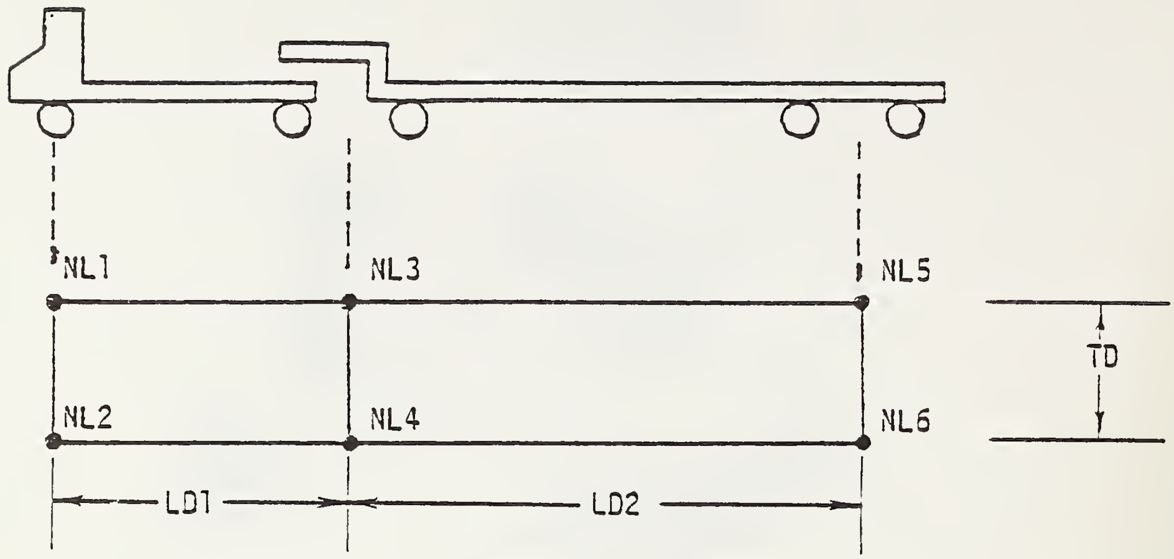
HARMONIC RESPONSE
ANALYSIS
(K20=3)

REDUCED HARMONIC
RESPONSE ANALYSIS
(K20=6)

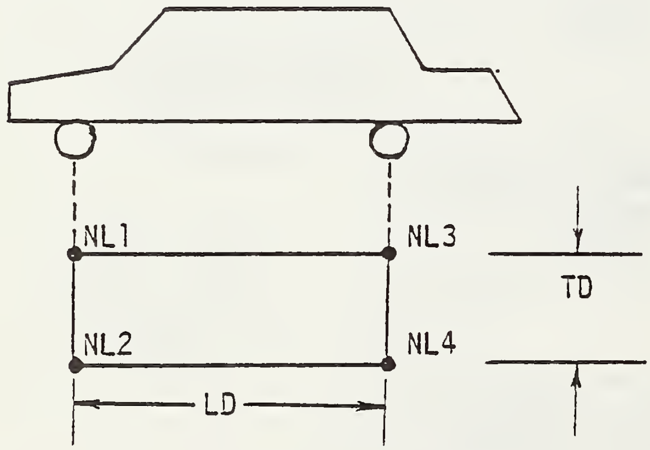
MODE-FREQUENCY
ANALYSIS
(K20=2)

* K20 is the key input on the C1 card to select the analysis type.

Figure 2.4 Summary of ANSYS Analysis Types



a) Truck Model



b) Passenger Auto Model

NL = Static Load on Node of Mesh
LD = Longitudinal Dimension of Automobile
TD = Transverse Dimension of Automobile

Figure 2.5 Static Load Model (Dimensions and Load Will Vary With Vehicle)

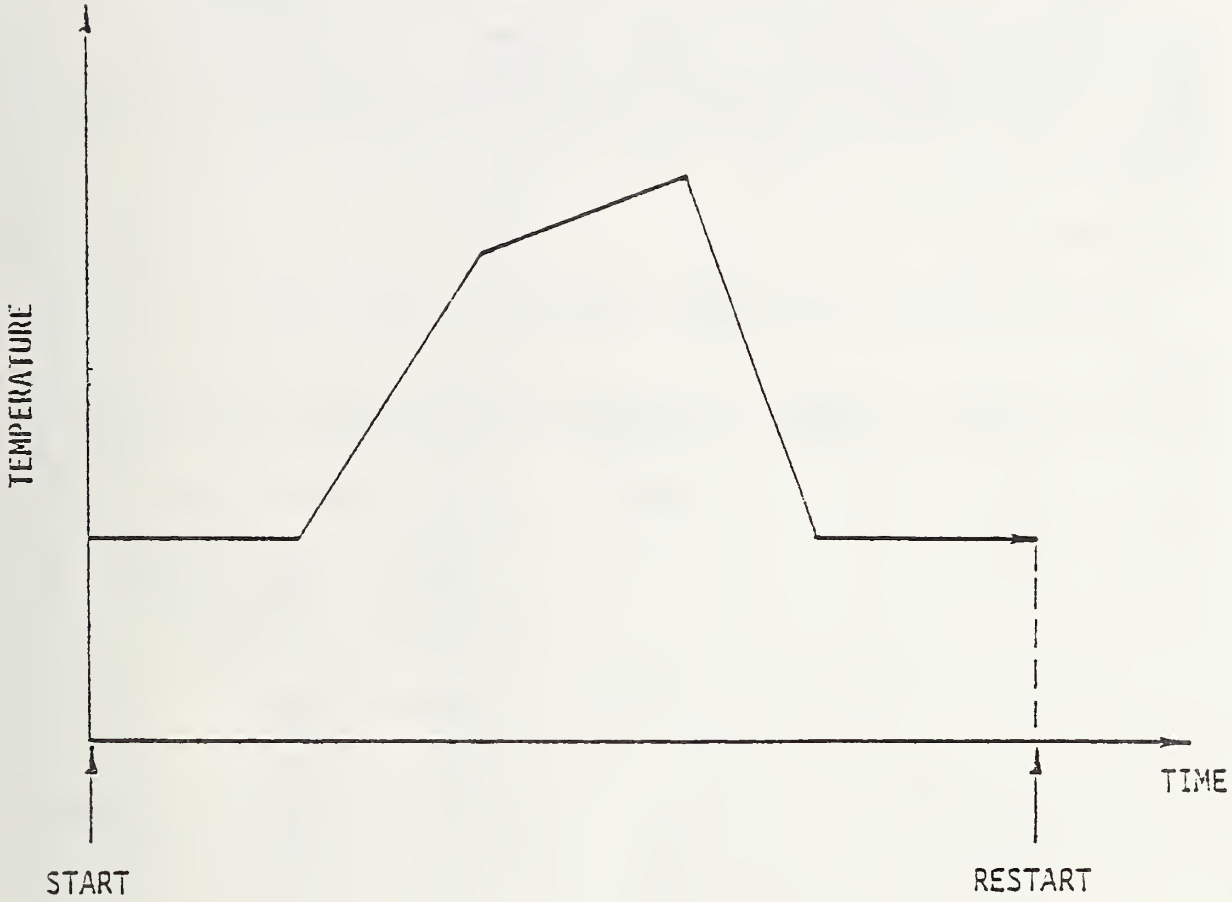


Figure 2.6 Temperature Load Model (actual values have been explained in the example problem discussed in Volume 3 of this series of reports entitled: Anchored Pavement System Designed for Edens Expressway)

CHAPTER 3
OPERATING INSTRUCTIONS FOR FEMESH

3.1 FEMESH SOURCE CODE

FEMESH was written with the specific intent of generating rectangular meshes for analysis of pavement systems. Provisions for including layers of varying materials is made. Output is available in SAP4 or ANSYS format, either as a punched card or directly written to a tape or mass storage file for use when the analysis is initiated (after data decks).

3.2 INPUT DATA

The following are the input formats for FEMESH data:

Card No.	Column	Format	Variable	Description
1	1-80	20A4	TITLE	Job Title
2	1-5	I5	NOPT	LT.0 generate nodes only
3	6-10	I5	N	Number of first node (useful when punching different meshes for assembly in ANSYS)
	1-5	I5	FORM	Output format .LE. 0 ANSYS .GT. 0 SAP4
	11-15	I5	NX	No. of nodes in x-direction
	16-20	I5	NY	No. of nodes in y-direction
	21-25	I5	NZ	No. of nodes in z-direction
	26-30	I5	NPUNCH	.GT. 0 Punched & Printed .LE. 0 Printed only
4	1-80	16I5	MAT	Material number
5	1-80	8F10.0	XP	x-coordinates of nodal lines
6	1-80	8F10.0	YP	y-coordinates of nodal lines
7	1-80	8F10.0	ZP	z-coordinates of nodal lines
8	-	-	-	Blank card

Note: Cards 5, 6, and 7 should be input in increasing order. Use as many 4, 5, 6, and 7 cards as necessary.

The listing of the program FEMESH is shown on the following pages (pages 23 through 29).

Listing of the Program FEMESH

```
C     PROGRAM TO GENERATE AND PUNCH 3D (OR 2D) RECTANGULAR MESHES IN
C     EITHER SAP4 OR ANSYSR3 FORMAT
C
C     INPUT PARAMETERS...
C
C        TITLE
C        NOPT        .LT.0  NODES ONLY
C        FORM        .GE. 0 SAP4
C                    .LT. 0 ANSYSR3
C        N           NUMBER OF FIRST NODE
C        NX,NY,NZ    NUMBER OF NODES IN X,Y,AND Z DIRECTIONS
C        NPUNCH      .EQ. 1  PUNCH CARDS, ELSE PRINT ONLY (DATA CHECK)
C        MAT...ARRAY OF LAYER MATERIAL NUMBERS,FROM BOTTOM TO TOP--(NY-1) LAYERS
C        XP,YP,ZP    ARRAYS CONTAINING COORD OF POINTS (PLANES) IN X,Y,Z
C                    DIRECTIONS
C
C
C
C
      INTEGER LR,FORM,ENN,BC
      DIMENSION ENN(8),NA(8),XP(50),YP(50),ZP(50),BC(6),TITLE(20)
      COMMON/EXTRA/MAT(50),NX,NY,NZ
      IP=1
      READ (5,102)  (TITLE(IX),IX=1,20)
      WRITE (6,196) (TITLE(IX),IX=1,20)
      READ (5,99) NOPT
      READ (5,100) FORM,N,NX,NY,NZ,NPUNCH
      NYM1=NY-1
      READ(5,105)(MAT(I),I=1,NYM1)
      READ (5,101) (XP(I),I=1,NX)
      READ (5,101) (YP(I),I=1,NY)
      READ (5,101) (ZP(I),I=1,NZ)
      IF (NPUNCH .NE. 1) GO TO 10
      WRITE (IP,102) (TITLE(IX),IX=1,20)
   10 CONTINUE
C
C*****************************************************************
C.....NODE GENERATION
C*****************************************************************
C
```

<u>Listing of the Program FEMESH</u> (continued)

```
      NFN=N
      NN=N-1
      N=NX*NY*NZ
      WRITE (6,150) NOPT
      WRITE (6,197) FORM,N,NX,NY,NZ
      WRITE(6,190)(MAT(I),I=1,NYM1)
      WRITE (6,193) (XP(I),I=1,NX)
      WRITE (6,194) (YP(I),I=1,NY)
      WRITE (6,195) (ZP(I),I=1,NZ)
      WRITE (6,198)
      DO 30 K=1,NZ
      DO 30 I=1,NX
      DO 30 J=1,NY
      NN=NN+1
      X=XP(I)
      Y=YP(J)
      Z=ZP(K)
      IF (FORM .LT. 0) GO TO 20
C
C.....SAP4 GENERATION
C
      CALL BCOND(IBC,IFLAG,X,Y,Z,NN,XP,YP,ZP,NX,NY,NZ)
      WRITE (6,200) NN,(IBC(II),II=1,6),X,Y,Z
      IF (IPUNCH .NE. 1) GO TO 25
      WRITE (IP,200) NN,(IBC(II),II=1,6),X,Y,Z
      GO TO 25
C

C.....ANSYSR3 GENERATION
C
   20 WRITE (6,300) NN,X,Y,Z
      IF (IPUNCH .NE. 1) GO TO 25
      WRITE (IP,300) NN,X,Y,Z
   25 CONTINUE
   30 CONTINUE
C
C*********************************************************************************
C.....ELEMENT GENERATION
C*********************************************************************************
C
      IF (NOPT .LT. 0) GO TO 80
      WRITE (6,199)
      EN=0
```

Listing of the Program FEMESH (continued)

```
      NXM1=NX-1
      NYM1=NY-1
      NZM1=NZ-1
      IF (NZ .EQ. 1) NZM1=NZ
      DO 50 K=1,NZM1
      DO 50 J=1,NXM1
      DO 50 I=1,NYM1
      EN=EN+1
      ENN(1)=I+(J-1)*NY+(K-1)*N/NZ+NEN-1
      ENN(2)= N/NZ+ENN(1)
      ENN(3)=ENN(2)+NY
      ENN(4)=NY+ENN(1)
      ENN(5)=ENN(1)+1
      ENN(6)=ENN(2)+1
      ENN(7)=ENN(3)+1
      ENN(8)=ENN(4)+1
      IF (FORM .LT. 0) GO TO 35
C
C.....SAP4 GENERATION
C
      CALL MISC(NA,D1,D2,D3)
      WRITE (6,202) EN,(ENN(II),II=1,8),(NA(JJ),JJ=1,6)
      IF (NPUNCH .NE. 1) GO TO 40
      WRITE (IP,202) EN,(ENN(II),II=1,8),(NA(JJ),JJ=1,6)
      GO TO 40
C
C.....ANSYSR3 GENERATION
C
   35 CALL MISCA(NA,EN)
      IF (NZ .EQ. 1) GO TO 65
      WRITE (6,302) (ENN(II),II=1,8),(NA(JJ),JJ=1,3)
      IF (NPUNCH .NE. 1) GO TO 40
      WRITE (IP,302) (ENN(II),II=1,8),(NA(JJ),JJ=1,3)
      GO TO 40
   65 WRITE (6,303) ENN(1),ENN(4),ENN(8),ENN(5),(NA(JJ),JJ=1,3)
      IF (NPUNCH .NE. 1) GO TO 40
      WRITE (IP,303) ENN(1),ENN(4),ENN(8),ENN(5),(NA(JJ),JJ=1,3)
   40 CONTINUE
   50 CONTINUE
   80 CONTINUE
C
C
```

Listing of the Program FEMESH (continued)

```
C.....FORMATS
C
   90 FORMAT (I5)
  100 FORMAT (16I5)
  101 FORMAT (8F10.0)
  102 FORMAT (20A4)
  105 FORMAT(10I5)
  196 FORMAT (1H1,1X,20A4,////)
  150 FORMAT (1X,15HNOPT                ,I5,1X,13H LT  0 NODES ONLY,/ )
  197 FORMAT((1X,15HFORMAT              ,I5,1X,11H GE  0 SAP4,/
     1        22X,                                12H LT  0 ANSYS,/
     2        1X,15HNUMBER OF NODES,I5,/
     3        1X,15HNX                  ,I5,/
     4        1X,15HNY                  ,I5,/
     5        1X,15HNZ                  ,I5,/      )
  190 FORMAT(//,1X,' MATERIAL NUMBER OF LAYERS (BOTTOM TO TOP)',/,16I5,/
     1//)
  193 FORMAT((1X,15HX COORD             ,/,13(F10.3),//      )
  194 FORMAT((1X,15HY COORD             ,/,13(F10.3),//  )
  195 FORMAT((1X,15HZ COORD             ,/,13(F10.3),////  )
  198 FORMAT ( 1H1,1X,24HNODE GENERATION RESULTS ,/////)
  199 FORMAT (1H1,1X,27HELEMENT GENERATION RESULTS.,//////)
  200 FORMAT (I5,6I5,3F10.4)
  202 FORMAT(I5,8I5,2I5,5X,4I2)
  300 FORMAT (I6,6X,3F12.4)
  302 FORMAT (8I6,3I6)
  303 FORMAT (4I6,24X,3I6)
C
C
      STOP
      END
```

Listing of the Program FEMESH (continued)

```
      SUBROUTINE MISC(NA,D1,D2,D3)
C
C****************************************************************
C.....SAP-MISC-ELEMENT DATA
C****************************************************************
C
      DIMENSION NA(6)
      NA(1)=2
      NA(2)=1
      NA(3)=1
      NA(4)=1
      NA(5)=1
      NA(6)=1
      RETURN
C
      END
```

Listing of the Program FEMESH (continued)

```
      SUBROUTINE BCOND(BC,IFLAG,X,Y,Z,NN,XP,YP,ZP,NX,NY,NZ)
C
C*****************************************************************************
C.....SAP BOUNDARY CONDITIONS
C*****************************************************************************
C
      INTEGER BC
      DIMENSION BC(6),XP(50),YP(50),ZP(50)
      BC(1)=0
      BC(2)=0
      BC(3)=0
      BC(4)=1
      BC(5)=1
      BC(6)=1
      IF (X .EQ. XP(1) .OR. X .EQ. XP(NX)) GO TO 10
      IF (Y .EQ. YP(1)) GO TO 20
      IF (Z .EQ. ZP(1) .OR. Z .EQ. ZP(NZ)) GO TO 30
      RETURN
   10 BC(1)=1
      RETURN
   20 BC(2)=1
      BC(1)=1
      BC(3)=1
      RETURN
   30 BC(3)=1
      RETURN
C
      END
```

Listing of the Program FEMESH (continued)

```
      SUBROUTINE MISCAN(NA,EN)
C
C**********************************************************************
C.....ANSYSR3 MISC ELEMENT DATA
C**********************************************************************
C
      INTEGER EN
      DIMENSION NA(6)
      COMMON/EXTRA/MAT(50),NX,NY,NZ
      NA(1)=1
      NA(2)=1
      NYM1=NY-1
      IF(MOD(EN,NYM1).EQ.0)GO TO 10
      MATMOD=MOD(EN,NYM1)
      NA(1)=MAT(MATMOD)
      GO TO 11
   10 NA(1)=MAT(NYM1)
   11 RETURN
C
      END
```

3.3 MESH GENERATION EXAMPLE

To use FEMESH, consider the example shown in Figure 3.1. All that is necessary is to enter the number of nodes in the x, y, and z directions, and the coordinates of the nodal lines in the x, y, and z directions, respectively. An example of the input is show below on the FORTRAN coding form.

<u>An Example of a FORTRAN Coding Form for FEMESH</u>

IBM FORTRAN Coding Form GX28-7327-6 U/M 050** Printed in U.S.A.

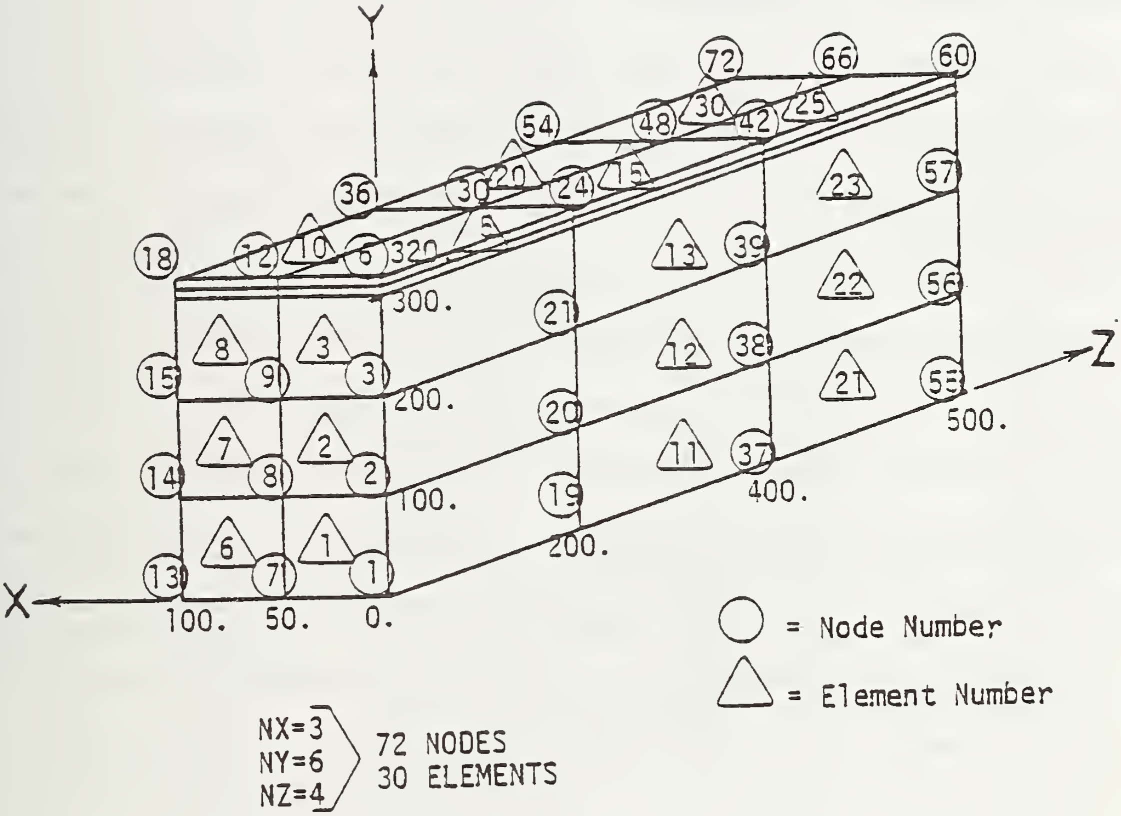
FEMESH

S.G. MILITSOPOULOS DATE 04/04/79

FORTRAN STATEMENT

```
 3-D  MESH GENERATION TEST (PRINTED ONLY)

   -1    1    3    6    4
    1    1    1    2    3
        0.       50.       100.
        0.      100.       200.       300.       310.       320.
        0.      200.       400.       600.
```



The uncircled numbers indicate the dimensions of the mesh in inches.

Figure 3.1 Example of Mesh Generation

The output of the example problem using FEMESH is as follows:

Output of Example Problem Using FEMESH

```
     3-D  MESH  GENERATION  TEST  (PRINTED  ONLY)

NOPT                        0  LT   0 NODES ONLY

FORMAT                     -1  GE   0 SAP4
                               LT   0 ANSYS
NUMBER OF NODES            72
NX                          3
NY                          6
NZ                          4

  MATERIAL NUMBER OF LAYERS (BOTTOM TO TOP)
    1     1     1     2     3
X COORD
      .000     50.000    100.000
Y COORD
      .000    100.000    200.000    300.000    310.000    320.000
Z COORD
      .000    200.000    400.000    600.000
```

Output of Example Problem Using FEMESH

NODE GENERATION RESULTS

1	.0000	.0000	.0000
2	.0000	100.0000	.0000
3	.0000	200.0000	.0000
4	.0000	300.0000	.0000
5	.0000	310.0000	.0000
6	.0000	320.0000	.0000
7	50.0000	.0000	.0000
8	50.0000	100.0000	.0000
9	50.0000	200.0000	.0000
10	50.0000	300.0000	.0000
11	50.0000	310.0000	.0000
12	50.0000	320.0000	.0000
13	100.0000	.0000	.0000
14	100.0000	100.0000	.0000
15	100.0000	200.0000	.0000
16	100.0000	300.0000	.0000
17	100.0000	310.0000	.0000
18	100.0000	320.0000	.0000
19	.0000	.0000	200.0000
20	.0000	100.0000	200.0000
21	.0000	200.0000	200.0000
22	.0000	300.0000	200.0000
23	.0000	310.0000	200.0000
24	.0000	320.0000	200.0000
25	50.0000	.0000	200.0000
26	50.0000	100.0000	200.0000
27	50.0000	200.0000	200.0000
28	50.0000	300.0000	200.0000
29	50.0000	310.0000	200.0000
30	50.0000	320.0000	200.0000
31	100.0000	.0000	200.0000
32	100.0000	100.0000	200.0000
33	100.0000	200.0000	200.0000

Output of Example Problem Using FEMESH

34	100.0000	300.0000	200.0000
35	100.0000	310.0000	200.0000
36	100.0000	320.0000	200.0000
37	.0000	.0000	400.0000
38	.0000	100.0000	400.0000
39	.0000	200.0000	400.0000
40	.0000	300.0000	400.0000
41	.0000	310.0000	400.0000
42	.0000	320.0000	400.0000
43	50.0000	.0000	400.0000
44	50.0000	100.0000	400.0000
45	50.0000	200.0000	400.0000
46	50.0000	300.0000	400.0000
47	50.0000	310.0000	400.0000
48	50.0000	320.0000	400.0000
49	100.0000	.0000	400.0000
50	100.0000	100.0000	400.0000
51	100.0000	200.0000	400.0000
52	100.0000	300.0000	400.0000
53	100.0000	310.0000	400.0000
54	100.0000	320.0000	400.0000
55	.0000	.0000	600.0000
56	.0000	100.0000	600.0000
57	.0000	200.0000	600.0000
58	.0000	300.0000	600.0000
59	.0000	310.0000	600.0000
60	.0000	320.0000	600.0000
61	50.0000	.0000	600.0000
62	50.0000	100.0000	600.0000
63	50.0000	200.0000	600.0000
64	50.0000	300.0000	600.0000
65	50.0000	310.0000	600.0000
66	50.0000	320.0000	600.0000
67	100.0000	.0000	600.0000
68	100.0000	100.0000	600.0000
69	100.0000	200.0000	600.0000
70	100.0000	300.0000	600.0000
71	100.0000	310.0000	600.0000
72	100.0000	320.0000	600.0000

Output of Example Problem Using FEMESH

ELEMENT GENERATION RESULTS

1	19	25	7	2	20	26	8	1	1	0
2	20	26	8	3	21	27	9	1	1	0
3	21	27	9	4	22	28	10	1	1	0
4	22	28	10	5	23	29	11	2	1	0
5	23	29	11	6	24	30	12	3	1	0
7	25	31	13	8	26	32	14	1	1	0
8	26	32	14	9	27	33	15	1	1	0
9	27	33	15	10	28	34	16	1	1	0
10	28	34	16	11	29	35	17	2	1	0
11	29	35	17	12	30	36	18	3	1	0
19	37	43	25	20	38	44	26	1	1	0
20	38	44	26	21	39	45	27	1	1	0
21	39	45	27	22	40	46	28	1	1	0
22	40	46	28	23	41	47	29	2	1	0
23	41	47	29	24	42	48	30	3	1	0
25	43	49	31	26	44	50	32	1	1	0
26	44	50	32	27	45	51	33	1	1	0
27	45	51	33	28	46	52	34	1	1	0
28	46	52	34	29	47	53	35	2	1	0
29	47	53	35	30	48	54	36	3	1	0
37	55	61	43	38	56	62	44	1	1	0
38	56	62	44	39	57	63	45	1	1	0
39	57	63	45	40	58	64	46	1	1	0
40	58	64	46	41	59	65	47	2	1	0
41	59	65	47	42	60	66	48	3	1	0
43	61	67	49	44	62	68	50	1	1	0
44	62	68	50	45	63	69	51	1	1	0
45	63	69	51	46	64	70	52	1	1	0
46	64	70	52	47	65	71	53	2	1	0
47	65	71	53	48	66	72	54	3	1	0

CHAPTER 4
OPERATING INSTRUCTIONS FOR ANSYS

4.1 WAVE FRONT SOLUTION AND LIMITATIONS

The ANSYS program uses the wave front direct solution method for the system of simultaneous linear equations which are developed by the matrix displacement method. The frontal direct solution gives results of high accuracy in a minimum of computer time.

There is no "band width" limitation in the problem definition. However, there is a "wave front" restriction. The "wave front" restriction depends on the amount of core storage available for a given problem. Up to 576 degrees of freedom on the wave front can be handled in a large core. An optional 1152 wave front is available on very large computers. However, it is recommended not to exceed the 571 wave front if the CYBER 176 (CDC 7600) computer is used. The wave front limitation tends to be restrictive only for the analysis of three-dimensional structures or in the use of ANSYS in small computers. There is no limit on the number of elements used in a problem, but there is a limit on the number of elements which consist the wave front. The number of equations which are active after an element has been processed during the solution procedure is called the wave front of that point.

For a banded solver, the band width is minimized by paying close attention to the ordering of the nodes. Alternatively, in the wave front procedure, the ordering of the element is crucial to minimize the size of the wave front. A degree of freedom becomes active when an element containing that degree is processed.

That degree of freedom remains active in core until all elements containing that degree of freedom have been processed. Therefore, the element cards must be arranged in such a way, so that the element for which each nodal point is mentioned first is as close in sequence to the element for which it is mentioned last.

The wave front must sweep through the model continuously from one end to the other in the direction which has the largest number of nodal points. The assembled matrix expands and contracts as nodal points make their first and last appearance in the element specifications. The optimum wave front for a simple line element model is a point; for a two-dimensional solid or plate element is a line of nodes; and for a three-dimensional solid element is an area of nodes.

An estimate of the wave front size can be made by multiplying the number of nodes in the wave front by the number of degrees of freedom per node. For example, consider the model shown in Fig. 3.1. The xz plane has the lesser number of nodes (3x4 = 12). Thus, the elements should be specified along the upper xz plane in the y direction. The new ordering of the elements is shown in Fig. 4.1.

If the elements described above have three degrees of freedom per node (say ux, uy, and uz), the maximum wave front size is approximately 12x3 = 36. Often, it is convenient to generate elements with FEMESH in an order that is not the best for an optimum wave front. If so, elements may be internally reordered by ANSYS using the F1 cards.

The F1 cards are called into the full ANSYS problem by inputting KORDER = 1 (column 78 of the C2 cards). A list of nodal points defining where the element reordering is to start is input on the first F1 card set. Additional lists may be defined to allow the user to guide the wave. The starting list usually consists of one node for a line element model, a line of nodes for an area element model, or a plane of nodes for a volume element model. There is a limit of 25 F1 cards or 1000 nodes, whichever comes first. All elements attached to the first node in the list are defined first, then all elements attached to the second node are defined next, etc., until all elements attached to all nodes input on the first F1 card set (but not on later F1 card sets) are defined.

This procedure is then repeated with the new set of nodal points brought in with the previously defined elements. If, during the course of reordering, an element would bring a node that is defined on a later F1 card set, that element is omitted until later. This feature allows the user to guide the wave front.

The element reordering, using the F1 cards of the mesh shown in Fig. 4.1 from the mesh shown in Fig. 3.1 (generated by FEMESH), is presented in Table 4.2. It is recommended not to use the F1 cards if interface elements (connecting the slab to the subgrade) are used. It is rather easier to reorder the elements by hand.

4.2 DATA INPUT INSTRUCTIONS

Abbreviated ANSYS input instructions and the proper formats are included in this section. Specific quantities to be used for some of the variables are given in the Element Library (Chapter 5) for the various element types which to be used. Standard FORTRAN conventions are used for the input quantities. Variables with first letters from I to N are integers and must be right justified (ending in the rightmost column) in the specified field. No decimal point should be included. Variables with other first letters are floating point numbers and may be placed anywhere in the field. Floating point numbers should have the decimal point input. The exponent, if any, must be right justified in the specified field.

No data should be punched on the cards in other than the specified fields. A blank input is treated as a zero or as a default option where indicated. Data cards must be in the order defined, and no additional cards (except for comment cards) are allowed. Comment cards may be inserted freely in the data deck. A comment card is identified by the characters C*** in columns 1 through 4. The remainder of the card is used for any comment that the user wants to have printed out along with the data input listing. All alphabetic labels (UX, FY, EX, END, etc.) must be left justified in their four space fields. Card sets requiring sentinel cards for termination are identified in the tables. A card having only a -1 in columns 5 and 6 may be used for any sentinel card. All geometric input angles are in degrees and output rotations are in radians. Right hand coordinate systems are used throughout except where specifically noted.

4.2.1 ANSYS Input Data for Static Analysis (ANSYS/Rev. 2)

CARD	COLUMN(S)	VARIABLE	MEANING
A	TITLE CARD		
	1-80	IHEDD	Title for output. If columns 77-79 are left blank and a comma punched in column 80, the title may be continued on the following card. No limit.
B	ACCOUNTING CARD		
	1-16	NAME	(Optional) User Identification Name.
	18	NONOTE	0 - Print notes (new features, modifications, announcements, etc.) at end of solution. 1 - Suppress printout of notes (continued use not recommended). (Not Available)
	25-32	IACCNT	(System Option) Account Number
	37-42	IEQRQD	(Optional) Maximum number of equations in wave front (to check for adequate core storage).
	75-80	ICORE	(System Option) Core size parameter.
	Note - If no values are input, a blank card is still required.		
C1	ANALYSIS OPTIONS		
	1-4	NSTEPS	Number of load steps (one set of L through Q cards per load step) (-NSTEPS for an input data check run).
	7	K20	0 - Static analysis
	11-12	KTB	0 - No element real constant table defined 1 - Define up to 8 element real constants per table entry (Card D2). N - Define up to N (for N greater than 8) element real constants per table entry (Card D2).
	16	K15	0 - No nodal force output. 1 - Calculate and print out nodal forces for each element and tabulate reaction forces at specified displacement constraints.

K20=0

CARD	COLUMN(S)	VARIABLE	MEANING
C1 (cont.)			
	16	K15	2 - Print out reaction force tabulation only
	18	K17	0 - Boundary conditions (Cards N, O, P, and Q) are linearly interpolated within a load step. The full boundary conditions (as input) are used in the last iteration of the load step. 1 - Boundary conditions are step changed at the first iteration to full values defined in the load step.
	22	K23	0 - No energy printout. 1 - Calculate and print out elastic strain energy for each element.
	74-75	KPROP	0 - Use polynomial material property equations. N - Use linear interpolation in all material tables, up to N points per table (24 max).
C2	ANALYSIS OPTIONS (CONTINUED)		
	1-12	TREF	Reference temperature for thermal expansions.
	13-24	TUNIF	Uniform temperature (used if no other temperatures are specified).
D	ELEMENT TYPES - One card for each element type. End card set with an I=0 card.		
	2-3	I	Element type number (arbitrary, between 1 and 20).
	5-6	J	Stiffness subroutine for this element. (A 0 will cause this element type to be ignored).
	7	KEYSUB(1B)	Parameter KEYSUB(1B) for this subroutine.
	8	KEYSUB(1A)	Parameter KEYSUB(1A) for this subroutine.
	9	KEYSUB(1)	Parameter KEYSUB(1) for this subroutine.
	10	KEYSUB(2B)	Parameter KEYSUB(2B) for this subroutine.
	11	KEYSUB(2A)	Parameter KEYSUB(2A) for this subroutine.
	12	KEYSUB(2)	Parameter KEYSUB(2) for this subroutine.
	14-15	KC	If J=0, enter for KC the stiffness subroutine number of the element type being ignored.
	18	INOTPR	If 1, suppress all stress and force printout for this element type.

(Return to next D card)

K20=0

CARD	COLUMN(S)	VARIABLE	MEANING
D2	ELEMENT REAL CONSTANT TABLE - (Include this card set only if KTB is greater than 0 on Card C1. The D2 cards may be repeated to form a table. End table with a blank (or 0.0 in the first field) card).		
	1-10, 11-20, 21-30, ----, 71-80	RC(1) RC(2) RC(3) ---- RC(8)	Element real constants (as given for element stiffness subroutine. Input constants in the same order as given. Several cards may be required for each table entry. Additional constants on cards are not used).
	If a +00000 is punched in columns 1-6 and the rest of the card is left blank, suppress the element constant table printout. If -99999, cancel the suppression.		
	If a +99999 is punched in columns 1-6 and a real number (D.) is input for RC(2), this card represents D blank table entries.		
			(Return to next D2 card)
E	ELEMENT DEFINITION CARDS - one card set (E1, E2) for each element - end with an I=-1 card.		
E1	ELEMENT DESCRIPTION		
	1-6	I	Number assigned to Node I on element (first node). If 99999, suppress element printout. If -99999, cancel the suppression.
	7-12	J	Number assigned to Node J on element (second node, if any).
	13-18,19-24, 25-30,31-36, 37-42,43-48	K, L, M, N, O, P	Other node numbers, if required.
	49-54	MAT	Material number of this element (1 if blank).
	55-60	ITYPE	Element type number for this element (1 if blank). (Refers to element types defined on D cards).
	61-66	ITABLE	0 - Element real constants, if any, are included on the next card (Card E2). K - Element real constants are included at entry number K of the D2 card set.

K20=0

CARD	COLUMN(S)	VARIABLE	MEANING
E1 (cont.)			
	(The following three parameters are required only for ·or second level element generation).		
	67-72	INUM	If positive (first level generation), INUM is the total number of element sets generated (including the specified set). The elements input on this and the next NEL-1 E1 cards form the specified set.
			If negative (second level generation), -INUM is the total number of element groups generated (including the NEL elements in the defined group). The defined group may include separately specified and/or first level generated elements. Columns 1-66 should be left blank.
	73-75	NINC	Number by which to increment each element node number to generate successive element sets or groups. (Assumed 1 if left blank).
	76-78	NEL	Number of elements in a specified set or a defined group to be repeated (assumed 1 if blank).
			Element limit per set = 960/N (where N=8 or KTB, if KTB (Card C1) is greater than 8). No element limit per group.
	79-80	KNEXT	If positive, the tape unit for additional element input data (defaults to the current input file).
			If -1, all of the following elements have INUM added to each node number.
E2	ELEMENT REAL CONSTANTS - (Include this card set only if the element has required real constants and if ITABLE (on the preceding E1 card) is zero or blank).		
	1-12, 13-24, 25-36, 37-48, 49-60, 61-72	RC(1) RC(2) RC(3) RC(4) RC(5) RC(6)	Element real constants (as given for element stiffness subroutine. Input constants in the same order as given. Several cards may be required for each table entry. Additional constants on cards are not used).
		(Return to next E1 card)	

K20=0

CARD	COLUMN(S)	VARIABLE	MEANING
F	NODE POINT LOCATIONS - One card for each node specified - end with an I=-1 card.		
	1-6	I	If positive, I is the node number being defined (not all numbers need to be used). If zero (or blank), this card is used to define a local coordinate system. If negative, this card is used to define second level nodal point generation. -I is the node number increment between successive nodal point groups. If 99999, suppress nodal point printout. If -99999, cancel the suppression.
	7-8	KCS	*** If I is not zero *** 0 - Nodes input (or generated in global cartesian coordinates. 1 - Nodes input (or generated) in global cylindrical coordinates. 2 - Nodes input (or generated) in global spherical coordinates. N - Nodes input (or generated) in local coordinate system N (N greater than 2). *** If I is zero *** 0 - A local cartesian coordinate system is being defined. 1 - A local cylindrical coordinate system is being defined. 2 - A local spherical coordinate system is being defined.
	9-10	KFILL	If I is positive *** 0 - No first level nodal point generation. N - Fill in nodes between the previously specified node and this one, incrementing node numbers by N and linearly interpolating the coordinates. (First level nodal point generation). (N must be positive).

K20=0

CARD	COLUMN(S)	VARIABLE	MEANING
F (cont.)	9-10	KFILL (cont.)	*** If I is zero *** N - The local coordinate system being defined is identified as coordinate system number N (N greater than 2). *** If I is negative *** N - The number of nodal points in the group to be repeated (defined on the following F cards). (Second level nodal point generation).
	11-12	KNEXT	*** If I is positive *** N - The tape unit number for additional nodal point input data (defaults to the current input file). *** If I is zero, KNEXT is not used *** *** If I is negative *** N - The total number of nodal point groups generated (including the defined group). (Second level nodal point generation). *** Special Combinations *** If KNEXT=-1, all of the following node numbers have I (positive or negative) added to them. All other parameters on the card should be left blank.

For defining nodal points use the appropriate node description column below. The THXY, .., THRP inputs are for nodal coordinate rotation. All angles are input in degrees. Use 3-D input if a 3-D element is included in the D card set.
For local coordinate system definition use Column 3 for origin translation and coordinate system rotation.

For second level generation, inputs are incremental values. Increments and nodal points must be specified in the same coordinate system (KCS).

	2-Dimensional		3-Dimensional		
	Rectangular	Polar	Cartesian	Cylindrical	Spherical
13-24	X	R	X	R	R
25-36	Y	THETA	Y	THETA	THETA
37-48	THXY	THRT	Z	Z	PHI
49-60			THXY	THRT	THRT
61-72			THYZ	THTZ	THTP
73-80			THXZ	THRZ	THRP

(Return to next F card)

K20=0

CARD	COLUMN(S)	VARIABLE	MEANING
H	MATERIAL PROPERTY DEFINITIONS - the H card set (H1, H2) may be repeated. End with a LABEL=END card.		
H1	MATERIAL PROPERTY EQUATIONS		
	1-4	LABEL	LABEL identifying the property EX EY EZ ALPX ALPY ALPZ NUXY NUYZ NUXZ DENS MU GXY GYZ GXZ COPR NOPR END (Note - All labels are left justifies) (Only properties required by element material descriptions need be input. In addition, for isotropic materials, only the X (or XY) property label need be input).
	5-8	MAT	Material number (assumed 1, if left blank).
	12	KEY	0 - Polynomial coefficients are input on this card. 1 - A curve must be fit to the set of temperature vs. property data points listed on the following H2 cards. 2 - Fit curve as described for KEY=1 and print out the fitting equation coefficients. 3 - Use linear interpolation in all material property tables (input table on H2 cards). (Note, if KEY=3 for any material, it must be 3 for all materials).
	13-24	C0	Constant term in the property polynomial equation.
	25-36	C1	Coefficient of linear term in equation.
	37-48	C2	Coefficient of quadratic term.
	49-60	C3	Coefficient of cubic term.
	61-72	C4	Coefficient of quartic term.
H2	MATERIAL PROPERTY TABLE - (Included only if KEY is greater than zero on previous H1 card)		
	First card -		
	1-12	POINTS	Number of temperature vs. property points in table. If KEY=1 or 2, at least 6 property points are required. If KEY=3, the number of points must not exceed the KPROP value input on Card C1.

K20=0

CARD	COLUMN(S)	VARIABLE	MEANING
H2 (cont.)			
	13-24	TSTART	Temperature corresponding to first property value input (required only if DELTAT is greater than zero).
	25-36	DELTAT	Constant value by which temperatures are incremented. Temperatures correspond to property values input on the next card(s).

Following cards - If DELTAT=0.0 (or blank), three temperature-property pairs may be input per card. Temperatures must be input in ascending order. If DELTAT is greater than zero, six properties may be input per card. Properties correspond to temperatures generated on first H2 card.

(Continue table on as many cards as required, Format (6E12.2))

(Return to next H1 card after table is complete).

L-Q The following load cards (L-Q) are repeated NSTEPS (Card C1) times unless the repeating sets are terminated with a KDIS=99 card before the last expected (NSTEPS) set.

L LOAD STEP DEFINITION

	1-3	KDIS	1 - Define new values for displacement, force, and pressure boundary conditions. Formulate new stiffness matrix. Zero all nonlinear terms and previous boundary conditions. 0 - Use the previous displacement, force and pressure boundary conditions (do not include N, O, or P card sets). Reformulate stiffness matrix. Continue nonlinear analysis. -1 - Change some of the previously defined displacement, force, and pressure boundary conditions (include changed values and N, O, and P card set terminators). Also use unchanged previous boundary conditions. Reformulate stiffness matrix. Continue nonlinear analysis. 2 - Same as KDIS=1 except use previously formulated stiffness matrix (specified displacement constraints (on N cards) must be repeated (and all zero)).

K20=0

CARD	COLUMN(S)	VARIABLE	MEANING
L (cont.)	1-3	KDIS (cont.)	-2 - Same as KDIS=-1, except use previously formulated stiffness matrix (specified displacement constraints must remain zero). 99 - Terminate the load card sets before the last expected set. An R or S card must follow.
	4-6	KTEMP	0 - Set all temperatures to TUNIF (Card C2). 1 - Read in element temperatures on the Q cards for all elements. 2 - Read in nodal point temperatures on the Q cards. 3 - Use temperatures from previous load step. -N - Use the temperatures calculated in the Nth cumulative iteration (file TAPE4) of a previous ANSYS heat transfer solution.
	7-9	NITTER	The number of sub-step (or iterative) calculations to be done this load step (defaults to 1). Note, boundary conditions are linearly interpolated if K17=0. If NITTER is negative, use covergence options (step boundary condition change imposed).
	10-12	NPRINT	Frequency of printout of stress, force, and displacement results - only every NPRINT iteration is printed out, beginning with iteration NPRINT. <u>If zero or blank, suppress all printout for this load step</u>. If negative, suppress boundary condition input printout only. For a negative value of NITTER, if NPRINT= NITTER , print the converged (or last) iteration. If NPRINT > NITTER , suppress all solution printout.

M ADDITIONAL LOAD, PLOT, AND PRINT DEFINITION CARD

The following four parameters may be used if more than three space fields are needed for the corresponding parameters on the L card.

COLUMN(S)	VARIABLE	MEANING
1-6	KDIS	If non-zero, use instead of the value on Card L.
7-12	KTEMP	If non-zero, use instead of the value on Card L.

K20=0

CARD	COLUMN(S)	VARIABLE	MEANING
M (cont.)			
	13-18	NITTER	If non-zero, use instead of the value on Card L.
	19-24	NPRINT	If non-zero, use instead of the value on Card L.
N	DISPLACEMENT DEFINITION CARDS - The N cards may be repeated. End with a LABEL=END card.		
	1-6	I	Node number at which displacement is specified. If 99999, suppress displacement printout. If -99999, cancel the suppression. If -2, add I2 to all the following nodes.
	7	IKEY	If -, delete this displacement specification.
	8-11	LABEL	Direction of displacement. (In nodal coordinate system) UX UY UZ ROTX ROTY ROTZ PRES END
	13-24	DISP	Value of displacement at this time (Radians for totations).
	37-42 43-48	I2 I5	If I2 is greater than I (for I positive), all nodes from I through I2 in stpes of I5 have this specified displacement (I5 is assumed to be 1, if left blank)
	51-54, 57-60,63-66, 69-72,75-78	LABELS(5)	Additional direction labels for which this displacement value applies at this node.
		(Return to next N card)	
O	FORCE DEFINITION CARDS - The O cards may be repeated. End with a LABEL=END card.		
	1-6	I	Node at which force acts If 99999, suppress force printout If -99999, cancel the suppression If -2, add I2 to all the following nodes.
	8-11	LABEL	Direction of force. (In nodal coordinate system) FX FY FZ MX MY MZ FLOW END K20=0

CARD	COLUMN(S)	VARIABLE	MEANING
O (cont.)			
	13-24	FORCE	Value of the force at this time.
	37-42	I2	If I2 is greater than I (for I positive), all nodes from I thru I2 in steps of I5 have this specified force (I5 assumed to be 1 if left blank).

(Return to next O card)

P PRESSURE DEFINITION CARDS - The P cards may be repeated. End with a blank (or I=0) card.

Pressures act in the element coordinate system. See Table 4.J.1 for pressures available for element type J.

COLUMN(S)	VARIABLE	MEANING
1-6	I	Element upon which pressure acts If 99999, suppress pressure printout. If -99999, cancel the suppression. If -2, add I2 to all the following elements.
7-12	IFACE	Face of element on which pressure acts. (If a super-element, IFACE is the load vector number).
13-24	PRESS	Value of the pressure at this time. (If a super-element, PRESS is the scale factor for load vector IFACE).
37-42	I2	If I2 is greater than I, all elements from I through I2 in stpes of I5 have this pressure on this face (I5 is assumed 1 if left blank).

(Return to next P card)

Q TEMPERATURE DEFINITION CARDS - (Include this card set only if KTEMP=1 or 2 on Card L).

Element temperature format (used if KTEMP is 1). One specification is required for each element, in the same order that the elements are specified. If KTEMP=2, use the node temperature format.

COLUMN(S)	VARIABLE	MEANING
1-8	T1	First temperature for this element.

K20=0

CARD	COLUMN(S)	VARIABLE	MEANING
Q (cont.)	9-16,... ...,57-64	T2,... ...,T8	Second temperature, etc. (Note - Fluences are also input where applicable).
	65-72	INUM	If 0 or 1, one element has this set of temperatures. If N, the next N elements (counting this element) have these temperatures.
	73-74	KNEXT	If positive, subsequent temperature input is to be from tape KNEXT (defaults to the current input file).
	76	KTCONT	0 - All temperatures and fluences to be specified are contained on this card. 1 - Additional temperatures and fluences continued on next card.

Note - If KTCONT=1, T9 through T16 should be input on the next (second) card. The continuation card format is the same as the first card except that INUM and KNEXT are not used. Values not input are assumed to be zero.

If a +99999 is punched in columns 1-6, suppress the element temperature printout. If -99999, cancel the suppression.

(If all element temperatures have not been specified, return to next Q card).

Nodal point temperature format (used if KTEMP is 2). Nodal temperature specification cards may be repeated. Nodal temperatures not specified are set equal to TUNIF (Card C2). End nodal temperature set with an I=-1 card.

COLUMN(S)	VARIABLE	MEANING
1-6	I	Node number at which temperature is specified (if -1, end of nodal temperature input). If 99999, suppress nodal temperature printout. If -99999, cancel the suppression. If -2, add I2 to all the following nodes.
13-24	TEMP	Specified nodal temperature.
25-36	FLUENCE	Specified nodal fluence.
37-42 43-48	I2 I5	If I2 is greater than I (for I positive), all nodes from I through I2, in steps of I5, have this temperature (I5 is assumed to be 1 if left blank).

K20=0

CARD	COLUMN(S)	VARIABLE	MEANING
Q (cont.)		(Return to next Q card)	

(Return to next L card if another load step is to be defined).

S	END OF DATA DECK CARD		
	1-6	FINISH	The word FINISH is punched in Columns 1-6 of the last card of a problem data deck. Another problem data deck (beginning with Card A) may follow.

****

K20=0

Oftentimes for real life problems, the engineer will have to use the 3rd Revision of ANSYS (Extended Core Version). To go from the 2nd Revision to the 3rd one, set NSTEPS = 0 (solution problem) or NSTEPS = -1 (model check problem) on card C1 and finish with L-Q card sets with a KDIS=END card.

4.2.2 ANSYS Input Data for Heat Transfer Analysis (ANSYS/Rev. 2)

CARD	COLUMN(S)	VARIABLE	MEANING
A	TITLE CARD - See section 4.2.1 for data input instructions.		
B	ACCOUNTING CARD - See section 4.2.1 for data input instructions.		
C1	ANALYSIS OPTIONS		
	1-4	NSTEPS	Number of load steps (one set of L through Q cards per load step). (-NSTEPS for an input data check run).
	6-7	K20	-1 - Heat Transfer analysis.
	11-12	KTB	0 - No element real constant table defined. 1 - Define up to 8 element real constants per table entry (Card D2). N - Define up to N (for N greater than 8) element real constants per table entry (Card D2).
	16	K15	0 - No nodal heat flow rate printout. 1 - Calculate and print out nodal heat flow rate for each element and tabulate heat flow rates at specified temperature constraints. 2 - Print out heat flow rate tabulation only.
	18	K17	0 - Boundary conditions (Cards N, O, P, and Q) are linearly interpolated within a load step. The full boundary conditions (as input) are used in the last iteration of the load step. 1 - Boundary conditions are step changed at the first iteration to the full values defined in the load step.
	32	KAY(2)	0 - First order integration for transient solutions.

K20=-1

CARD	COLUMN(S)	VARIABLE	MEANING
C1 (cont.)	32	KAY(2) (cont.)	1 - Second order integration for transient solutions (recommended) (required for convergence or optimization procedures).
	74-75	KPROP	0 - Use polynomial material property equations. N - Use linear interpolation in all material tables, up to N point per table (24 max.).
	77	K18	0 - Nodal coordinate directions rotated for nodes input in global cylindrical and global spherical coordinates (nodal x-axis is along input radius unless otherwise specified on F card.
C2	ANALYSIS OPTIONS (CONTINUED)		
	13-24	TUNIF	Used only if KDIS=1. If so, all nodal temperatures, the temperature boundary conditions (Card N) and the bulk temperatures (Card P) at the beginning of the load step are set to TUNIF. Also temperature dependent material properties are evaluated at TUNIF for the first iteration.
	49-54	NUMEL	Number of elements (required only for restart).
	55-60	MAXNP	Maximum node number (required only for restart).
	61-64	KRSTRT	The last load step already done. (restart key).
	65-68	TOFFST	Degrees between absolute 0 and 0 of temperature system used (required for radiation).
	69-76	TRSTRT	Time at end of run to be continued (required only for restart).
D	ELEMENT TYPES - See Section 4.2.1 for data input instructions.		

Note, the INOTPR parameter is used to suppress all heat flow printout for this element type.

K20=-1

CARD	COLUMN(S)	VARIABLE	MEANING
D2	ELEMENT REAL CONSTANT TABLE - (Include this card set only if KTB is greater than 0 on Card C1. See section 4.2.1 for data input instructions).		
E	ELEMENT DEFINITION CARDS - See Section 4.2.1 for data input instructions.		
F	NODE POINT LOCATIONS - See Section 4.2.1 for data input instructions.		
H	MATERIAL PROPERTY DEFINITIONS - See Section 4.2.1 for data input instructions.		
	Note, the list of structural property labels (Cols. 1-4) should be replaced with the following thermal property identification list. KXX KYY KZZ DENS C HF OHMS VISC EMIS NOPR GOPR END		
L-Q	The following load cards (L-Q) are repeated NSTEPS (Card C1) times unless the repeating sets are terminated with a KDIS=99 card before the last expected (NSTEPS) set.		
L	LOAD STEP DEFINITION		
	1-3	KDIS	1 - Define new values for temperature, heat flow, and convection boundary conditions. Formulate new conductivity and specific heat matrices. Zero all transient terms and previous boundary conditions. 0 - Use the previous temperature, heat flow, and convection boundary conditions (do not include N, O, or P card sets). Re-formulate matrices. Continue transient analysis. -1 - Change some of the previously defined temperature, heat flow, and convection boundary conditions (include changed values and N, O, and P card set terminators). Also use unchanged previous boundary conditions. Re-formulate matrices. Continue transient analysis.

K20=-1

CARD	COLUMN(S)	VARIABLE	MEANING
L (cont.)	1-3	KDIS (cont.)	99 - Terminate the load card sets before the last expected set. An R or S card must follow.
	4-6	KTEMP	0 - Set all internal heat generation rates to 0.0. 1 - Read in element internal heat generation rates on the Q cards for all elements. 3 - Use heat generation rates from previous load step.
	7-9	NITTER	The number of sub-step (or iterative) calculations to be done this load step (defaults to 1). Note, if K17=0, boundary conditions are linearly interpolated. If NITTER is negative, use steady-state convergence (step boundary condition change required) or transient optimization procedure.
	10-12	NPRINT	Frequency of printout of heat flows and temperature results - only every NPRINT iteration is printed out, beginning with iteration NPRINT. <u>If zero or blank, suppress all printout for this load step</u>. If negative, suppress boundary condition input printout only. For a negative value of NITTER, if NPRINT= NITTER , print the converged (or last) iteration. If NPRINT > NITTER , suppress all solution printout.
	13-24	TIME	Time characterizing the end of this load step (If TIME is 0.0, blank, or less than the time of the previous load step, a steady-state solution is done).

M ADDITIONAL LOAD, PLOT, AND PRINT DEFINITION CARD

The following four parameters may be used if more than three space fields are needed for the corresponding parameters on the L card.

COLUMN(S)	VARIABLE	MEANING
1-6	KDIS	If non-zero, use instead of the value on Card L.
7-12	KTEMP	If non-zero, use instead of the value on Card L.
13-18	NITTER	If non-zero, use instead of the value on Card L.
19-24	NPRINT	If non-zero, use instead of the value on Card L.

K20=-1

CARD	COLUMN(S)	VARIABLE	MEANING
N	TEMPERATURE DEFINITION CARDS - The N cards may be repeated. End with a LABEL=END card.		
	1-6	I	Node number at which temperature is specified. If 99999, suppress temperature printout. If -99999, cancel the suppression. If -2, add I2 to all the following nodes.
	7	IKEY	If -, delete this temperature specification.
	8-11	LABEL	Input one of the following words (left justified). TEMP PRES VOLT END
	13-24	TEMPER	Value of temperature (etc.) at this time.
	37-42 43-48	I2 I5	If I2 is greater than I (for I positive), all nodes from I through I2 in steps of I5 have this specified temperature (I5 is assumed to be 1, if left blank).
		(Return to next N card)	
P	CONVECTION DEFINITION CARDS - The P cards may be repeated. End with a blank (or I=0) card.		
	1-6	I	Element upon which convection acts. If 99999, suppress convection printout. If -99999, cancel the suppression. If -2, add I2 to all the following elements.
	7-12	IFACE	Face ~~of element~~ on which convection acts. If a super-element, IFACE is the load vector number.
	13-24	HCOEF	Value of the film coefficient at this time. Note, if KDIS=1, the film coefficient at the beginning of this load step is also set to this value. If -N., use HF vs. TFILM equation input for material N on the H cards. If a super-element, HCOEF is the scale factor for load vector IFACE.
	25-36	TBULK	Bulk temperature of adjacent fluid at this time.

K20=-1

CARD	COLUMN(S)	VARIABLE	MEANING
P (cont.)		TBULK (cont.)	
	37-42 43-48	I2 I5	If I2 is greater than I (for I positive), all elements from I through I2 in steps of I5 have this convection on this face (I5 is assumed to be 1 if left blank).

(Return to next P card)

Q HEAT GENERATION RATE DEFINITION CARDS - (Include this card set only if KTEMP=1 on Card L. One specification is required for each element, in the same order that the elements are specified).

1-8	HTGEN or C1	Internal heat generation rate for this element.
9-16, --,57-64	C2,-- --,C8	Constants defining polynomial equation for variable heat generation rate (applicable to STIF71 elements).
65-72	INUM	If 0 or 1, one element has this rate. If N, the next N elements (counting this element) have this rate.
73-74	KNEXT	If positive, subsequent heat generation rate input is to be from tape KNEXT (defaults to the current input file).

If a +99999 is punched in Columns 1-6, suppress the internal heat generation rate printout. If -99999, cancel the suppression.

(If all element heat generation rates have not been specified return to the next Q card).

(Return to next L card if another load step is to be defined)

S END OF DATA DECK CARD

1-6	FINISH	The word FINISH is punched in Columns 1-6 of the last card of a problem data deck. Another problem data deck (beginning with Card A) may follow.

K20=-1

4.2.3 ANSYS Input Data for Thermal Stress Analysis

The thermal stress is equivalent to the static analysis except as follows:

1. Save file TAPE 4 from heat transfer analysis.
2. Assign file TAPE 4 to thermal stress analysis.
3. Set KTEMP=-N (read temperatures fo the Nth iteration of previous heat transfer solution from file TAPE 4).

Table 4.1 Element Reordering Instructions

FIELD	COLUMNS	VARIABLE	MEANING
CARD F1 ****	ELEMENT REORDERING INSTUCTIONS--Use as many cards as necessary to define a starting wave set. Use additional cards to define additional starting wave sets. End F1 card set with and N1= END card.		
1	1-6	N1	First node, second node, third
2	7-12	N2	node, etc., on starting wave.
3	13-18	N3	Continue on additional cards if necessary (Format 12I6).
12	67-72	N12	Blank fields are ignored. End starting wave list with a -1 node number.

(Return to next F1 card)

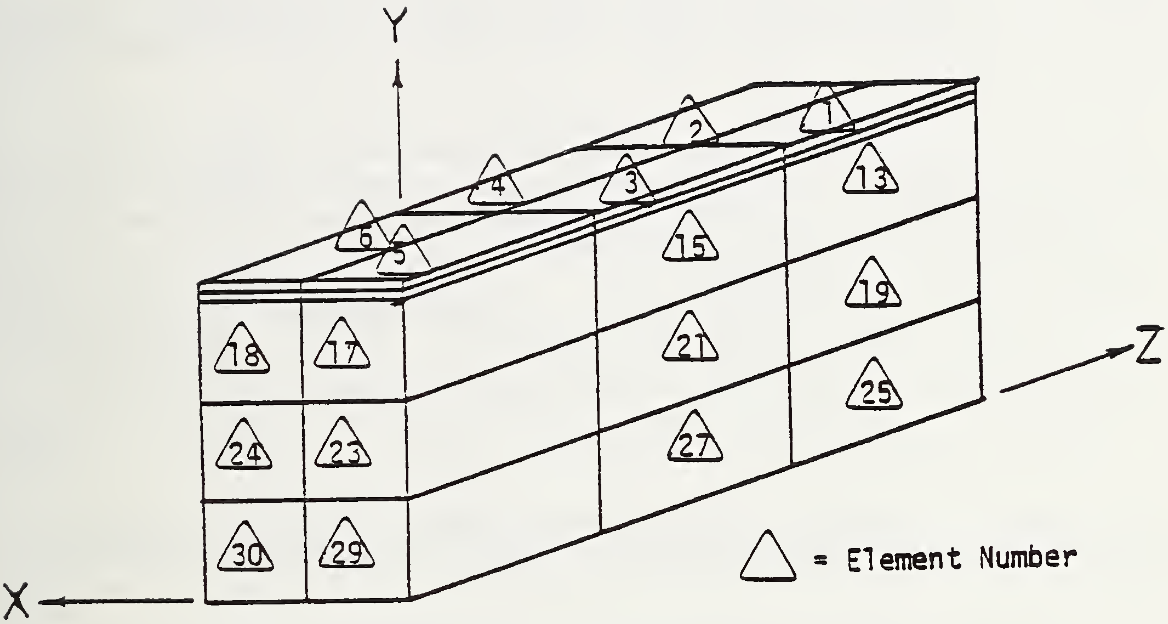
Table 4.2 Element Reordering for Sample Problem

IBM — FORTRAN Coding Form — GX28-7327 Printed in U.S.A.

PROGRAM: ANSYS
PROGRAMMER: S. G. MILITSOPOULOS — DATE: 05/15/79

60	66	72
42	48	54
24	30	36
6	12	18
-1		

END



NX=3
NY=6
NZ=4

NY>NZ>NX

NOTE THAT THE ORDERING OF THE NODES IS THE SAME AS IN FIGURE 3.1

WAVE FRONT=(NX)(NZ)(DEGREES OF FREEDOM PER NODE)

Figure 4.1 Example of Element Reordering to Minimize the Wave Front

CHAPTER 5
ELEMENT LIBRARY OF ANSYS

5.1 ELEMENT SELECTION

Table 5.1 is a summary of the available elements in ANSYS program. The above table lists the element identification number, the name, the number of dimensions, the number of degrees of freedom per node, the number of nodes, and some features.

ANSYS models are either two-dimensional or three-dimensional, depending upon the element types used. Two-dimensional models must be defined in the x-y plane and the nodes must be input using the two-dimensional format on the F cards. Three-dimensional models must be defined in the x-y-z plane and the nodes must be input using the three-dimensional formate on the F cards. The element input is included on the ANSYS program data input cards as shown in Table 5.2.

The degrees of freedom associated with the model should be sufficient to characterize the actual response. Including unnecessary degrees of freedom or selecting elements with unnecessary features increases the solution core size and running time.

The units of the element input and output parameters are described in Table 5.3 in terms of force (F), length (L), time (t), temperature (T), and heat (Q). Mass units can also be expressed as Ft^2/L.

5.2 ELEMENT LIBRARY FOR STATIC ANALYSIS

The three-dimensional isoparametric element (STIF45) and the three-dimensional interface element (STIF52) are recommended to use in a static analysis.

5.2.1 Three-dimensional Isoparametric Solid Element

The three-dimensional isoparametric solid element is a higher-order version of the three-dimensional elastic solid element (STIF5). The higher-order element gives a considerable improvement of accuracy over the constant strain element. The advantage of isoparametric elements over constant strain elements is that, for a given accuracy, the number of degrees of freedom necessary to describe the structure may be reduced. Accordingly, not only the data preparation time, but also the computer wave-front solution time is reduced.

The element has plasticity, but no creep or swelling capabilities. If all capabilities are needed, STIF49 should be used. The isoparametric solid element is defined by eight nodal points having three degrees of freedom at each node: translations in the nodal x, y , and z direction.

An option is available to print out the stresses and strains on particular element surfaces when the surfaces are free surfaces of the structure. Other options are available to print stresses at the integration points or at the nodes. A summary of the isoparametric solid element parameters is given in Table 5.4.

Input Data. The geometry, nodal point locations, face numbers, loading, and the coordinate system for this family of elements are shown in Fig. 5.1. The element is defined by eight nodal points and the material properties. The nodal points should be numbered in the order shown in Fig. 5.1. The number of nodes input on Card E1 defines the type of solid element used. The material may be orthotropic, with ten elastic constants required for its specification. The three additional shear modulus terms are optional and may be included for a more complete description of the material. If not included, the values are computed from the other input properties. There are no real constants required for this element.

The element loading can be either temperature gradients (specified by nodal temperatures) or pressures (on one or more faces), or a combination of both.

The data input for the isoparametric solid element is as follows: 1) only the eight node element with six pressure surfaces is available; 2) plasticity capability is included; 3) printout is available on a second surface for elastic solutions, as the numerical integration points, and at the nodal points; 4) the incompatible displacement modes may be suppressed with KEYSUB(1B), and 5) the number of number integration points may be selected for elastic solutions with KEYSUB(1A).

Output Data. The solution printout associated with the isoparametric solid element is summarized in Table 5.5. Figure 5.2 shows a schematic STIF45 element output.

Theory. The element formulation includes incompatible displacement modes. Either a 3x3x3 or a 2x2x2 lattice of integration points is available for use with the numerical (Gaussian) integration procedure.

Assumptions and Restrictions. Zero volume elements are not allowed. Elements may be numbered either as shown in Fig. 5.1 or may have the planes IJKL and MNOP interchanged. Also, the element may not be twisted such that the element has two separate volumes. This occurs most frequently when the elements are not numbered properly. The dihedral angle between adjacent element faces should be less than 180°.

All elements must have eight nodes. A "triangular" shaped element may be formed by defining duplicate K and L and duplicate O and P node numbers. The extra mode shapes are automatically deleted for "triangular" shaped elements so that a "constant strain" element results.

The first two lines of the element solution printout are valid for both isotropic and orthotropic materials. The principal strains (line 3) are not valid for orthotropic materials. The principal stresses and the maximum shear stresses, however, are valid for orthotropic materials.

Surface stress outputs are valid only for isotropic elastic materials for which this face is a free surface of the structure. Surface stresses should not be requested on the zero area face of "triangular" shaped elements.

The 2x2x2 lattice of integration points is automatically used with plasticity solutions (K13 > 0 on Card C1).

5.2.2 Three-dimensional Interface Element

The three-dimensional interface element represents two parallel surfaces in space which may maintain or break physical contact and may slide relative to each other. The element is capable of supporting only compression in the direction normal to the surfaces and shear (Coulomb friction) in the tangential directions. The element has three degrees of freedom at each node: translations in the nodal x, y, and z directions.

The element may be initially pre-loaded in the normal direction or it may be given a gap specification. A specified stiffness acts in the normal and tangential directions when the gap is closed and not sliding. Because of the nonlinearity of the element an iterative solution procedure is required. A summary of the three-dimensional interface element parameters is given in Table 5.6.

Input Data. The geometry, nodal point locations, and the coordinate system for the interface element are shown in Fig. 5.3. The element is defined by two nodal points, an interface stiffness, an initial gap (or interference, and an initial element status. The orientation of the interface plane (unlike STIF12) is defined by the nodal point locations. The plane is assumed to be perpendicular to the I-J line. The element coordinate system has its origin at node I and the x-axis is directed toward node J. The interface plane is parallel to the element y, z plane.

The stiffness, k, may be computed from EA/L_{eff} where the parameters are determined from the adjacent element. The effective length, L_{eff}, is arbitrary, but may be on the order of 1/10 of the adjacent element length. The stiffness may also be computed from the maximum expected force divided by the maximum allowable surface displacement. In most cases k is several orders of magnitude greater than the other stiffnesses in series with it so that its exact value is not critical. The initial gap (GAP) may be positive or negative. If negative, an initial interference of this amount exists. The initial element status (START) is used to define the "previous" condition of the interface to be used at the start of the first iteration. This input overrides the condition implied by the gap specification and is useful in anticipating the final interface configuration and thereby reducing the number of iterations required for convergence. This parameter is also useful for inputting the element status in a run which is to be continued, as determined from a previous ANSYS run.

The only material property required is the interface coefficient of friction, μ. A zero value should be used for frictionless surfaces. Temperatures (used if μ is temperature dependent) may be specified at the element nodes. For some problems, a loss of contact or a sliding at the interface isolates a portion of the structural model not having sufficient displacement constraints. The KEYSUB(1) option, therefore, may be used to maintain a small force across and along the interface, maintaining stability while causing only a negligible inaccuracy in the analysis. The KEYSUB(2) option may be used whenever friction may cause some gap elements to oscillate slightly between a sliding and a sticking status.

Output Data. The solution printout associated with the three-dimensional interface element is summarized in Table 5.7. The value USEP is the normal displacement (in the element x-direction) betw een the interface surfaces at the end of this iteration, that is, USEP = $(u_n)_J - (u_n)_I$ + GAP. This value is used in determining the normal force. Note, the normal force will not be an equilibrium value unless this iteration represents a converged solution.

The value USLIDE is the accumulated amount of surface sliding at the end of this iteration. Sliding may occur in both the element y and z coordinate directions. Note, sliding occurs in the iteration after the limiting tangential force is exceeded. KTYPE describes the status of the element at the end of this iteration for use in the next iteration. The surface may be in rigid contact (KTYPE=1), sliding contact (KTYPE=2), or free (KTYPE=3). If, for example, KTYPE=3 at the end of an iteration, an element stiffness of zero is used for the next iteration. The KTYPE values may be input for START if a new run is to continue from this iteration.

If no other effects are present and KEYSUB(2)=0, convergence occurs whenever the gap status remains unchanged. For a frictionless surface, the converged gap status is either KTYPE=2 or 3. Whenever KEYSUB(2) > 0, an element having sliding force oscillations within a defined tolerance range on μF_n, resulting in an oscillating gap status (KTYPE=1, 2, etc.), is accepted as converged. This tolerance range is usually within the uncertainty range of μ.

Theory. The displacement functions for the interface element can be separated into the normal and tangential directions since they are basically independent. In the normal (element x) direction, when the normal force (F_n) is negative, the interface remains in contact and responds as a linear spring. As the normal force becomes positive, contact is broken and no force is transmitted (unless KEYSUB(1)=1, then a small force is supplied to prevent a portion of the structure from being isolated).

In the tangential directions, for $F_n < 0$ and the absolute value of the tangential force (F_s) less than or equal to (μF_n), the interface does not slide and responds as a linear spring in the tangential direction. However, for $F_n < 0$ and $F_s > \mu F_n$, sliding occurs. Note that F_n is a variable and if contact is broken, the tangential function degenerates to a zero slope straight line through the origin (or of slope $K/10^6$, if KEYSUB(1)=1) indicating that no (or little) tangential force is required to produce sliding. Figure 5.4 shows the displacement functions for this element.

Assumptions and Restrictions. The gap size may be specified independently of the nodal point locations. Nodes I and J, however, may not be coincident since the nodal locations define the interface plane orientation. The element is defined such that a positive normal displacement (in the element coordinate system) of node J relative to node I tends to open the gap. Recall that the element coordinate system is defined by the I and J node locations. The nodes defining the element may have arbitrarily rotated nodal coordinate systems since a displacement transformation into the element coordinate system is included.

The friction coefficient may be input as a function of temperature and is evaluated at the average of the two node temperatures. For this nonlinear element an iterative solution procedure is required with the stiffness matrix re-formulation each iteration. Note, the effect of the element status changed in this iteration does not appear until the next iteration. Non-converged solutions are not in equilibrium. If GAP=0.0 (or blank), the element stiffness is included in the first iteration, unless START=3.0.

The element operates only in the Static (K20=0) and the Nonlinear Transient Dynamic (K20=4) analyses. If used in other analysis types, the element maintains its initial status throughout the analysis. Note, a gap condition capability is also included in the Reduced LInear Transient Dynamic (K20=5) analysis.

The element coordinate system orientation angles α and β (shown in Fig. 5.3) are computed by the program from the nodal point locations. α ranges from 0° to 360° and β from -90° to +90°. Elements lying along the $\pm Z$ axis are assigned values of α=0°, β=$\pm$90°, respectively. The element coordinate system for α=0°, β=90° is shown in Fig. 5.3. Elements lying off the Z-axis have their coordinate system oriented as shown for the general α, β position. Note, for α=90°, $\beta \to$90°, the element coordinate system flips 90° at the Z-axis.

5.3 ELEMENT LIBRARY FOR HEAT TRANSFER

For a heat transfer analysis, it is recommended to use the isoparametric quadrilateral temperature element (STIF55) and the two-dimensional conducting bar (STIF32).

5.3.1 Isoparametric Quadrilateral Temperature Element

The isoparametric quadrilateral temperature element can be used as a biaxial plane element or as an axisymmetric ring element with a two-dimensional thermal conduction capability. The element has four nodal points with a single degree of freedom, temperature, at each node. The isoparametric temperature element is a higher-order version of the two-dimensional linear temperature element (STIF35). The advantage of isoparametric temperature elements over linear temperature elements is that, for a given accuracy, the number of degrees of freedom necessary to describe the structure may be reduced. Accordingly, the data preparation time and the computer wave front solution time are also reduced.

The isoparametric temperature element is applicable to a two-dimensional, steady-state or transient, Thermal (K20=-1) analysis. If the model containing the isoparametric temperature element is also to be analyzed structurally, the element should be replaced by an equivalent structural element. The nodal temperatures determined from the isoparametric temperature element are applied to the corresponding structural nodal points. A summary of the isoparametric quadrilateral temperature element parameters is given in Table 5.8.

Input Data. The geometry, nodal point location, face numbers, loading and the coordinate system for the isoparametric temperature element are shown in Fig. 5.5 The isoparametric temperature element must have four nodes.

The thermal conductivities are defined in the global X and Y directions. The specific heat and the density may be assigned any values for steady-state solutions. An average internal heat generation rate may be applied to the element. All of the element lateral surfaces have convection capability and are numbered as shown in Fig. 5.5.

Output Data. The solution printout associated with the isoparametric temperature element is as shown in Table 5.9.

Theory. The theory on which the isoparametric temperature element is based as described for the STIF35 element, except for the temperature function. The temperature function in this element is not a linear polynomial, but includes additional incompatible temperature modes.

A 3x3 lattice of integration points is used for the numerical (Gaussian) integration procedure.

Assumptions and Restrictions. The isoparametric quadrilateral temperature element must not have a negative or a zero area. The element must lie in an X-Y plane and the X-axis must be the radial direction for axisymmetric problems. Also, axisymmetric structures should be modeled in the +X quadrants.

A triangular elememt may be formed by defining duplicate K and L node numbers. The extra mode shapes are automatically deleted for triangular elements so that a linear temperature element results. Face 3 should not be defined as a convection surface if nodes K and L are coincident.

If the thermal element is to be replaced by an analogous structural element with surface stresses requested, the thermal element should be oriented such that face 1 and/or face 3 is a free surface.

5.3.2 Two-dimensional Conducting Bar

The two-dimensional conducting bar is a uniaxial element with the ability to conduct heat between its nodal points. The element has a single degree of freedom, temperature, at each node point. The conducting bar is applicable to a two-dimensional (plane or axisymmetric) steady-state or transient Thermal (K20=-1) analysis.

If the model containing the conducting bar element is also to be analyzed structurally, the bar element should be replaced by an equivalent structural element. The node temperatures determined from the conducting bar element are applied to the corresponding structural element's nodal points. Structural elements accepting a transverse temperature gradient are given a uniform temperature in that direction by averaging the nodal temperatures. A summary of the two-dimensional conducting bar element parameters is given in Table 5.10.

Input Data. The geometry, nodal point locations, loading, and coordinate system for the conducting bar element are shown in Fig. 5.6. The element is defined by two nodal points, a cross-sectional area, and the material properties. Note that for an axisymmetric analysis, the area must be defined on a "per radian" basis. The specific heat and the density may be assigned any values for steady state solutions. The thermal conductivity is in the element longitudinal direction. An average internal heat generation rate may be applied to the element.

Output Data. The solution printout associated with the conducting bar element consists of the node temperatures, T(I) and T(J), which are included in the overall nodal temperature solution printout.

Theory. The temperature distribution for this element is obtained from the numerical solution of the following equation:

$$\rho C_p \frac{\partial T}{\partial t} = k_x \frac{\partial^2 T}{\partial x^2} + q$$

where K_x = thermal conductivity (Heat/Length*Time*Deg)

ρ = density (Weight (or Mass)/Volume)

C_p = specific heat (Heat/Weight (or Mass)*Deg)

q = internal heat generation rate (Heat/Volume*Time)

The temperature function is a linear polynomial of the form:

$$T(x) = C_1 + C_2 x$$

where the x-axis extends from node I to node J.

Assumptions and Restrictions. Heat is assumed to flow only in the longitudinal element direction. The element must be in an X-Y plane and the global X-axis must be the radial direction for axisymmetric problems. The element must not have a zero length, so nodes I and J must not be coincident.

5.4 ELEMENT LIBRARY FOR THERMAL STRESS ANALYSIS

It is recommended to use the two-dimensional isoparametric element (STIF42) and the two-dimensional interface element (STIF12) in a thermal stress analysis.

5.4.1 Two-dimensional Isoparametric Element

The two-dimensional isoparametric element is a higher-order version of the two-dimensional constant strain element (STIF2). The higher-order element gives a considerable improvement of accuracy over the constant strain element. The advantage of more complex elements over constant strain elements is that, for a given accuracy, the number of degrees of freedom necessary to describe the structure may be reduced. Accordingly, the data preparation time and the computer wave-front solution time is also reduced. The element has plasticity, but no creep or swelling capabilities. If all capabilities are needed, STIF2 should be used.

The isoparametric element is defined by four nodal points having two degrees of freedom at each node: translations in the nodal x and y directions. The element may be used as a biaxial plane element or as an axisymmetric ring element. An option is available to print out the stresses and strains on particular surfaces of the element when the surfaces are free surfaces of the structure. Other options are available to print stresses at the integration points or at the nodes. A summary of the two-dimensional isoparametric element parameters is given in Table 5.11.

Input Data. The geometry, nodal point locations, face numbers, loading, and the coordinate system for this element are shown in Fig. 5.7. The element input data includes four nodal points, a thickness (for a plane stress option only) and the orthotropic material properties. The element loading may be input as any combination of node temperatures, node fluences, and element pressures. The nodal forces should be input per unit of depth for a plane analysis (except for KEYSUB(1)=3) and per radian for an axisymmetric analysis.

The data input for the isoparametric element is as follows: 1) only the four-node element with four pressure surfaces is available; 2) creep and swelling capabilities are not included; 3) printout is avaiable on a second free surface for elastic solutions, at the numerical integration points and at the nodal points; and 4) the incompatible displacement modes may be suppressed.

Output Data. The solution printout associated with the two-dimensional isoparametric element is summarized in Table 5.12. Line K-L is analogous to line I-J except that it applies to the opposite surface. Figure 5.8 shows a schematic STIF42 element output.

Theory. The element formulation includes incompatible displacement modes. A 3x3 lattice of integration points is used with the numerical (Gaussian) integration procedure.

Assumptions and Restrictions. The area of the element must be positive. Zero area elements will print out an error message and contribute nothing to the total stiffness. Negative area elements print out a warning message and will not plot correctly. The numbering of the nodes should be counter-clockwise in the coordinate system shown in Fig. 5.7. The two-dimensional isoparametric element must lie in an X-Y plane and the global X-axis must be the radial direction for axisymmetric problems. An axisymmetric structure should be modeled in the +X quadrants.

A triangular element may be formed by defining duplicate K and L node numbers. The extra mode shapes are automatically deleted for triangular elements so that a constant strain element results. The surface stress printout is valid only for isotropic, elastic elements for which this face is a free surface. Surface strains, however, are valid for both isotropic and orthotropic elements. Surface stress printout on an X=0 face of axisymmetric elements or on the zero length side of a triangular element should not be requested.

5.4.2 Two-dimensional Interface Element

The two-dimensional interface element represents two plane or axisymmetric surfaces which may maintain or break physical contact and may slide relative to each other. The element is capable of supporting only compression in the direction in the normal to the surfaces and shear (Coulomb friction) in the tangential direction. The element has two degrees of freedom at each node: translations in the nodal x and y directions.

The element may be initially pre-loaded in the normal direction or it may be given a gap specification. A uniform stiffness acts in the normal and tangential directions. Because of the overall nonlinearity of the element an iterative solution procedure is required. A summary of the two-dimensional interface element parameters is given in Table 5.13.

Input Data. The geometry, nodal point locations, and the coordinate system for the interface element are shown in Fig. 5.9. The element is defined by two nodal points, an angle to define the interface plane, a stiffness, an initial displacement interference, and an initial element status. The stiffness, if left eq ual to zero, defaults to 10^5. An element coordinate system (x-y) is defined on the interface plane. The angle θ is input in degrees and is measured from the global X axis to the element-x axis. Note, the orientation of the interface plane is defined by the angle θ and not by the nodal point locations.

The stiffness, k, may be estimated from EA/L_{eff} where the parameters are determined from the adjacent element. The effective length, L_{eff}, is arbitrary, but may be on the order of 1/10 of the adjacent element length. The stiffness may also be computed from the maximum expected force divided by the maximum allowable surface displacement. In most cases k is several orders of magnitude greater than the other stiffnesses in series with it so that its exact value is not critical. The stiffness should be "per radian" for an axisymmetric analysis.

The initial displacement interference, δ, defines the displacement interference (if positive) or the gap size (if negative). The initial element status (START) is used to define the "previous" condition of the interface to be used at the start of the first iteration. This input is used to override the condition implied by the interference specification and is useful in anticipating the final interface configuration and reducing the number of iterations required for convergence. This procedure may also be used to continue a previous analysis .

The only material property required is the interface coefficient of friction, μ. A zero value should be used for frictionless surfaces. Temperatures may be specified at the element nodes. For some problems, a loss of contact or a sliding at the interface isolates a portion of the structural model not having sufficient displacement constraints. The KEYSUB(1) option may be used to maintain a small force across and along the interface, maintaining stability while causing a negligible inaccuracy in the analysis. The KEYSUB(2)=1 option should be used whenever friction is present and there is the possibility of some gap elements oscillating slightly between a sliding-sticking status.

Output Data. The solution printout associated with the two-dimensional interface element is summarized in Table 5.14.

The value USEP is the normal displacement between the interface surfaces at the end of this iteration, that is: USEP = $(u_y)_J - (u_y)_I - \delta$. This value is used in determining the normal force. For an axisymmetric analysis, the element forces are expressed per radian of circumference. The value USLIDE is the accumulated amount of surface sliding at the end of this iteration.

KTYPE describes the status of the element at the end of this iteration. It KTYPE=1, the gap is closed and no sliding occurs. If KTYPE=3, the gap is open. If at the end of an iteration KTYPE=3, an element stiffness of zero is used for the next iteration. A value of KTYPE=+2 indicates that node J moves to the right of node I as shown in Fig. 5.9. KTYPE=-2 indicates a negative slide. If no other effects are present and KEYSUB(2)=0, convergence occurs whenever the element status remains unchanged. For a frictionless surface (μ=0.0), the converged element status is either KTYPE=$\pm$2 or 3. Wherever KEYSUB(2) > 0, an element having sliding force oscillations within a defined tolerance on μF_y, resulting in an oscillating element status (KTYPE=1, 2, etc.), is accepted as converged. This tolerance range is usually within the uncertainty range of μ.

Theory. The displacement functions for the interface element can be separated into the normal and tangential directions because they are basically independent.

In the normal direction, when the normal force (F_y) is negative, the interface remains in contact and responds as a linear spring. As the normal force becomes positive, contact is broken and no force is transmitted (unless KEYSUB(1)=1, then a small force is supplied to prevent a portion of the structure from being isolated).

In the tangential direction, for $F_y < 0$ and the absolute value of the tangential force (F_x) less than or equal to (μF_y), the interface does not slide and responds as a linear spring in the tangential direction. However, for $F_y < 0$ and $F_x > \mu F_y$, sliding occurs . Note that F_x is a variable and if contact is broken, the tangential function degenerates to a zero slope straight line through the origin (or of slope $k/10^6$, if KEYSUB(1)=1) indicating that no (or little) tangential force is required to produce sliding. Figure 5.10 shows the displacement functions for this element.

Assumptions and Restrictions. The gap interference is specified independent of the nodal point locations. Nodes I and J may be coincident since the orientation of the interface plane is defined only by the angle θ. The element is defined such that a positive normal displacement (in the element coordinate system) of node J relative to node I tends to open the gap, as shown in Fig. 5.9. If, for a given set of conditions, nodes I and J are interchanged, or if the interface is rotated $\theta \pm 180°$, the gap element appears to act as a hook element, i.e., the gap closes as the nodes separate. The element may have rotated nodal coordinates since a displacement transformation into the element coordinate system is included.

The friction coefficient is evaluated at the average of the two node temperatures. The two-dimensional interface element must be defined in an X-Y plane and the global X axis must be the radial direction for axisymmetric problems. The element operates only in the Static (K20=0) and the Nonlinear Transient Dynamic (K20=4) analyses. If used in other analysis types, the element maintains its initial status throughout the analysis. Note, a gap condition capability is also included in the Reduced Linear Transient Dynamic (K20=5) Analysis.

No moment effects are included due to nodal points offset from a line perpendicular to the interface. If INTERFERENCE is zero (or blank), the element stiffness is included in the first iteration, unless START = 3.0. The element requires an iterative solution with the stiffness matrix reformulated each iteration. Note that if the element status changes within an iteration, the effect of the changed status is included in the next iteration non-converged iterations are not in equilibrium.

TABLE 5.1

ELEMENT SUMMARY TABLE

STIFFNESS NO.	NAME	DIM.	DOF	NODES	TYPE	
1	SPAR, 2-D	2	2	2	PLASTIC	
2	CONSTANT STRAIN ELEM.	2	2	3,4	PLASTIC	
3	ELASTIC BEAM, 2-D	2	3	2	LINEAR	
4	ELASTIC BEAM, 3-D	3	6	2	LINEAR	
5	ELASTIC SOLID (CST)	3	3	4,6,8	LINEAR	
6	ELAS. FLAT TRI. PLATE	2	3	3	LINEAR	
8	SPAR, 3-D	3	3	2	PLASTIC	
9	ELASTIC STRAIGHT PIPE	3	6	2	LINEAR	
10	CABLE	3	3	2	NON-LIN	
11	AXISYM. CONICAL SHELL	2	3	2	LINEAR	
12	INTERFACE ELEM. (2-D)	2	2	2	NON-LIN	
13	ELAS. FLAT TRI. SHELL	3	6	3	LINEAR	
14	SPRING-DAMPER	2,3	3	2	LINEAR	
15	MASS WITH ROTARY INER.	2	3	1	LINEAR	USE STIF21
16	MASS, 2-D	2	2	1	LINEAR	USE STIF21
17	MASS, 3-D	3	3	1	LINEAR	USE STIF21
18	SPRING, 2-D	2	2	2	LINEAR	USE STIF14
19	DAMPER, 2-D	2	2	2	LINEAR	USE STIF14
20	PLASTIC STRAIGHT PIPE	3	6	2	PLASTIC	
21	GENERAL MASS	2,3	6	1	LINEAR	
22	CORE SPACER AND GAP	2	1	2	NON-LIN	
23	PLASTIC BEAM, 2-D	2	3	2	PLASTIC	
24	TORSION SPRING-DAMPER	3	3	2	LINEAR	USE STIF14
25	AXISYM. HARMONIC QUAD.	2	3	4	LINEAR	
26	PLAS. FLAT TRI. PLATE	2	3	3	PLASTIC	
27	STIFF,DAMP,MASS MATRIX	3	6	2	LINEAR	
28	PL FLAT SHELL (3 TEMP)	3	6	3	PLASTIC	
29	CURVED PIPE (ELBOW)	3	6	3	LINEAR	
30	CONDUCTING SOLID	3	1	4,6,8	LINEAR	
31	RADIATION LINK	2 OR 3	1	2	NON-LIN	
32	CONDUCTING BAR, 2-D	2	1	2	LINEAR	
33	CONDUCTING BAR, 3-D	3	1	2	LINEAR	
34	CONVECTION LINK	2 OR 3	1	2	LINEAR	
35	LINEAR TEMPER. EL.	2	1	3,4	LINEAR	
36	HYDRAULIC CONDUCTANCE	2 OR 3	1	2	NON-LIN	
37	CONDUCTING FLAT SHELL	3	1	3	LINEAR	
38	FLUID COUPLING	2 OR 3	2	2	LINEAR	
39	SLIDING INTERFACE	2 OR 3	1	2	NON-LIN	USE STIF40
40	COMBINATION ELEMENT	2 OR 3	1	2	NON-LIN	

(CONTINUED ON NEXT PAGE)

TABLE 5.1 (CONTINUED)

STIFFNESS NO.	NAME	DIM.	DOF	NODES	TYPE
41	3-D ELAS. QUAD. MEMS.	3	3	4	LINEAR
42	LINEAR STRAIN ISOPAR.	2	2	4	PLASTIC
43	ELAS. FLAT RECT. SHELL	3	6	4	LINEAR
44	TAPERED UNSYM. BEAM	3	6	3	LINEAR
45	ISOPARAMETRIC SOLID	3	3	8	PLASTIC
46	ELAS. FLAT RECT. PLATE	2	3	4	LINEAR
47	TRANSV.HT.COND.SHELL	2 OR 3	1	5	LINEAR
48	PL.FLAT SHELL (S TEMP)	3	6	3	PLASTIC
49	PLASTIC SOLID	3	3	4,6,8	PLASTIC
50	SUPER-ELEMENT	3	-	-	LINEAR
52	INTERFACE ELEM. (3-D)	3	3	2	NON-LIN
53	LAMINATED SHELL	3	6	3	LINEAR
54	TAPER.UNSYM.BEAM (2-D)	2	3	2	LINEAR
55	ISOPAR.QUAD.TEMP.ELEM	2	1	4	LINEAR
56	FLUID FL-HT TRANS PIPE	3	2	2	NON-LIN
57	ISO.QUAD.SHELL TEMP.	3	1	4	LINEAR
58	PLASTIC HINGE ELEM.	3	6	2	NON-LIN
59	IMMERSED PIPE ELEM.	3	6	2	LINEAR
60	PLASTIC ELBOW	3	6	3	PLASTIC
61	AXISYM. HARMONIC SHELL	2	4	2	LINEAR
62	2-D WAVE ELEMENT	2	2	4	LINEAR
63	ELAS. FLAT QUAD. SHELL	3	6	4	LINEAR
65	3-D WAVE ELEMENT	3	2	8	LINEAR
66	TRANS. THERM-FLOW PIPE	3	2	2	NON-LIN
67	HT TRANS-ELECTRIC QUAD	2	2	4	ITERATIVE
68	HT TRANS-ELECTRIC LINE	3	2	2	ITERATIVE
69	HT TRAN-ELECTRIC SOLID	3	2	8	ITERATIVE
70	ISO. CONDUCTING SOLID	3	1	8	LINEAR
71	LUMPED THERMAL MASS	2 OR 3	1	1	LINEAR
75	AXISY HARMONIC TEMP EL	2	1	4	LINEAR

Table 5.2. Input of Element Parameters on ANSYS Program Data Cards

Element Parameter	Input on Card
NODE NUMBERS	E1
REAL CONSTANTS	D2 or E2
TEMPERATURES, FLUENCES	Q
PRESSURES	P
HEAT GENERATION RATES	Q
CONVECTION SURFACES	P
MATERIAL PROPERTY EQUATIONS	H
KEYSUB(N)	D

Table 5.3. Units of Input and Output Parameters

Input Parameter	Units
Area	L^2
Volume	L^3
Pressure	F/L^2
Moment of Inertia	L^4
Fluence (φt)	Neutrons/L^2
Density	M/L^3
Convection Coefficient	Q/L^2-t-T
Conductivity	Q/L-t-T
Specific Heat	Q/M-T
Heat Generation Rate (except for STIF71)	Q/L^3-t (Q/t)
Spring Constant	F/L
Damping Coefficient	F-t/L
Rotational Inertia	F-L-t^2

Output Parameters	Units
Stress	F/L^2
Strain	——
Moment or Torque	L-F
Twist	Radians
Heat Flow Rate	Q/t

TABLE 5.4

ISOPARAMETRIC SOLID ELEMENT - THREE DIMENSIONAL

SUBROUTINE NAME	STIF45	
NO. OF NODES PER ELEMENT	8	I,J,K,L,M,N,O,P
DEGREES OF FREEDOM PER NODE	3	UX,UY,UZ
REQUIRED REAL CONSTANTS	0	
TEMPERATURES	8	T(I),T(J),T(K),T(L),T(M),T(N), T(O),T(P)
PRESSURES	6	P(IJKL),P(IJNM),P(JKON),P(KLPO), P(LIMP),P(NMOP)
MATERIAL PROPERTY EQUATIONS	10	EX,EY,EZ,ALPX,ALPY,ALPZ, NUXY,NUYZ,NUXZ,DENS. GXY,GYZ,GXZ (OPTIONAL)
MATRICES CALCULATED	MASS, STIFFNESS	
PLASTICITY	YES	
CREEP AND SWELLING	NO	
FORCES SAVED ON TAPE	0	
KEYSUB(1)	0 - GENERAL 3-D APPLICATION 1 - GENERALIZED PLANE STRAIN OPTION	
KEYSUB(1A)	0 - USE 3X3X3 LATTICE OF INTEGRATION POIN (USED FOR INCREASED ACCURACY WITH WARPED ELEMENTS AND ELEMENTS HAVING HIGHLY NON-RECTANGULAR SHAPES) 1 - USE 2X2X2 LATTICE OF INTEGRATION POIN (KEYSUB(1A) IS INTERNALLY SET TO 1 F PLASTICITY SOLUTIONS)	
KEYSUB(1B)	0 - DISPLACEMENT FORMULATION INCLUDES THE EXTRA MODE SHAPES 1 - DISPLACEMENT FORMULATION DOES NOT INCLUDE THE EXTRA MODE SHAPES	

(CONTINUED ON NEXT PAGE)

TABLE 5.4 (CONTINUED)

KEYSUB(2)	0 - NO SURFACE STRESS OUTPUT 1 - PRINT OUT STRESSES FOR SURFACE 2 2 - PRINT OUT STRESSES FOR BOTH SURFACES 2 AND 4 (SURFACE STRESSES AVAILABLE FOR ISOTROPIC, ELASTIC MATERIALS ONLY) 3 - PRINT OUT SOLUTION AT EACH INTEGRATION POINT AS WELL AS AT CENTROID (FOR PLASTICITY SOLUTIONS ONLY. NOTE - ADDS 21 MORE LINES PER ELEMENT) 8 - PRINT STRESSES AT THE 8 NODES AS WELL AS AT CENTROID
SUBROUTINE DATE	4/30/72

TABLE 5.5

ISOPARAMETRIC SOLID ELEMENT - THREE DIMENSIONAL

ELEMENT PRINTOUT EXPLANATIONS

LABEL	NUMBER OF CONSTANTS	FORMAT	EXPLANATION
LINE 1			
SOLID	1	I5	ELEMENT NUMBER
NODES	8	8I5	NODES I,J,K,L,M,N,O,P
XC,YC,ZC	3	3F8.3	X,Y,Z COORDINATES OF ELEMENT CENTROID
TEMP	1	F6.0	ELEMENT AVERAGE TEMPERATURE
LINE 2			
EPS	6	6F9.6	EPX,EPY,EPZ,GAMMAXY,GAMMAYZ,GAMMAXZ (GLOBAL (ELASTIC STRAIN COMPONENTS)
SIG	6	6F8.0	SIGX,SIGY,SIGZ,TAUXY,TAUYZ,TAUXZ (GLOBAL)
LINE 3			
SIGPR	3	3F8.0	PRINCIPAL STRESSES SIG1,SIG2,SIG3
TAUMAX	1	F7.0	MAXIMUM SHEAR STRESS
EPPR	3	3F9.6	PRINCIPAL STRAINS EP1, EP2, EP3 (ISOTROPIC
VOL	1	F12.3	ELEMENT VOLUME
VM	1	F8.0	VON MISES EQUIVALENT STRESS
LINE IJNM SURFACE 2 STRESS CONDITIONS (PRINTED ONLY IF KEYSUB(2) IS GREATER THAN ZERO)			
AREA	1	F10.4	SURFACE AREA
TEMP	1	F9.0	AVERAGE SURFACE TEMPERATURE
XY STR	3	3F8.0	SIGX, SIGY, AND TAUXY (X AXIS PARALLEL TO THE AVERAGE OF LINES I-J AND M-N)
VMS	1	F8.0	VON MISES EQUIVALENT STRESS FOR THIS FACE
LINE IJNM SURFACE 2 STRESS CONDITIONS (CONTINUED)			
MAX-MIN STR	3	3F8.0	MAXIMUM, MINUMUM, AND MAXIMUM SHEAR STRESS ON SURFACE 2 OF THIS ELEMENT
A	1	F5.1	ANGLE OF PRINCIPAL STRESSES (MEASURED FROM LOCAL X TOWARD LOCAL Y)
STRAINS	3	3F9.6	EPSX, EPSY, AND GAMMAXY
PRESSURE	1	F8.0	SURFACE PRESSURE

(CONTINUED ON NEXT PAGE)

TABLE 5.5 (CONTINUED)

LINES KLPO SURFACE 4 STRESS CONDITIONS (PRINTED ONLY IF KEYSUB(2) = 2)

(SAME AS SURFACE 2 OUTPUT BUT APPLIED TO SURFACE 4)

LINES 4 AND 5 NON-LINEAR SOLUTION (PRINTED ONLY IF K13 IS GREATER THAN ZERO ON CARD C1)

EPPLAV	6	6F10.7	AVERAGE PLASTIC STRAINS AT CENTROID (X,Y,Z,XY,YZ,XZ)
EPORAV	6	6F10.7	AVERAGE ORIGIN SHIFT STRAINS AT CENTROID

LINE 6 NON-LINEAR SOLUTION (CONTINUED)

EPGNAV	1	F10.7	AVERAGE GENERALIZED STRAIN AT CENTROID
POSGAV	1	F10.4	AVERAGE GENERALIZED POISSONS RATIO AT CENTROID
SIGEAV	1	F10.2	AVERAGE EQUIVALENT STRESS AT CENTROID

NOTE - STRESSES AND STRAINS ARE PRINTED AFTER THE PLASTICITY CORRECTIONS.

****

TABLE 5.3

INTERFACE ELEMENT - THREE-DIMENSIONAL

SUBROUTINE NAME	STIF52	
NO. OF NODES PER ELEMENT	2	I,J
DEGREES OF FREEDOM PER NODE	3	UX,UY,UZ
REQUIRED REAL CONSTANTS	3	STIFFNESS,GAP,START A NEGATIVE GAP ASSUMES AN INITIAL INTERFERENCE CONDITION IF START = 0.0 OR BLANK, PREVIOUS STA OF ELEMENT DETERMINED FROM GAP INPUT IF START = 1.0, GAP PREVIOUSLY CLOSED AND NOT SLIDING IF START = 2.0, GAP PREVIOUSLY CLOSED AND SLIDING IF START = 3.0, GAP PREVIOUSLY OPEN
TEMPERATURES	2	T(I),T(J)
PRESSURES	0	
MATERIAL PROPERTY EQUATIONS	1	MU
MATRICES CALCULATED	STIFFNESS	
PLASTICITY	NO	
NON-LINEAR	YES	(ITERATIVE SOLUTION REQUIRED)
FORCES SAVED ON TAPE	2	NORMAL FORCE(FN),TANGENTIAL FORCE(FS)
KEYSUR(1)	0 - NO STIFFNESS ASSOCIATED WITH SEPARATED INTERFACE 1 - STIFFNESS*1.0E-6 ASSOCIATED WITH SEPARATED INTERFACE FOR NORMAL MOTION AND WITH SLIDING INTERFACE FOR TANGENTIAL MOTION	
KEYSUR(2)	0 - CONVERGENCE BASED ON UNCHANGING GAP STATU 1 - CONVERGENCE BASED ON CHANGING GAP STATUS WITHIN A 5 PERCENT UNCERTAINTY ON MU 2 - CONVERGENCE BASED ON CHANGING GAP STATUS WITHIN A 10 PERCENT UNCERTAINTY ON MU	
SUBROUTINE DATE	5/30/73	

****

TABLE 5.7

INTERFACE ELEMENT - THREE-DIMENSIONAL

ELEMENT PRINTOUT EXPLANATIONS

LABEL	NUMBER OF CONSTANTS	FORMAT	EXPLANATION
LINE 1			
3-D GAP	1	I5	ELEMENT NUMBER
NODES	2	2I5	NODES I AND J
USEP,USLIDE(Y,Z)	2	3F9.5	GAP SIZE, SLIDING DISTANCE IN LOCAL DIRECTION, SLIDING DISTANCE IN LOC. Z DIRECTION
KTYPE	1	I2	INTERFACE CONDITION INDICATOR 1 - RIGID CONTACT 2 - SLIDING CONTACT 3 - FREE
KOLD	1	I2	KTYPE VALUE OF THE PREVIOUS ITERATI(
LINE 2			
FN	1	G14.6	NORMAL FORCE (ALONG I-J LINE)
FS	1	G14.6	TANGENTIAL FORCE (VECTOR SUM)

****

TABLE 5.8

ISOPARAMETRIC QUADRILATERAL TEMPERATURE ELEMENT

SUBROUTINE NAME	STIF55	
NO. OF NODES PER ELEMENT	4	I,J,K,L
DEGREES OF FREEDOM PER NODE	1	TEMP
REQUIRED REAL CONSTANTS	0	
HEAT GENERATION RATES	1	AVERAGE
CONVECTION SURFACES	4	IJ,JK,KL,LI
MATERIAL PROPERTY EQUATIONS	4	KXX,KYY,DENS,C (GLOBAL)
MATRICES CALCULATED	CONDUCTIVITY, SPECIFIC HEAT	
KEYSUB(1)	0 - PLANE 1 - AXISYMMETRIC	
KEYSUB(1A)	0 - INCLUDE EXTRA TEMPERATURE SHAPES 1 - SUPPRESS EXTRA TEMPERATURE SHAPES	
KEYSUB(2)	0 - NO CONVECTION SURFACE PRINTOUT 1 - PRINT OUT HEAT FLOW RATE FROM CONVECTION SURFACES	
SUBROUTINE DATE	6/22/73	

TABLE 5.9

ISOPARAMETRIC QUADRILATERAL TEMPERATURE ELEMENT

ELEMENT PRINTOUT EXPLANATIONS

LABEL	NUMBER OF CONSTANTS	FORMAT	EXPLANATION
LINE 1 (PRINTED ONLY IF KEYSUB(2) = 1)			
ELEMENT	1	I5	ELEMENT NUMBER
NODES	2	2I5	CONVECTION FACE NODES (I,J OR J,K OR K,L OR L,I)
FACE	1	I2	CONVECTION FACE NUMBER
AREA	1	F8.4	CONVECTION SURFACE AREA
TAVG.TBULK	2	2F6.0	AVERAGE FACE TEMP, FLUID BULK TEMP.
HEAT FLOW	1	G11.4	HEAT FLOW RATE ACROSS FACE
LINES 2,3,4 (SAME AS ABOVE FOR OTHER CONVECTION SURFACES, IF DEFINED)			

TABLE 5.10

CONDUCTING BAR - TWO-DIMENSIONAL

SUBROUTINE NAME	STIF32	
NO. OF NODES PER ELEMENT	2	I,J
DEGREES OF FREEDOM PER NODE	1	TEMP
REQUIRED REAL CONSTANTS	1	AREA
HEAT GENERATION RATES	1	AVERAGE
CONVECTION SURFACES	0	
MATERIAL PROPERTY EQUATIONS	3	KXX,DENS,C
MATRICES CALCULATED	CONDUCTIVITY,SPECIFIC HEAT	
ELEMENT PRINTOUT	NONE	
SUBROUTINE DATE	7/01/70	

****

TABLE 5.11

TWO-DIMENSIONAL ISOPARAMETRIC ELEMENT

SUBROUTINE NAME	STIF42	
NO. OF NODES PER ELEMENT	4	I,J,K,L (NUMBER COUNTER-CLOCKWISE)
DEGREES OF FREEDOM PER NODE	2	UX,UY
REQUIRED REAL CONSTANTS	0 1	IF KEYSUB(1) = 0,1,2 THICKNESS, IF KEYSUB(1) = 3
TEMPERATURES	4	T(I),T(J),T(K),T(L)
PRESSURES	4	P(1),P(2),P(3),P(4)
MATERIAL PROPERTY EQUATIONS	6	IF PLANE STRESS - EX,EY,NUXY,ALPX,ALPY,DENS GXY (OPTIONAL)
	10	IF AXISYM OR PLANE STRAIN - EX,EY,EZ,NUXY,NUYZ,NUXZ,ALPX,ALPY, ALPZ,DENS GXY (OPTIONAL)
MATRICES CALCULATED	MASS,STIFFNESS	
PLASTICITY	YES	
CREEP AND SWELLING	NO	
FORCES SAVED ON TAPE	26	SIGMAX,SIGMIN,TAUMAX,SIGZ,SIGE, EPGEN,SIGX,SIGY,TAUXY,TEMPERATURE, 4 ELASTIC STRAINS, 4 PLASTIC STRAI 4 O.SHIFT STRAINS, 4 THERMAL STRAI
KEYSUB(1)	0 - PLANE STRESS 1 - AXISYMMETRIC 2 - PLANE STRAIN (Z STRAIN = 0.0) 3 - PLANE STRESS WITH THICKNESS INPUT	
KEYSUB(1A)	0 - DISPLACEMENT FORMULATION INCLUDES THE EXTRA MODE SHAPES 1 - DISPLACEMENT FORMULATION DOES NOT INCLUDE THE EXTRA MODE SHAPES	

(CONTINUED ON NEXT PAGE)

TABLE 5.II (CONTINUED)

KEYSUB(2)	0 - NO SURFACE STRESS PRINTOUT 1 - PRINT OUT STRESSES FOR SURFACE I-J 2 - PRINT OUT STRESSES FOR BOTH SURFACES I-J AND K-L (SURFACE STRESS PRINTOUT AVAILABLE ONLY FOR ISOTROPIC, ELASTIC MATERIALS) 3 - PRINT OUT SOLUTION AT ALL INTEGRATION POINTS AS WELL AS AT CENTROID (FOR PLASTIC SOLUTIONS ONLY. ADDS 15 MORE LINES PER ELEMENT) 4 - PRINT STRESSES AT THE 4 NODES AS WELL AS AT CENTROID (KEYSUB(1A) MUST = 0)
KEYSUB(2A)	0 - PRINT SOLUTION AT ELEMENT CENTROID 1 - REPEAT LINES 1 AND 2 OF SOLUTION FOR ALL OTHER INTEGRATION POINTS (ADDS 16 MORE LINES PER ELEMENT)
SUBROUTINE DATE.	4/30/72

TABLE 5.12

TWO-DIMENSIONAL ISOPARAMETRIC ELEMENT

ELEMENT PRINTOUT EXPLANATIONS

LABEL	NUMBER OF CONSTANTS	FORMAT	EXPLANATION
LINE 1			
ELEM	1	I5	ELEMENT NUMBER
NODES	4	4I5	NODES I, J, K, AND L
VOL	1	F10.4	VOLUME OF ELEMENT
MATERIAL	1	I2	MATERIAL NUMBER
STRESS INT.	1	F8.0	STRESS INTENSITY IN THIS ELEMENT
SIGEFF	1	F8.0	VON MISES EQUIVALENT STRESS
LINE 2			
X	1	F6.2	X COORDINATE OF CENTROID OF ELEMENT
Y	1	F6.2	Y COORDINATE OF CENTROID OF ELEMENT
T	1	F5.0	AVERAGE TEMPERATURE OF ELEMENT
XY STR	4	4F8.0	SIGX, SIGY, TAUXY, AND SIGZ (SIGZ=0.0 FOR PLANE STRESS ELEMENTS)
MAX-MIN STR	3	3F8.0	SIGMAX, SIGMIN, AND TAUMAX (IN-PLANE PRINCIPAL STRESSES)
A	1	F5.1	ANGLE OF PRINCIPAL STRESSES RELATIVE TO THE GLOBAL X-Y AXES
LINE I-J SURFACE I-J STRESS CONDITIONS (PRINTED ONLY IF KEYSUB(2) IS 1 OR 2)			
T	1	F5.0	AVERAGE TEMPERATURE OF I-J SURFACE
STRAIN	3	3F10.7	ELASTIC SURFACE STRAIN COMPONENTS (PARALLEL, PERPENDICULAR, Z OR HOOP)
STRESS	3	3F8.0	ELASTIC SURFACE STRESS COMPONENTS (PARALLEL, PERPENDICULAR, Z OR HOOP)
	2	2F9.0	SURFACE STRESS INTENSITY, SURFACE VON MISES EQUIVALENT STRESS
LINE K-L SURFACE K-L STRESS CONDITIONS (PRINTED ONLY IF KEYSUB(2) = 2)			
(SAME AS LINE I-J ABOVE BUT APPLIED TO FACE K-L)			
LINES 3 AND 4 NON-LINEAR SOLUTION (PRINTED ONLY IF K13 IS GREATER THAN ZERO ON CARD C1)			
EPEL	4	4F10.7	ELASTIC STRAIN COMPONENTS (X,Y,XY,Z)
EPPL	4	4F10.7	PLASTIC STRAIN COMPONENTS (X,Y,XY,Z)
LINE 5 NON-LINEAR SOLUTION (CONTINUED)			
EPORIG	4	4F10.7	SHIFT OF ORIGIN OF STRESS-STRAIN CURVE DUE TO PREVIOUS LOAD CYCLES
EPGEN	1	F10.7	EQUIVALENT STRAIN
POSGEN	1	F10.4	EFFECTIVE POISSONS RATIO
SIGE	1	F10.2	EQUIVALENT STRESS

NOTE 1 - STRESSES AND STRAINS ARE PRINTED AFTER THE PLASTICITY CORRECTIONS.

NOTE 2 - FOR AXISYMMETRIC SOLUTIONS, THE X,Y,XY, AND Z STRESS AND STRAIN OUTPUTS CORRESPOND TO THE RADIAL, AXIAL, IN-PLANE SHEAR, AND HOOP STRESSES AND STRAINS, RESPECTIVELY.

TABLE 5.13

INTERFACE ELEMENT - TWO DIMENSIONAL

SUBROUTINE NAME	STIF12	
NO. OF NODES PER ELEMENT	2	I,J
DEGREES OF FREEDOM PER NODE	2	UX,UY
REQUIRED REAL CONSTANTS	4	THETA,STIFFNESS,INTERFERENCE,START. A NEGATIVE INTERFERENCE ASSUMES AN INITIALLY OPEN GAP IF START = 0.0 OR BLANK, PREVIOUS CONDITION OF GAP DETERMINED FROM INTERFERENCE IF START = 1.0, GAP PREVIOUSLY CLOS AND NOT SLIDING IF START = 2.0, GAP PREVIOUSLY CLOS AND SLIDING RIGHT IF START = -2.0, GAP PREVIOUSLY CLOSED AND SLIDING LEFT IF START = 3.0, GAP PREVIOUSLY OPEN
TEMPERATURES	2	T(I),T(J)
PRESSURES	0	
MATERIAL PROPERTY EQUATIONS	1	MU
MATRICES CALCULATED	STIFFNESS	
PLASTICITY	NO	(NON-LINEAR ELEMENT)
FORCES SAVED ON TAPE	2	NORMAL FORCE(FN), TANGENTIAL FORCE(F
KEYSUB(1)	0 - NO STIFFNESS ASSOCIATED WITH SEPARATED INTERFACE 1 - STIFFNESS*1.0E-6 ASSOCIATED WITH NORMAL STIFFNESS FOR SEPARATED INTERFACES AND WITH TANGENTIAL STIFFNESS FOR SLIDING INTERFACES	
KEYSUB(2)	0 - CONVERGENCE BASED ON UNCHANGING ELEMEN STATUS 1 - CONVERGENCE BASED ON CHANGING ELEMENT STATUS WITHIN A 5 PERCENT TOLERANCE C MU 2 - CONVERGENCE BASED ON CHANGING ELEMENT STATUS WITHIN A 10 PERCENT TOLERANCE MU	
SUBROUTINE DATE	4/04/72	

TABLE 5.14

INTERFACE ELEMENT - TWO DIMENSIONAL

. ELEMENT PRINTOUT EXPLANATIONS

LABEL	NUMBER OF CONSTANTS	FORMAT	EXPLANATION
ELEM	1	I5	ELEMENT NUMBER
NODES	2	2I5	NODES I AND J
USEP,USLIDE	2	2F9.5	GAP SIZE, SLIDING DISTANCE
FN	1	G13.5	NORMAL FORCE
FS	1	G13.5	TANGENTIAL FORCE
KTYPE	1	I2	ELEMENT STATUS
			1 - GAP CLOSED, NO SLIDING
			2 - SLIDING CONTACT (NODE J MOVING TO RIGHT OF NODE I)
			-2 - SLIDING CONTACT (NODE J MOVING TO LEFT OF NODE I)
			3 - GAP OPEN
OLD	1	I2	KTYPE VALUE OF THE PREVIOUS ITERATION

++

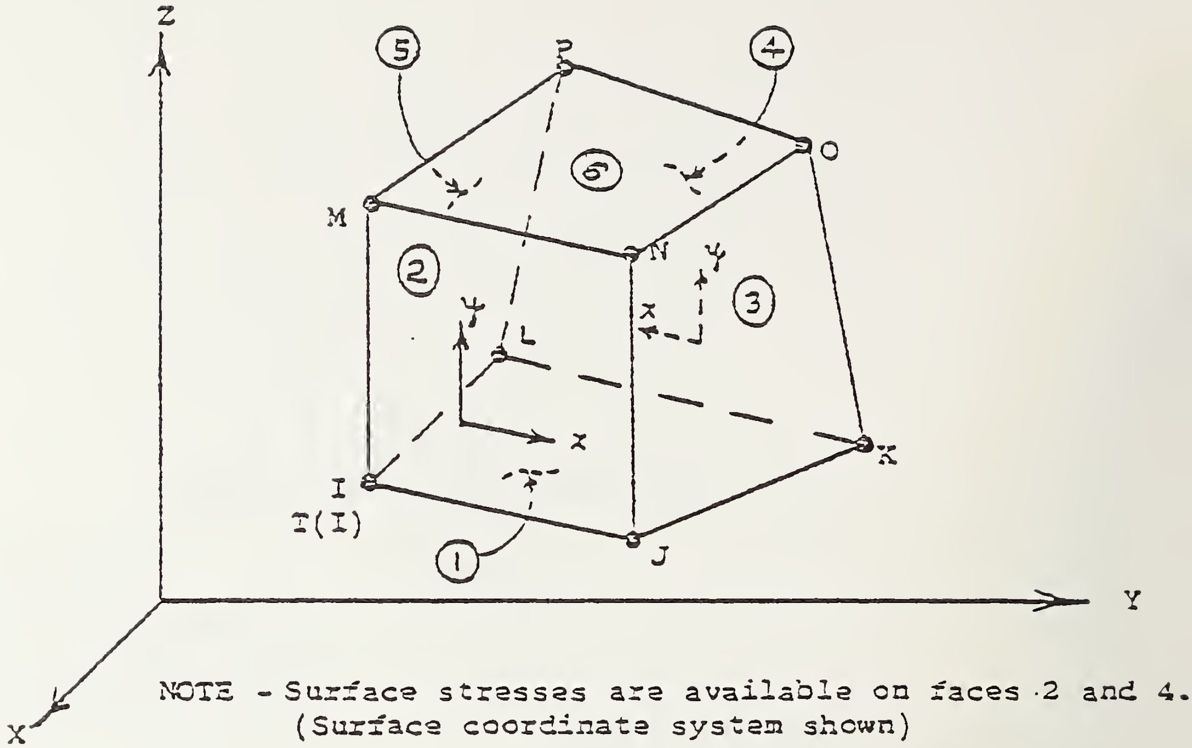


Figure 5.1. Three-Dimensional Isoparametric Solid Element

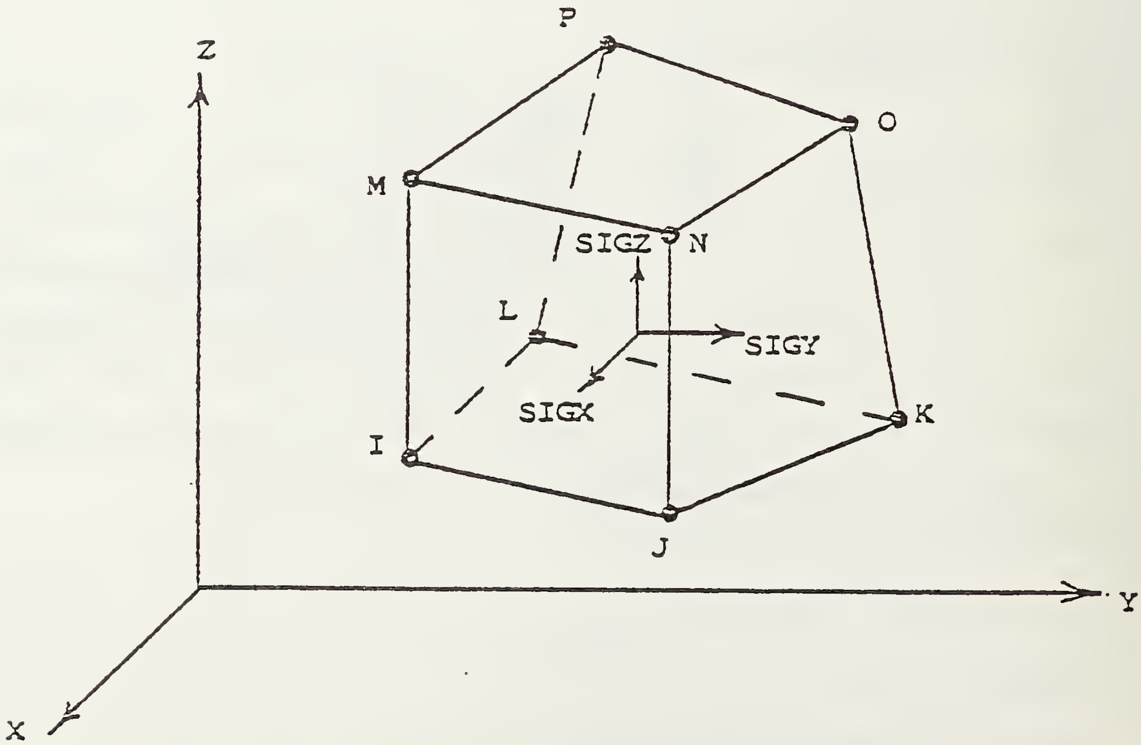


Figure 5.2. Three-Dimensional Isoparametric Solid Element Output

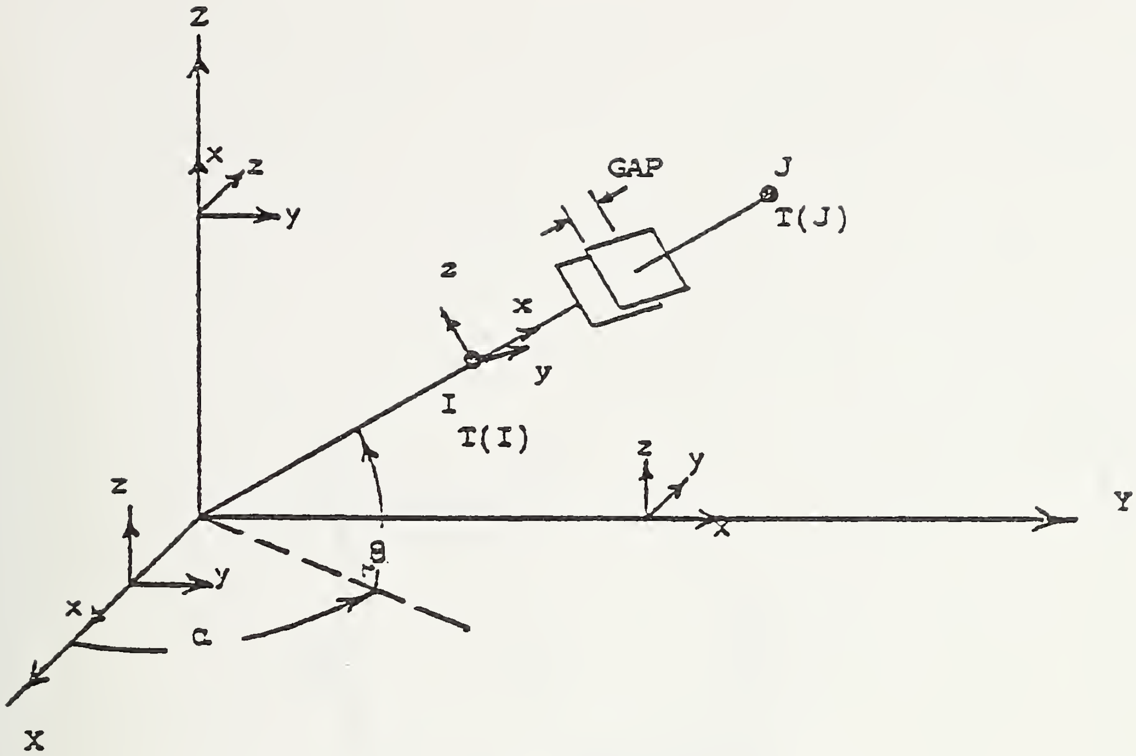


Figure 5.3. Three-Dimensional Interface Element.

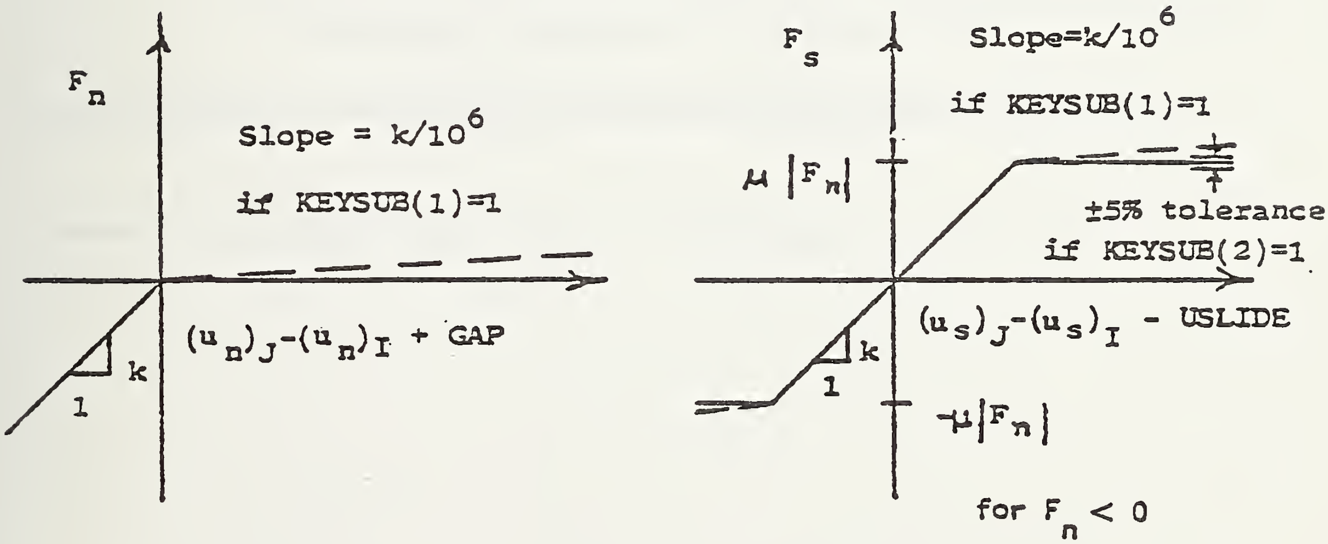


Figure 5.4. Three-Dimensional Interface Element Output.

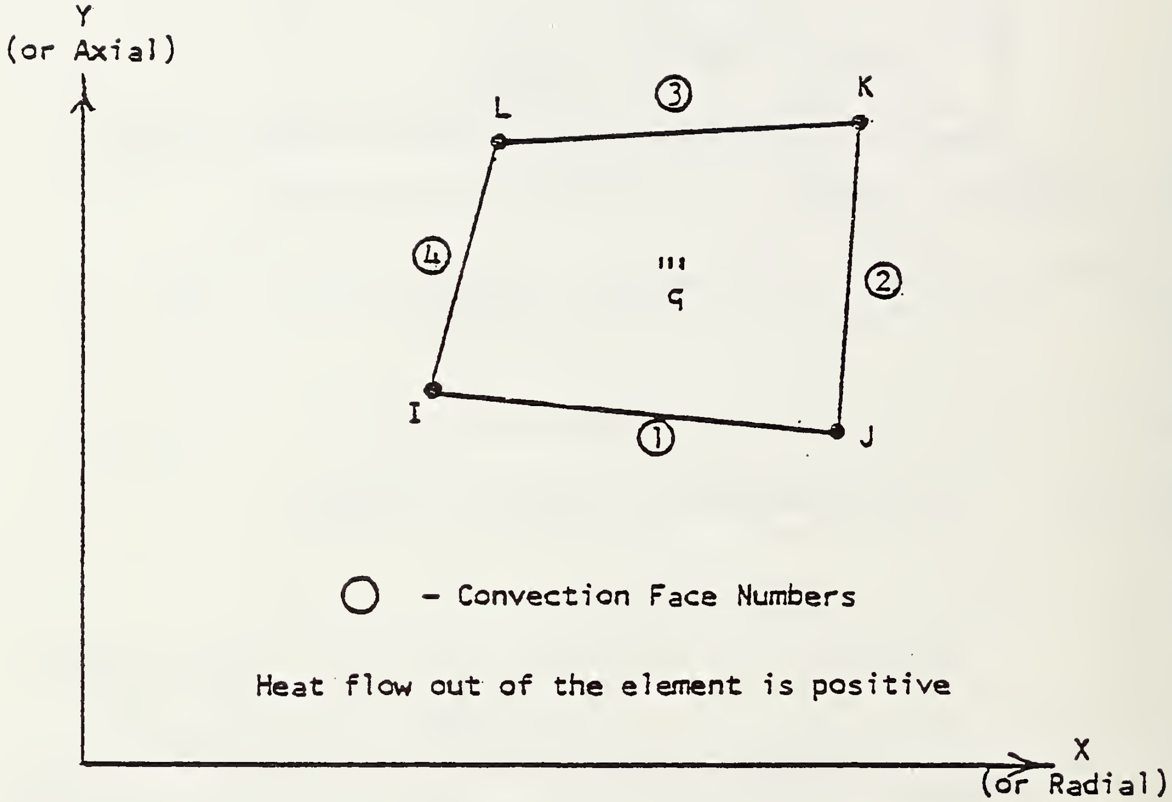


Figure 5.5. Isoparametric Quadrilateral Temperature Element

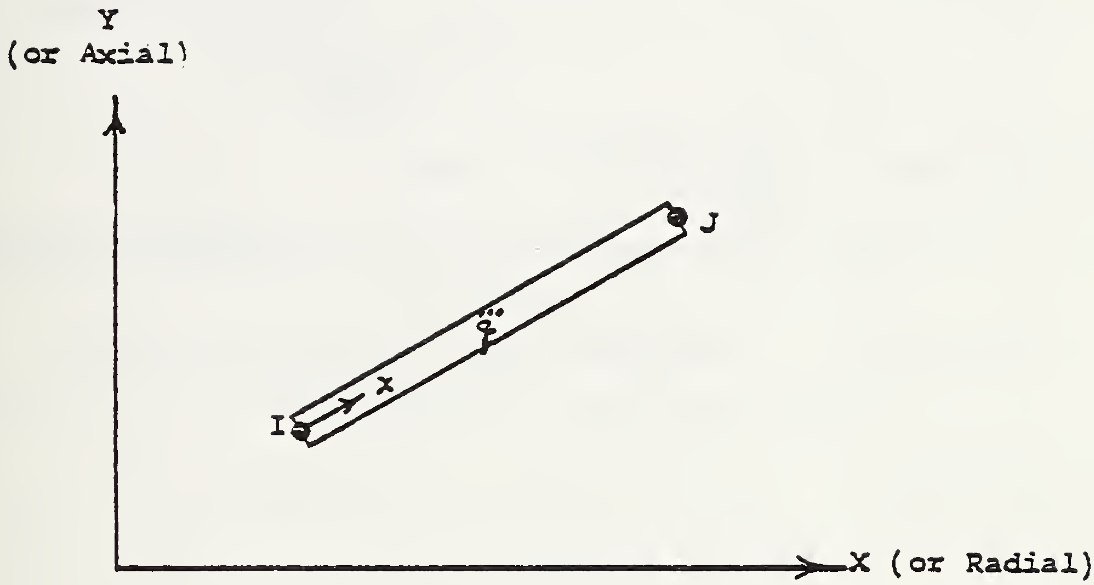


Figure 5.6. Two-Dimensional Conducting Bar Element

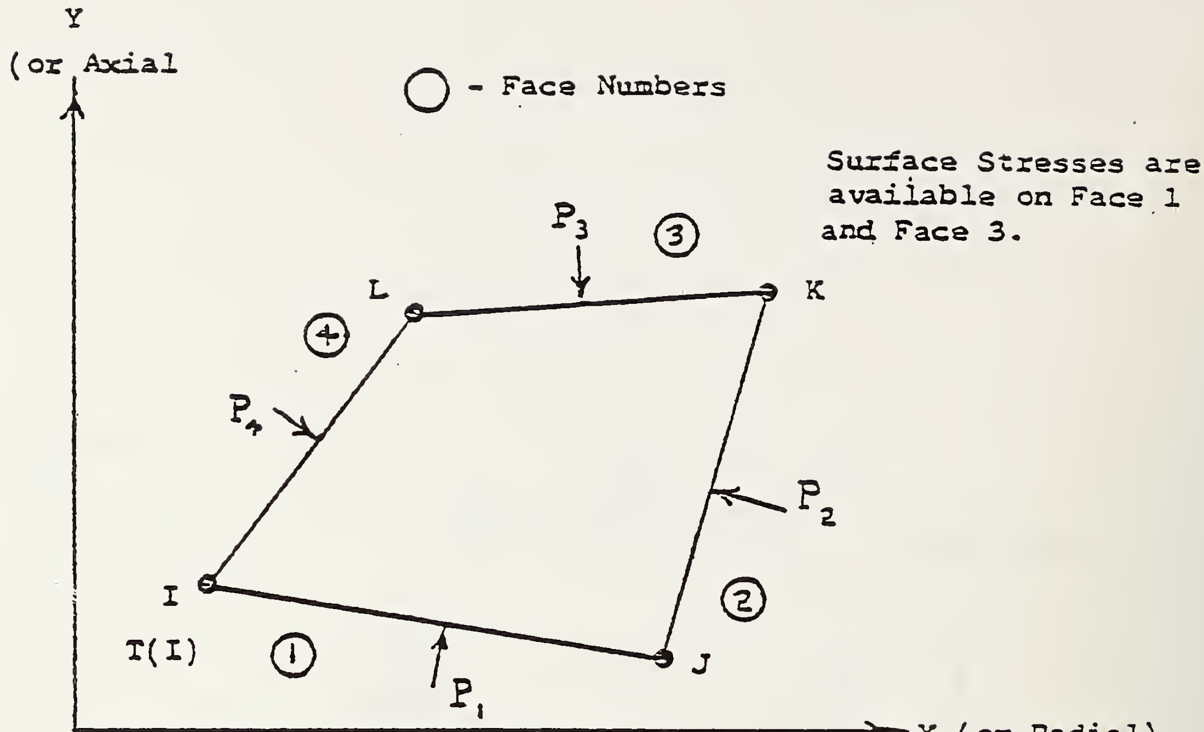


Figure 5.7. Two-Dimensional Isoparametric Element

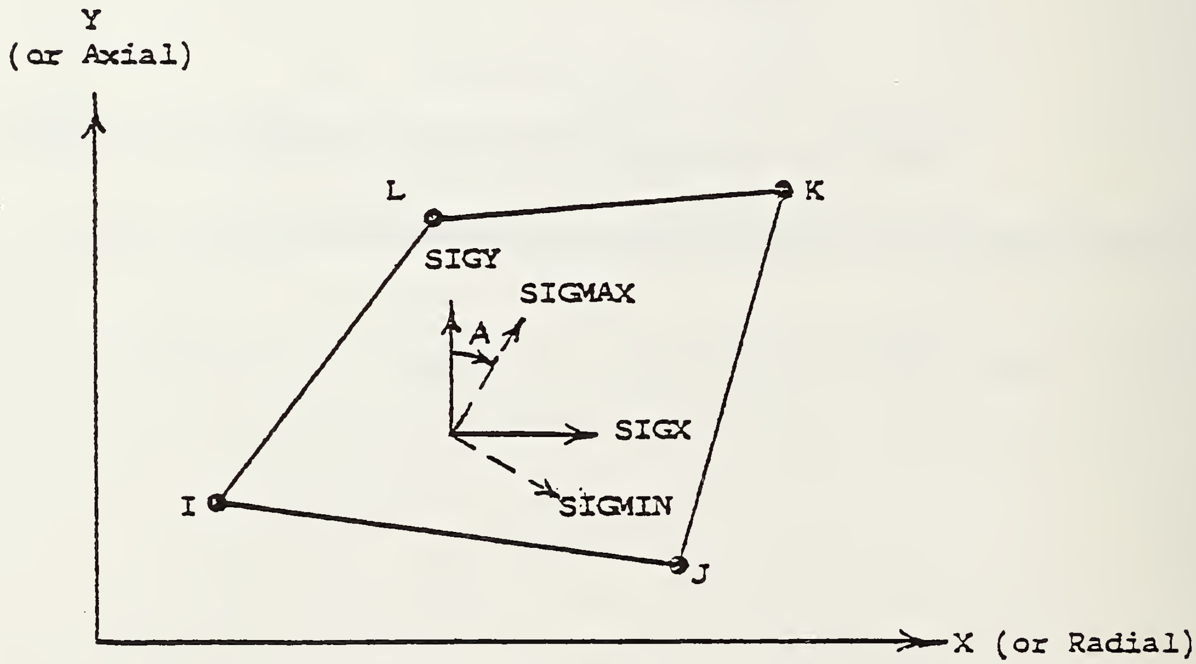


Figure 5.8. Two-Dimensional Isoparametric Element Output

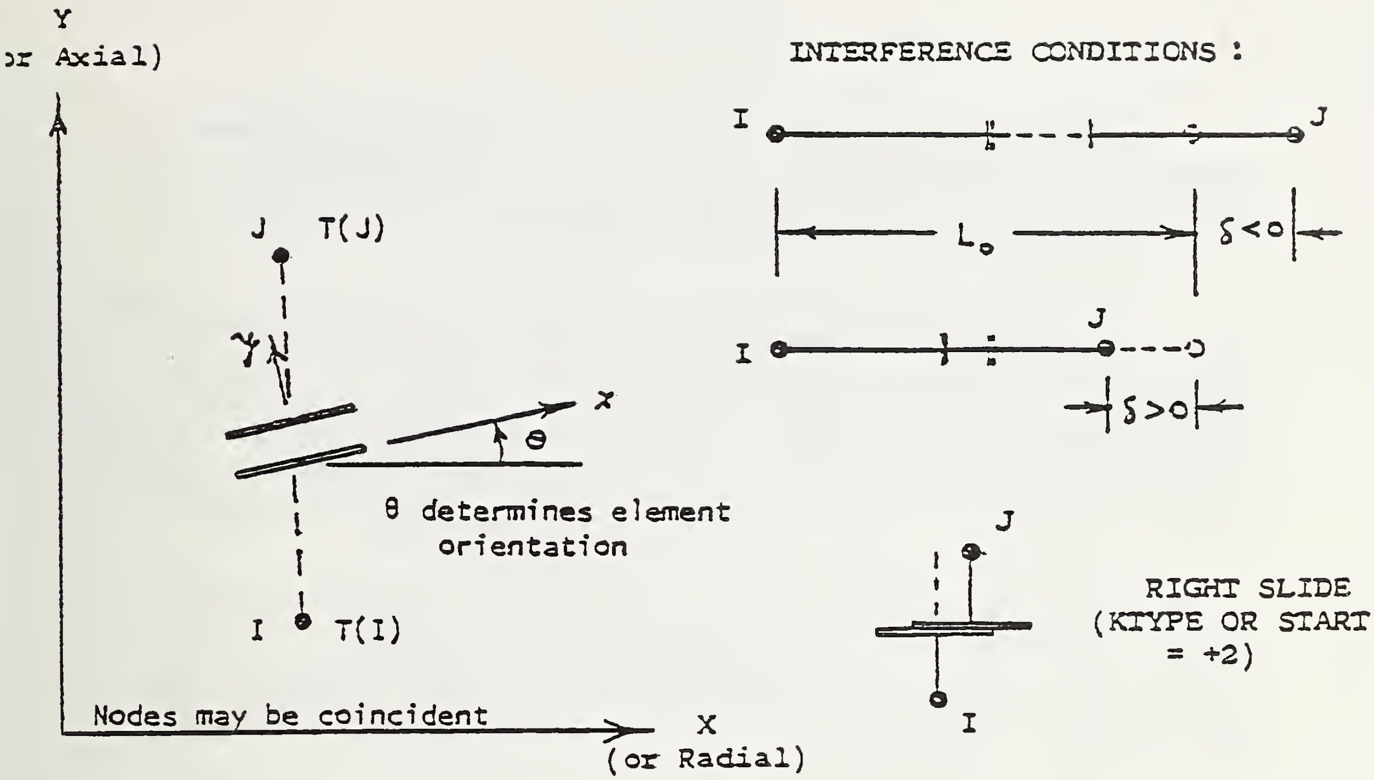


Figure 5.9. Two-Dimensional Interface Element

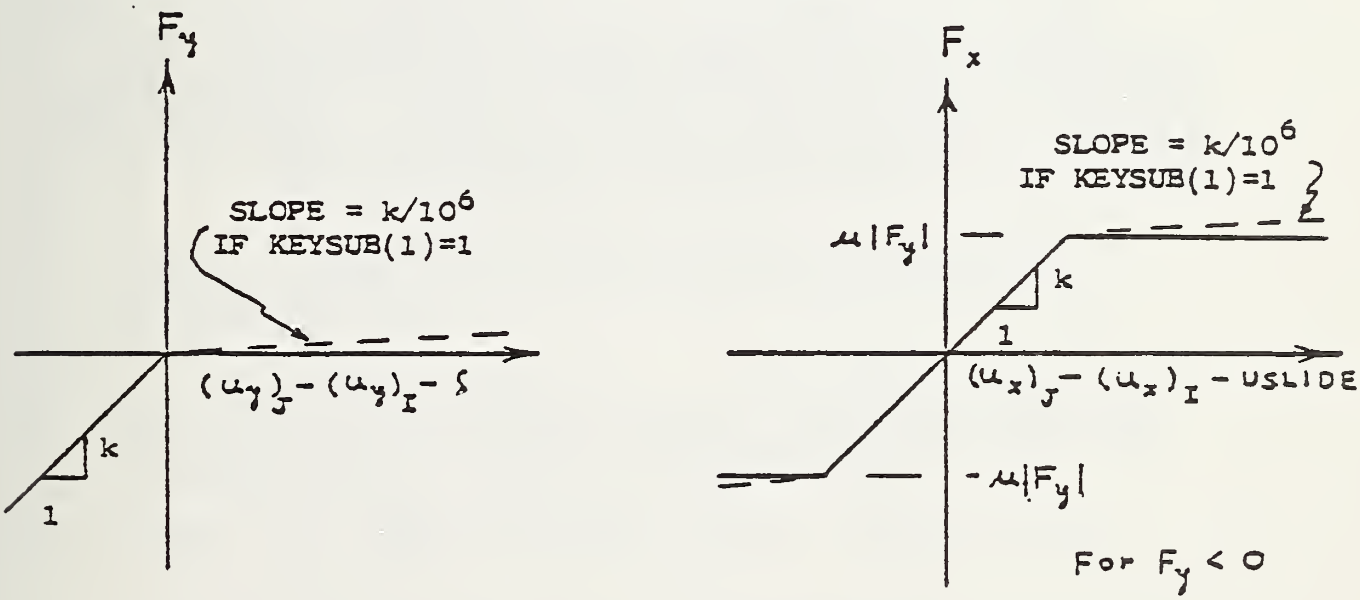


Figure 5.10. Two-Dimensional Interface Element Output

CHAPTER 6
CONCLUSION

The manual for ANSYS for analysis of anchored pavements has been prepared to provide the user a ready reference for analyzing the response of anchored pavement system subjected to vehicle static loads, moisture variation, and temperature variations.

The manual is prepared so that it can be used with a minimum number of references. For preprocessing, Chapter 3 provides the details of a program developed at IIT called FEMESH. The User's Guide for ANSYS has its own preprocessing subroutines, however, the FEMESH is more efficient for preprocessing the anchored pavement system. If any postprocessing (post-plotting) is desired, the User's Guide for ANSYS should be consulted. In the particular analysis performed, postprocessing was not utilized as plotting was done by hand.

The computer program provides the numerical values of stresses, strains, deflections in all elements of various materials. There is no practical limit of restriction of material numbers, that is the program can be used with different materials in any direction. For heat transfer, the program provides the distribution of temperature versus time at any point.

ANSYS in general has the capability of obtaining response of the pavement system under transient dynamic loads, however, this has not been incorporated in this manual.

The most noteworthy point for the ANSYS program is the wave front solution and certain limitations caused by the said solution. The ordering of nodes therefore must be done to minimize the size of wave front as has been explained in detail in Chapter 4. The program has been found versatile and capable of solving complex geometrical structures resting on complex geologically earth mass.

REFERENCES

1. Bathe, K.J. and Wilson, E. L., "Numerical Methods in Finite Element Analysis," Prentice-Hall, Inc., New Jersey, 1976.

2. Cook, R. D., "Concepts and Applications of Finite Element Analysis," John Wiley and Sons, New York, 1974.

3. Desai, C. S. and Abel, J. F., "Introduction to the Finite Element Method," Von Nostrand Company, New York, 1972.

4. DeSalvo, G. J., "ANSYS Verification Manual," Swanson Systems Inc., 1976.

5. DeSalvo, G. J. and Kohnke, P. C., "ANSYS Introductory Manual," Swanson Analysis Systems Inc., 1975.

6. DeSalvo, G. J. and Swanson, J. A., "ANSYS Examples Manual," Swanson Analysis Systems Inc., 1972.

7. DeSalvo, G. J. and Swanson, J. A., "ANSYS User's Manual (Revision 2)," Swanson Analysis Systems Inc., 1975.

8. DeSalvo, G. J. and Swanson, J. A., "ANSYS User's Manual (Revision 3)," Swanson Analysis Systems Inc., 1978.

9. FORTRAN Extended Reference Manual, Publication No. 60497800, Control Data Corporation.

10. Guyan, R. J., "Reduction of Stiffness and Mass Matrices," AIAA Journal, Vol. 3, No. 2, Feb. 1965.

11. INTERCOM Reference Manual, Publication No. 60494600, Control Data Corporation.

12. Irons, B. M., "A Frontal Solution Program for Finite Element Analysis," International Journal for Numerical Methods in Engineering, Vol. 2, No. 1, Tan., p.p. 5-23, (Discussion May, 1970, p. 149), 1970.

13. Jones Jr., R. F. and Costello, M. G., "A Solution Procedure for Nonlinear Structural Problems," Numerical Solution of Nonlinear Structural Problems, ASME, pp. 157-169, 1973.

14. Kohnke, P. C., "ANSYS Theoretical Manual," Swanson Analysis Systems Inc., 1977.

15. Kohnke, P. C. and Swanson, J. A., "Thermo-Electric Finite Elements," International Conference on Numerical Methods in Electrical and Magnetic Field Problems, Santa Margherita Ligure, Italy, June 1-4, 1976.

16. Lekhnitskii, S. G., "Theory of Elasticity of an Anisotropic Elastic Body," Holden-Day, San Francisco, 1963.

17. Loader Reference Manual, Publication No. 60429800, Control Data Corporation.

18. Melosh, R. J. and Bamford, R. M., "Efficient Solution of Load-Deflection Equations," Journal of the Structural Division, ASCE, Vol. 95, No. ST4, Proc. Paper 6510, pp. 661-676, (Discussions, Dec. 1976, Jan., Feb., May 1970, Closure Feb. 1971), April 1969.

19. NOS/BE Reference Manual, Publication No. 60493800, Control Data Corporation.

20. NOS/BE Users Guide, Publication No. 60494000, Control Data Corporation.

21. Przemieniecki, J. S., "Theory of Matrix Structural Analysis," McGraw-Hill, 1968.

22. Ralston, A. and Wilf, H. S., "Mathematical Methods for Digital Computers," John Wiley & Sons Inc., New York, 1962.

23. UPDATE Reference Manual, Publication No. 60449900, Control Data Corporation.

24. Wilson, E. L., Taylor, R. L., Doherty, W. P., and Ghaboussi, J., "Incompatible Displacement Models," Numerical and Computer Methods in Structural Mechanics, Edited by S. J. Fenves, et al., Academic Press Inc., New York and London, pp. 43-57, 1973.

25. Zienkiewicz, O. C., "The Finite Element Method in Engineering Science," McGraw-Hill Company, London, 1971.

APPENDIX A

NOTATION

The notation defined below is used throughout the appendices B and C.

General

Term	Meaning
$[B]$	Strain-displacement matrix
$[C],[\bar{C}]$	Damping matrix, thermal damping matrix
$[D]$	Stress-strain matrix
E	Young's modulus
$\{F\}$	Force vector
$[K],[\bar{K}]$	Stiffness matrix, conductivity matrix
$[M]$	Mass matrix
$\{Q\}$	Heat flow vector
$\{T\}$	Temperature vector
$[T_R]$	Local to global conversion matrix
$u,v,w,\{u\}$	Displacement, displacement vector
δU	Virtual internal work
δV	Virtual external work
x,y,z	Element coordinates
X,Y,Z	Nodal coordinates (usually global cartesian)
α	Coefficient of thermal expansion
ε	Strain
ν	Poisson's ratio
σ	Stress

Superscripts and Subscripts on [M], [C], [K], {u}, {T}, and/or {F}

No subscript implies the total matrix in final form, ready for solution.

a	nodal effects caused by an acceleration field
c	convection surface
cr	creep
e	based on element in nodal coordinates
g	internal heat generation
ℓ	based on element in element coordinates
ld	large displacement
m	master
n	nodal effects caused by externally applied loads
pl	plasticity
pr	pressure
s	slave
sw	swelling
t	thermal
—	(bar over term) heat transfer matrices
^	(flex over term) reduced matrices and vectors
•	(dot over term) time derivative

APPENDIX B

ANALYSIS PROCEDURES

This section of the manual is designed to give users an understanding of the theoretical basis of each analysis type. The derivation of the individual element matrices and vectors is discussed in Appendix C.

In the matrix displacement method of analysis based upon finite element idealization, the structure being analyzed must be approximated as an assembly of discrete structural elements connected at a finite number of points (called nodal points). If the force-displacement relationship for each of these discrete structural elements is known (the element "stiffness matrix") then the force-displacement relationship for the entire structure can be assembled using standard matrix methods.

Figure B1 gives a summary of the ANSYS analysis procedures available and may be used as a guide in selecting which type to use. Each of the analysis procedures is described in the following sections.

STATIC ANALYSIS

The overall equilibrium equations for static analysis are:

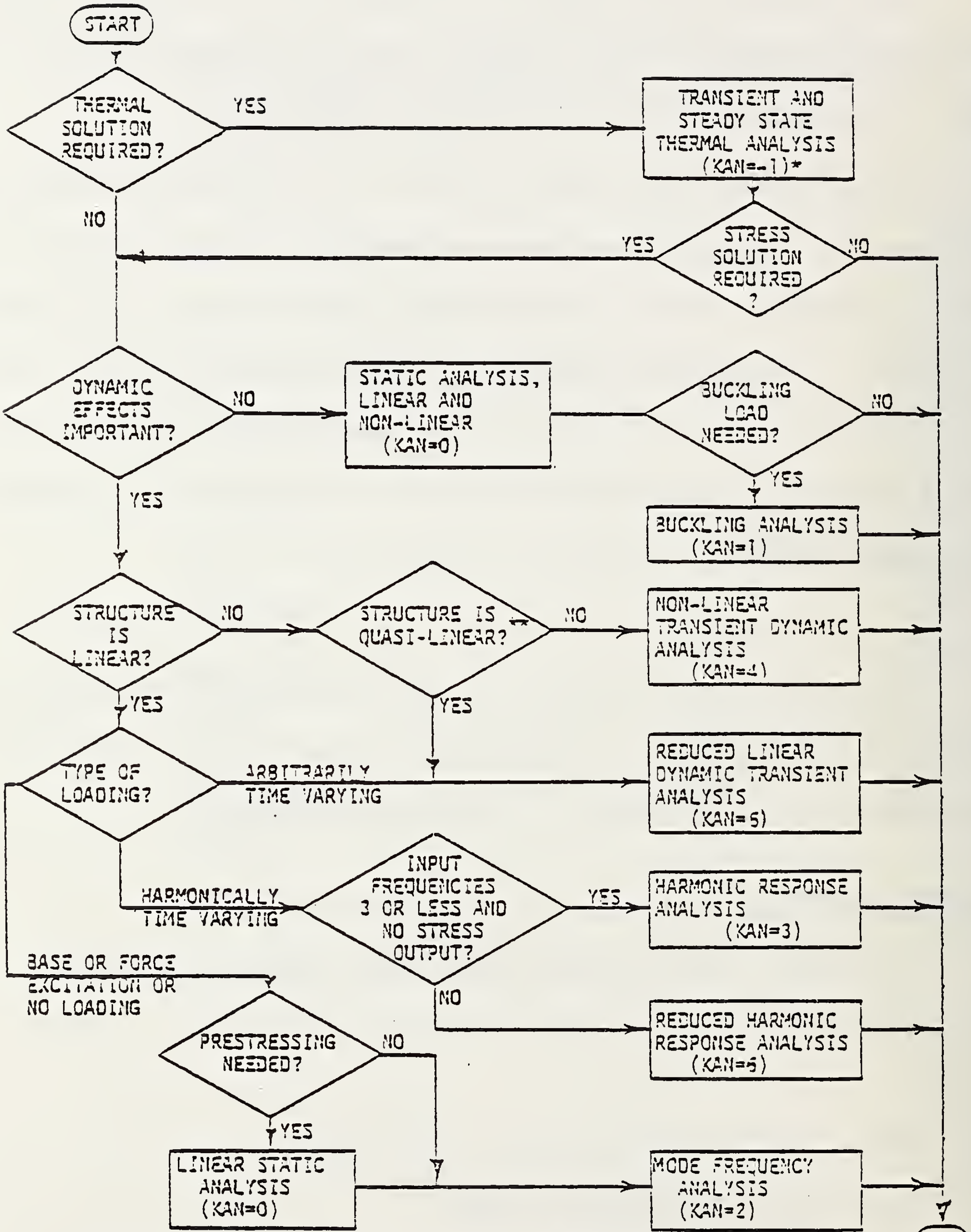
$$[K]\{u\} = \{F\} \qquad (B.1)$$

where: $[K]$ = total stiffness matrix = $\sum_{m=1}^{N} [K_e]$

$\{u\}$ = nodal displacement vector

N = number of elements

$[K_e]$ = element stiffness matrix (may include the element stress stiffness matrix)



* KAN is the key input on the C1 card to select the analysis type

** Quasi-Linear - the only non-linearities permitted are gaps.

Figure B1. Summary of ANSYS Analysis Types

$\{F\}$, the total force vector, is defined by:

$$\{F\} = \{F^n\} + \{F^a\} + \sum_{m=1}^{N} (\{F_e^t\} + \{F_e^{pr}\} + \{F_e^{pl}\} + \{F_e^{cr}\}$$
$$+ \{F_e^{sw}\} + \{F_e^{ld}\}) \qquad (8.2)$$

where:

$\{F^n\}$ = applied nodal load vector

$\{F^a\}$ = $[M]\{A_c\}$ = acceleration load vector

$[M]$ = total mass matrix $\sum_{m=1}^{N} [M_e]$

$[M_e]$ = element mass matrix

$\{A_c\}$ = nodal acceleration vector

$\{F_e^t\}$ = element thermal load vector

$\{F_e^{pr}\}$ = element pressure load vector

$\{F_e^{pl}\}$ = element plastic strain load vector

$\{F_e^c\}$ = element creep strain load vector

$\{F_e^s\}$ = element swelling strain load vector

$\{F_e^{ld}\}$ = element large displacement load vector

The same definitions used here apply to all other analysis procedures except heat transfer analysis.

If sufficient boundary conditions are specified on $\{u\}$ to guarantee a unique solution, equation 8.1 can be solved to obtain the nodal point displacements at each node in the structure. The simultaneous equations with all degrees of freedom (including those with specified displacements) are given in equation 8.3.

$$\begin{bmatrix} K & K_r \\ K_r^T & K_{rr} \end{bmatrix} \begin{Bmatrix} u \\ u_r \end{Bmatrix} = \begin{Bmatrix} F \\ F_r \end{Bmatrix} \tag{B.3}$$

The subscript r is associated with the reaction forces. Note that $\{u_r\}$ is known, but not necessarily equal to {0}. The top half of equation B.3 may be solved for {u}:

$$\{u\} = -[K]^{-1}[K_r]\{u_r\} + [K]^{-1}\{F\} \tag{B.4}$$

The reaction forces $\{F_r\}$ may then be computed from the bottom half of equation B.3 :

$$\{F_r\} = [K_r]^T\{u\} + [K_{rr}]\{u_r\} \tag{B.5}$$

These reaction forces should always be in equilibrium with the applied loads. The following circumstances could cause a disequilibrium, usually a moment disequilibrium:

1. The presence of stress stiffening
   Note that moment equilibrium is not preserved.
   This may be accounted for as as
   implicit updating of the coordinates.
2. The presence of four-noded shell elements where the four nodes do not lie in a flat plane.
3. The presence of nodal coupling or constraint equations. The user can write any form of relationship between the displacements, and these may induce fictitious forces or moments. Thus, the reaction force printout has been used to detect input errors.

1.8 HEAT TRANSFER ANALYSIS (KAN=-1)

Steady state and transient heat transfer problems may be solved by finite element techniques analogous to those used for structural analysis.

A. Steady State

The basic thermal equilibrium equation is:

$$[\overline{K}]\{T\} = \{Q\} \tag{B.6}$$

where: $[\overline{K}]$ = thermal conductivity matrix

$\{Q\}$ = heat flow vector

$\{T\}$ = vector of the nodal point temperatures

This equation is identical in form to equation B.1 for static analysis.

If the material properties and film coefficients are not temperature dependent, equation B.6 can be solved directly with one iteration. If the material properties (or film coefficients) are temperature dependent, they are evaluated at the temperature of the previous iteration. The procedure used is shown in Figure B2.

An optional convergence criterion is available with steady-state analysis. All nodes are monitored for the largest change in temperature. If this largest change is less than the criterion, then the solution is said to be converged. This criterion is input on the MD card (TCV) and defaults to 1.0 degree.

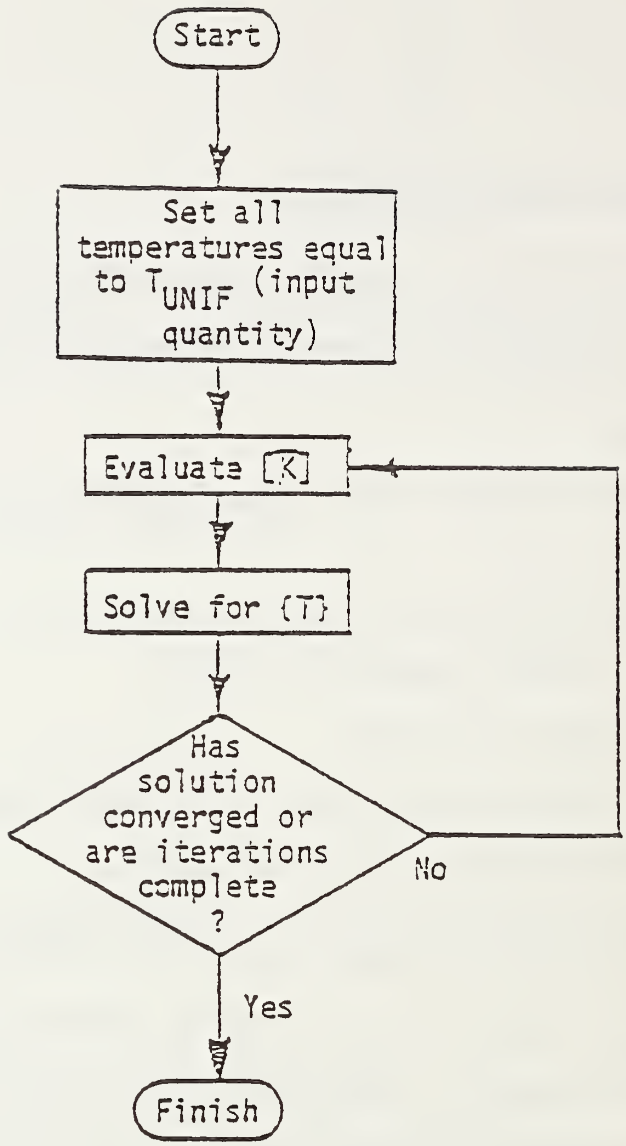


Figure B2. Flow chart for Steady State Heat Transfer Analysis with Temperature Dependent Material Properties.

B. Transient

The basic thermal diffusion equation is:

$$[\bar{C}]\{\dot{T}\} + [\bar{K}]\{T\} = \{Q\} \tag{B.7}$$

where $[\bar{C}]$ is the specific heat matrix.

The form of this equation is identical to the non-linear dynamic transient equation (KAN=4) except that the mass term is not present. For temperature dependent material properties (or film coefficients), the evaluation of the properties is made at a temperature extrapolated from the previously calculated temperatures.

The time-integration schemes are also the same as that of the non-linear transient dynamic analysis type (KAN=4) except that the options offered are one order lower, i.e., linear (KAY(2)=2) and quadratic (KAY(2)=0 or 1, the recommended usage). The linear (first order) equation is:

$$\left(\frac{1}{\Delta t_0}[\bar{C}] + [\bar{K}]\right)\{T_t\} = \{Q(t)\} + [\bar{C}]\{T_{t-1}\}\frac{1}{\Delta t_0} \qquad \text{(B.8)}$$

The quadratic (second order) equation is:

$$\left(\frac{2\Delta t_0 + \Delta t_1}{\Delta t_{01}}\frac{1}{\Delta t_0}[\bar{C}] + [\bar{K}]\right)\{T_t\} = \{Q(t)\} +$$

$$[\bar{C}]\left(\frac{\Delta t_{01}}{\Delta t_0 \Delta t_1}\{T_{t-1}\} - \frac{\Delta t_0}{\Delta t_{01}\Delta t_1}\{T_{t-2}\}\right) \qquad \text{(B.9)}$$

where:
t_0 = present time
t_1 = previous time
t_2 = time at second previous time
$\Delta t_0 = t_0 - t_1$
$\Delta t_1 = t_1 - t_2$
$\Delta t_{01} = t_0 - t_2$
T_t = temperature at this time step (to be calculated)
T_{t-1} = temperature at previous time step (known)
T_{t-2} = temperature at second-previous time step (known)

The starting procedure of the transient thermal analysis is as follows: If the first load step is run at time = 0. (TIME=0. on L card), a steady-state analysis is performed at that time. Alternatively, if the first load step is run at time > 0. (TIME > 0. on L card), all temperatures at time = 0. are set equal to TUNIF. The temperatures at time = t_1 (t_i means the solution at time i) are determined by the user selected time interpretation procedure (linear or quadratic, depending on the value of KAY(2)). If the quadratic integration is used, it is started by setting all temperatures at a previous time point (t_{-1}) to those values at time = 0.

It is not recommended that the time step size between adjacent iterations be changed by more than a factor of ten, unless $\frac{d^2T}{dt^2}$ is very small.

An option is available to increase the time step size automatically if the rate of change of temperature at all nodes is less than an input criterion. This optimization criterion is input on the MD card as TOV, which defaults to 5.0 degrees. Using the default value, this criterion may be expressed as:

$$\max \left| \frac{d^2T}{dt^2} \Delta t^2 \right| < 5.° \qquad (B.10)$$

Because a history has to be developed, the time step size may be increased only after the second iteration.

APPENDIX C

ELEMENT LIBRARY

Each element in the ANSYS program is discussed in this section. The assumptions required to generate the element matrices and load vector are given, including the assumed shape functions. Certain aspects are also discussed in the chapters on the nonlinear capabilities of ANSYS. Elements with nonlinear material properties (plasticity, creep, and/or swelling) have appropriate quantities saved at the integration points, except as noted.

In broad terms, all stress and thermal elements have their appropriate matrices and vectors derived using the procedures in the following two sections entitled:

Virtual Work Derivation for Stress Analysis Elements.

Virtual Work Derivation for Thermal Analysis Elements.

These derivations assume the use of an isoparametric element, as that element family is one of the simplest. On the other hand, a complete virtual work derivation is also given with elements STIF46, STIF53, and STIF61, which are not isoparametric elements.

Virtual Work Derivation for Stress Analysis Elements.

The principle of virtual work says that a virtual (very small) change of the internal strain energy must be offset by an identical change in external work due to the applied loads, or:

$$\delta U = \delta V \qquad \text{(C.1)}$$

where: δU = virtual strain energy (internal work)

δV = virtual (external) work

The virtual strain energy is:

$$\delta U = \int_{vol} \{\delta\varepsilon\}^T \{\sigma\}\, d(vol) \tag{C.2}$$

where: $\{\varepsilon\}$ = strain vector

$\{\sigma\}$ = stress vector

vol = volume of element

The stress is related to the strains by:

$$\{\sigma\} = [D](\{\varepsilon\} - \{\varepsilon_{th}\}) \tag{C.3}$$

where: $[D]$ = material property (constitutive) matrix

$\{\varepsilon_{th}\}$ = thermal strain vector

Equation C.3 may also be written as:

$$\{\varepsilon\} = \{\varepsilon_{th}\} + [D]^{-1}\{\sigma\} \tag{C.4}$$

For the case of three-dimensional solid elements, equation (2.0.4) may be expanded to:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{xy} \\ \gamma_{xz} \end{Bmatrix} = \begin{Bmatrix} \alpha_x \Delta T \\ \alpha_y \Delta T \\ \alpha_z \Delta T \\ 0 \\ 0 \\ 0 \end{Bmatrix} + \begin{bmatrix} \frac{1}{E_x} & -\frac{\nu_{xy}}{E_y} & -\frac{\nu_{xz}}{E_z} & 0 & 0 & 0 \\ -\frac{\nu_{yx}}{E_x} & \frac{1}{E_y} & -\frac{\nu_{yz}}{E_z} & 0 & 0 & 0 \\ -\frac{\nu_{zx}}{E_x} & -\frac{\nu_{zy}}{E_y} & \frac{1}{E_z} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G_{xy}} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G_{yz}} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G_{xz}} \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{xz} \end{Bmatrix} \tag{C.5}$$

where ΔT = the difference at the point in question between its own temperature and the reference (strain free) temperature (TREF).

The [D] matrix is presumed to be symmetric, so that:

$$\frac{\nu_{yx}}{E_x} = \frac{\nu_{xy}}{E_y} \quad (C.6)$$

$$\frac{\nu_{zx}}{E_x} = \frac{\nu_{xz}}{E_z} \quad (C.7)$$

$$\frac{\nu_{zy}}{E_y} = \frac{\nu_{yz}}{E_z} \quad (C.8)$$

Thus, in terms of ANSYS input variables:

$$[D]^{-1} = \begin{bmatrix} \frac{1}{EX} & -\frac{NUXY}{EY} & -\frac{NUXZ}{EZ} & 0 & 0 & 0 \\ -\frac{NUXY}{EY} & \frac{1}{EY} & -\frac{NUYZ}{EZ} & 0 & 0 & 0 \\ -\frac{NUXZ}{EZ} & -\frac{NUYZ}{EZ} & \frac{1}{EZ} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{GXY} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{GYZ} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{GXZ} \end{bmatrix} \quad (C.9)$$

and:

$$\{\varepsilon_{th}\} = \begin{Bmatrix} ALPX(\Delta T) \\ ALPY(\Delta T) \\ ALPZ(\Delta T) \\ 0 \\ 0 \\ 0 \end{Bmatrix} \quad (C.10)$$

If GXY, GYZ, and GXZ are not input, they are computed as:

$$GXY = \frac{EX\ EY}{EX + EY + 2\ NUXY\ EX} \tag{C.11}$$

$$GYZ = \frac{EY\ EZ}{EY + EZ + 2\ NUYZ\ EY} \tag{C.12}$$

$$GXZ = \frac{EX\ EZ}{EX + EZ + 2\ NUXZ\ EX} \tag{C.13}$$

A further comment on the [D] matrix: It must be positive definite. This condition is always met if the material is isotropic or NUXY, NUYZ, and NUXZ are all zero. But, for example, if EY is less than or equal to $EX(NUXY)^2$, the material is not positive definite.

$\{\varepsilon_{th}\}$ may also be considered to include plastic, creep, and swelling effects, where applicable.

Equations C.2 and C.3 are combined to give:

$$\delta U = \int_{vol} \left(\{\delta\varepsilon\}^T[D]\{\varepsilon\} - \{\delta\varepsilon\}^T[D]\{\varepsilon_{th}\}\right) d(vol) \tag{C.14}$$

The strains may be related to the nodal displacements by:

$$\{\varepsilon\} = [B]\{u\} \tag{C.15}$$

where: [B] = strain-displacement matrix

{u} = nodal displacements

Combining C.15 with C.14 , and noting that {u} does not vary over the volume:

$$\delta U = \{u\}^T \int_{vol} [B]^T[D][B]\{u\}\ d(vol) - \{u\}^T \int_{vol} [B]^T[D]\{\varepsilon_{th}\}\ d(vol) \tag{C.16}$$

Next, the virtual work will be considered. The inertial effects will be studied first:

$$\delta V = \int_{vol} \{\delta w\}^T\{(F^a/vol)\}\ d(vol) \tag{C.17}$$

where: $\{w\}$ = vector of displacements of a general point

$\{F^a\}$ = acceleration (D'Alembert) force vector

According to Newton's second law.

$$\frac{-\{F^a\}}{vol} = \rho \frac{\partial^2}{\partial t} \{w\} \tag{C.18}$$

where: ρ = density

t = time

The point-wise displacements are related to the nodal displacements by:

$$w = [A]\ u \tag{C.19}$$

where $[A]$ = matrix of shape functions. Combining equations C.18 and C.19:

$$\delta V = \{\delta u\}^T \rho \int_{vol} [A]^T[A]\ \ d(vol)\ \frac{\partial^2}{\partial t^2}\{u\} \tag{C.20}$$

The pressure force vector formulation starts with:

$$\delta V = \int_{area} \{\delta w\}^T\{p\}\ d(area) \tag{C.21}$$

where $\{p\}$ = the applied pressure vector (normally contains only one non-zero component).

Combining equations

$$\delta V = \{\delta u\}^T \int_{area} [A]^T \{p\} \ d(area) \tag{C.22}$$

Finally, equations

combined to give:

$$\begin{aligned}
&\{\delta u\}^T \int_{vol} [B]^T [D][B]\{u\} \ d(vol) \\
&- \{\delta u\}^T \int_{vol} [B]^T [D]\{\varepsilon_{th}\} \ d(vol) \\
&= \{\delta u\}^T \rho \int_{vol} [A]^T [A] \ d(vol) \frac{\partial^2}{\partial t} \{u\} \\
&+ \{\delta u^T\} p \int_{area} [A]^T \ d(area)
\end{aligned} \tag{C.23}$$

Noting that the $\{\delta u\}^T$ is common in all of the above terms, and that its terms are independent of each other, it may be cancelled out. Thus, equation C.23 reduces to:

$$[K_e]\{u\} - \{F_e^t\} = [M_e]\{\ddot{u}\} + \{F_e^{pr}\} \tag{C.24}$$

where:

$[K_e] = \int_{vol} [B]^T [D][B] \ d(vol)$ = element stiffness matrix

$\{F_e^t\} = \int_{vol} [B]^T [D]\{\varepsilon_{th}\} \ d(vol)$ = element thermal load vector

$[M_e] = \rho \int_{vol} [A]^T [A] \ d(vol)$ = element (consistent) mass matrix

$\{\ddot{u}\} = \frac{\partial^2}{\partial t} \{u\}$ = acceleration vector (such as gravity effects)

$\{F_e^{pr}\} = \int_{area} [A]^T \{p\} \ d(area)$ = element pressure vector

Those elements which use a lumped sum mass matrix rather than a consistent mass matrix are noted with the individual element description.

The element stresses are computed by combining equations C.3 and C.15 to get:

$$\{\sigma\} = [D]([B]\{u\} - \{\varepsilon_{th}\}) \tag{C.25}$$

Note that [B] is the strain-displacement matrix that must be specialized for each stress calculation point (centroid, integration point, node point, etc.)

<u>Virtual Work Derivation for Thermal Analysis Elements</u>

As before, the basic expression of virtual work is:

$$\delta U = \delta V \tag{C.26}$$

where: δU = virtual internal work

δV = virtual external work

In thermal terms, the virtual work within one element is:

$$\delta U = \int_{vol} \{\delta S\}^T \{Q_v\} \; d(vol) \tag{C.27}$$

where:

$$\{S\} = \begin{Bmatrix} \frac{\partial T(x,y,z)}{\partial x} \\ \frac{\partial T(x,y,z)}{\partial y} \\ \frac{\partial T(x,y,z)}{\partial z} \end{Bmatrix} = \text{vector of temperature}$$

$$\{Q_v\} = \begin{Bmatrix} Q_x \\ Q_y \\ Q_z \end{Bmatrix} = \text{vector of heat flows}$$

$T(x,y,z)$ = temperature at point x,y,z

Q_x = heat flow in the x-direction per unit area

Q_y = heat flow in the y-direction per unit area

Q_z = heat flow in the z-direction per unit area

The heat flows are related to the temperature gradients by:

$$\{Q_v\} = [D]\{S\} \tag{C.28}$$

where:

$$[D] = \begin{bmatrix} KXX & 0 & 0 \\ 0 & KYY & 0 \\ 0 & 0 & KZZ \end{bmatrix}$$

The temperature distribution within an element is based on the assumed temperature shapes:

$$T(x,y,z) = \{N\}^T\{T_e\} \tag{C.29}$$

where: $\{N\}$ = vector of shape functions

$\{T_e\}$ = nodal temperature vector

Then, $\{S\}$ is related to $\{T_e\}$ using the definition of $\{S\}$ and equation C.29 to give:

$$\{S\} = [B]\{T_e\} \tag{C.30}$$

where

$$[B] = \begin{bmatrix} \{\frac{\partial N}{\partial x}\}^T \\ \{\frac{\partial N}{\partial y}\}^T \\ \{\frac{\partial N}{\partial x}\}^T \end{bmatrix}$$

Combining equations C.27, C.28, and C.30 and realizing that --

the nodal temperature vector does not change over the volume of the element,

$$\delta U = \{\delta T_e\}^T \int_{vol} [B]^T[D][B]\ d(vol)\ \{T_e\} \tag{C.31}$$

Next, consider the virtual internal work associated with convection surfaces:

$$\delta U = \int_{area} \delta \Delta T\ Q_n\ d(area) \tag{C.32}$$

where n = ~~the~~ direction normal to the surface. ΔT is defined by:

$$\Delta T = T(x,y,z)\big|_s - T_B \tag{C.33}$$

where: $T(x,y,z)\big|_s$ = the temperature function evaluated at the convection surface

T_B = temperature of the coolant (bulk temperature)

Note that T_B is a constant so that

$$\delta \Delta T = \delta T(x,y,z)\big|_s \tag{C.34}$$

The heat flow over the unit area is defined by:

$$Q_n = h_f\ \Delta T \tag{C.35}$$

where h_f = film coefficient for heat transfer of the surface. Combining equations C.29, C.32, C.33, C.34, and C.35, and noting that $\{T_e\}$ does not vary over the surface, and that T_B and h_f are assumed not to:

$$\delta U = \{\delta T_e\}^T h_f \int_{area} \{N|_s\}^T \; d(area)\{T_e\}$$

$$-\{\delta T_e\}^T h_f T_B \int_{area} \{N|_s\} \; d(area) \qquad (C.36)$$

where $\{N|_s\}$ are the shape functions evaluated at the convection surface.

The internal heat generation rate effect is included by considering:

$$\delta V = \int_{vol} \delta T(x,y,z) \; \dddot{q} \; d(vol) \qquad (C.37)$$

where $\dddot{q}$ = the heat generation rate per unit volume. Combining equations C.29 and C.37 and realizing that the nodal temperatures vector $\{T_e\}$ does not change over the volume of the element, and assuming that $\dddot{q}$ does not change over the volume of the element,

$$\delta U = \{\delta T_e\}^T \; \dddot{q} \int_{vol} \{N\} \; d(vol) \qquad (C.38)$$

The virtual internal work associated with a change of stored energy is:

$$\delta U = \int_{vol} \delta T(x,y,z)\gamma \; d(vol) \qquad (C.39)$$

where: $\gamma = \rho C_p \dfrac{\partial T(x,y,z)}{\partial t}$ = total heat change per unit volume per unit time

ρ = density

C_p = specific heat

t = time

Combining equations C.29 and C.39 and noting that $\{T_e\}$ does not vary over the element, and assuming that ρ and C_p do not vary over the element,

$$\delta U = \{\delta T_e\}^T \rho C_p \int_{vol} \{N\}\{N\}^T \, d(vol) \, \{\dot{T}_e\} \tag{C.40}$$

where $\{\dot{T}_e\} = \frac{\partial}{\partial t} \{T_e\}$

The effect of the nodal heat flows may be considered by,

$$\delta V = \{\delta T_e\}^T \{Q_e\} \tag{C.41}$$

where $\{Q_e\}$ is the nodal heat flow vector.

Combining equations C.26, C.31, C.36, C.38, C.40, and C.41, and noting that since $\{\delta T_e\}^T$ is an arbitrary set of virtual temperature changes which may be cancelled out,

$$[K_e] + [K_e^c] \;\; \{T_e\} + [C_e]\{\dot{T}_e\} = \{Q_e^c\} + \{Q_e^g\} + \{Q_e\} \tag{C.42}$$

where:

$$[K_e] = \int_{vol} [B]^T[D][B] \, d(vol)$$

$$[K_e^c] = h_f \int_{area} \{N|_s\}\{N|_s\}^T \, d(area)$$

= total element conductivity matrix

$$[C_e] = \rho C_p \int_{vol} \{N\}\{N\}^T \, d(vol)$$

= specific heat (thermal damping) matrix

$$\{Q_e^c\} = h_f \, T_B \int_{area} \{N|_s\} \, d(area)$$

$$\{Q_e^g\} = \ddot{q} \int_{vol} \{N\} \, d(vol)$$

= total element heat flow vector

This is the final temperature heat flow equilibrium equation. The above definitions are used to develop the element matrices and vectors.

STIF12 - TWO-DIMENSIONAL INTERFACE ELEMENT

The displacement functions for the interface element can be separated into the normal and tangential directions because they are basically independent.

In the normal direction, when the normal force (F_n) is negative, the interface remains in contact and responds as a linear spring. As the normal force becomes positive, contact is broken and no force is transmitted; unless KEYSUB(1)=1, in which case a small force is supplied to prevent a portion of the structure from being isolated.

In the tangential direction, for $F_n < 0$ and the absolute value of the tangential force (F_s) less than or equal to ($\mu|F_n|$), the interface does not slide and responds as a linear spring in the tangential direction. However, for $F_n < 0$ and $F_s > \mu|F_n|$, sliding occurs. Note that F_s is a variable and if contact is broken, the tangential function degenerates to a zero slope straight line through the origin (or of slope $K/10^6$, if KEYSUB(1)=1) indicating that no (or little) tangential force is required to produce sliding. These may be related to each other by $\mu|F_n| = K(u_{s_J} - u_{s_I} - u_{slide})$ where u_{slide} is the distance of sliding. Figure C2 shows the force-deflection relationships for this element.

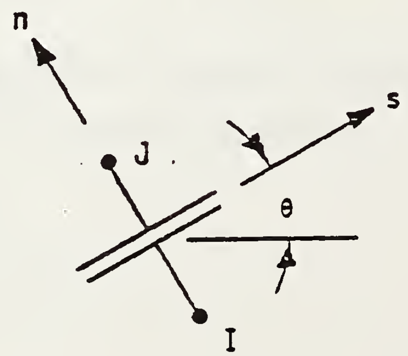


Figure C1.

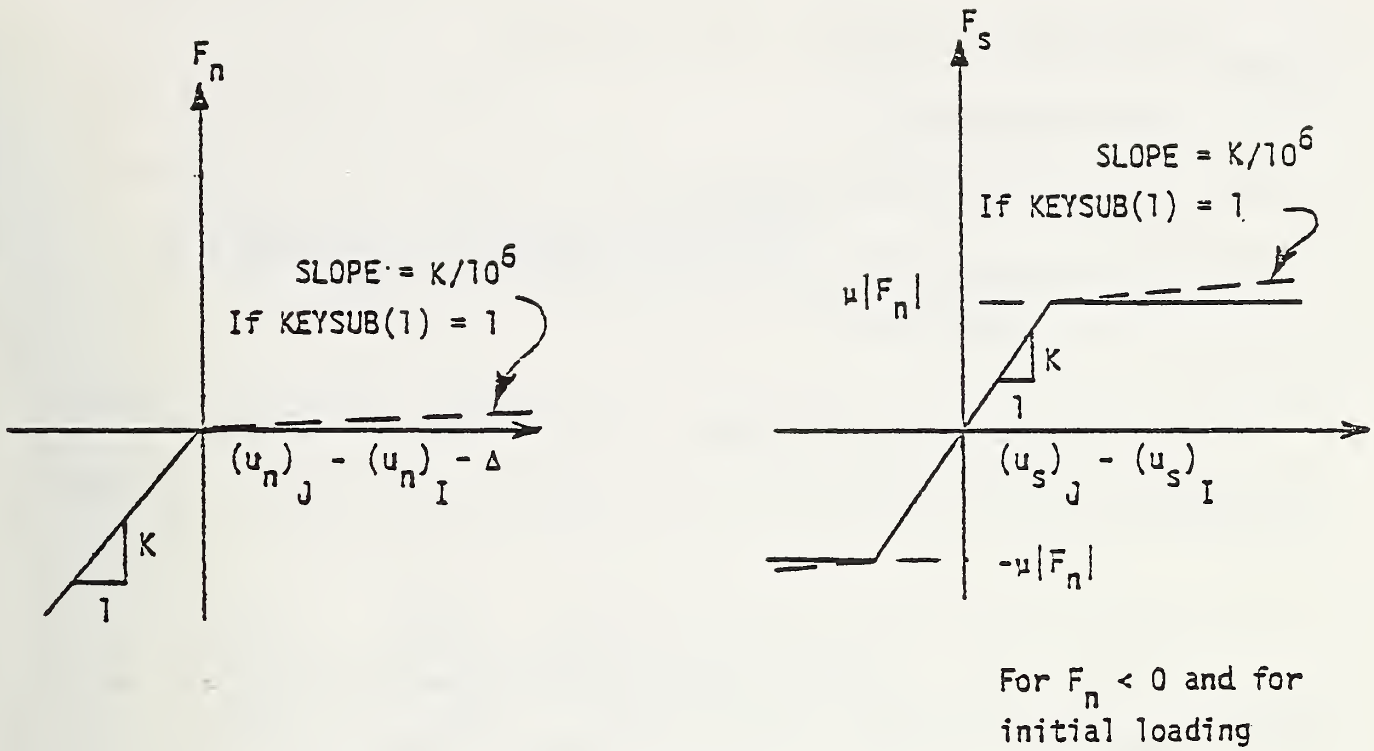


Figure C2. STIF12 Force-Deflection Relations

STIF12 may have one of three conditions: in contact and not sliding, in contact and sliding, or open. The following matrices are derived with the assumption that θ (theta) is input as 0.0.

1. In contact and not sliding - The resulting equilibrium equation is:

$$\underbrace{\begin{bmatrix} K & 0 & -K & 0 \\ 0 & K & 0 & -K \\ -K & 0 & K & 0 \\ 0 & -K & 0 & K \end{bmatrix}}_{\text{Stiffness matrix}} \underbrace{\begin{Bmatrix} u_{s,I} \\ u_{n,I} \\ u_{s,J} \\ u_{n,J} \end{Bmatrix}}_{\text{Displacement Vector}} = \underbrace{\begin{Bmatrix} F^n_{sI} \\ F^n_{nI} \\ F^n_{sJ} \\ F^n_{nJ} \end{Bmatrix}}_{\text{Applied Nodal Force Vector}} + \underbrace{\begin{Bmatrix} -Ku_o \\ -K\Delta \\ Ku_o \\ K\Delta \end{Bmatrix}}_{\text{Element Load Vector}} \quad (C.43)$$

where: K = input stiffness

Δ = interference

F_n = normal force across gap

u_o = distance that nodes I and J have slid with respect to each other

2. In contact and sliding - In this case, the element equilibrium equation is:

$$\underbrace{\begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & K & 0 & -K \\ 0 & 0 & 0 & 0 \\ 0 & -K & 0 & K \end{bmatrix}}_{\text{Stiffness Matrix}} \underbrace{\begin{Bmatrix} u_{s,I} \\ u_{n,I} \\ u_{s,J} \\ u_{n,J} \end{Bmatrix}}_{\text{Displacement Vector}} = \underbrace{\begin{Bmatrix} F^n_{sI} \\ F^n_{nI} \\ F^n_{sJ} \\ F^n_{nJ} \end{Bmatrix}}_{\text{Applied Nodal Force Vector}} + \underbrace{\begin{Bmatrix} \mu F_n \\ -K\Delta \\ -\mu F_n \\ K\Delta \end{Bmatrix}}_{\text{Element Load Vector}} \quad (C.44)$$

where μ = coefficient of friction.

3. Open - When there is no contact between nodes I and J, the stiffness matrix and load vector are null matrices.

The stress pass of STIF12 always uses the latest possible information concerning gap status. Therefore, for non-converged iterations, it may not agree with the reaction forces which are based on the previously calculated stiffness matrix and load vector.

STIF32 - 2-D CONDUCTING BAR ELEMENT

The temperature function is a linear polynomial of the form:

$$T(x) = C_1 + C_2x \tag{C.45}$$

where the element x-axis extends along the element axis.

STIF42 - 2-D ISOPARAMETRIC SOLID ELEMENT

The displacement shape functions are repeated here for convenience. A local coordinate system is developed as shown in Figure C3:

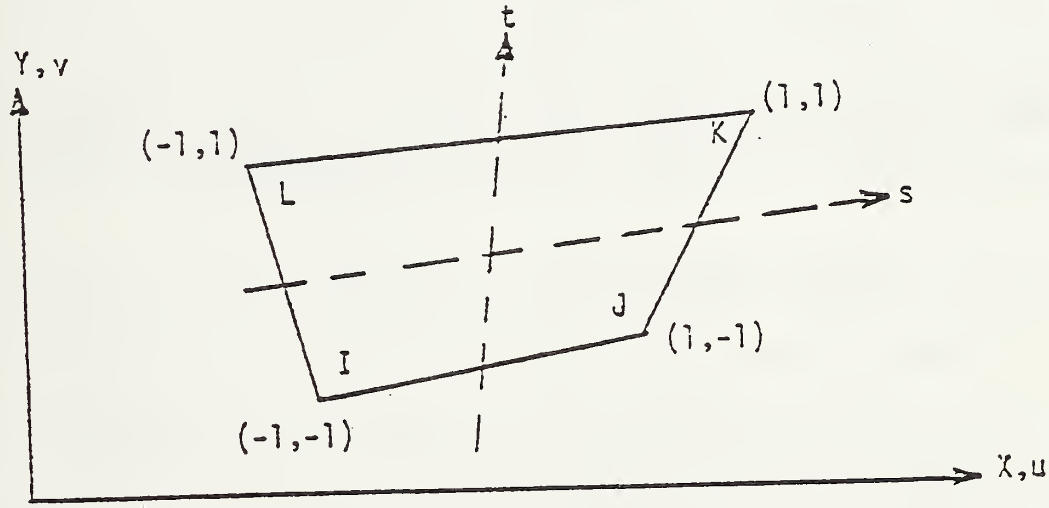


Figure C3. Local Coordinate System

It is seen that s and t vary between -1. and +1. The basic isoparametric shapes yield the following set of shape functions:

$$u_b(s,t) = \frac{1}{4}(1 - s)(1 - t)u_I + \frac{1}{4}(1 + s)(1 - t)u_J$$

$$+ \frac{1}{4}(1 + s)(1 + t)u_K + \frac{1}{4}(1 - s)(1 + t)u_L \qquad (C.46)$$

$$v_b(s,t) = \frac{1}{4}(1 - s)(1 - t)v_I + \frac{1}{4}(1 + s)(1 - t)v_J$$

$$+ \frac{1}{4}(1 + s)(1 + t)v_K + \frac{1}{4}(1 - s)(1 + t)v_L \qquad (C.47)$$

Note that these shapes do not permit the edges to bend.

The extra (and optional) shapes are defined as:

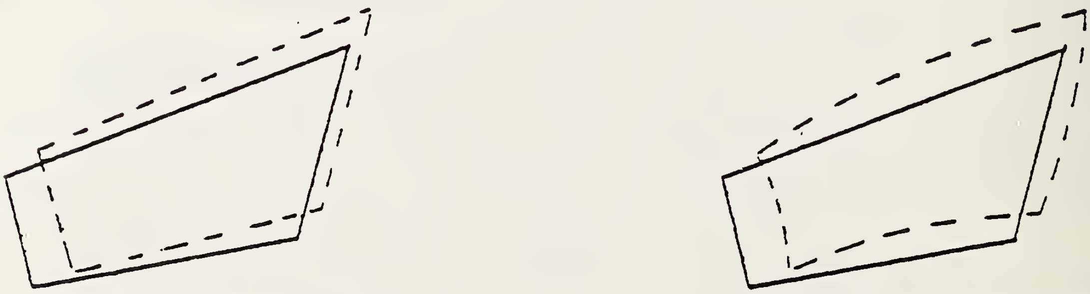
$$u_e(s,t) = (1 - s^2)c_1 + (1 - t^2)c_2 \qquad (C.48)$$

$$v_e(s,t) = (1 - s^2)c_3 + (1 - t^2)c_4 \qquad (C.49)$$

Their effect may be seen in Figure C4. c_1 through c_4 may be referred to as nodeless variables. The total displacements are then:

$$u = u_b + u_e \qquad (C.50)$$

$$v = v_b + v_e \qquad (C.51)$$



Without extra shapes

With extra shapes

Figure C4. Effect of Extra Shapes

These displacement shapes are used to generate a 12 by 12 stiffness matrix. This matrix is then condensed to an 8 by 8 matrix, because there are only 8 degrees of freedom to connect to the rest of the structure. The condensation is analogous to that associated with superelement generation. The load vector is also generated with 12 terms and is then condensed to 8. The mass matrix is consistent and is generated as an 8 by 8.

A 3 by 3 lattice of integration points is used with the numerical (Gaussian) integration procedure.

Note that the extra shapes permit a parabolic deformation along an element edge. Normally this is helpful in modeling a structure, but occasionally it may cause a problem because of the incompatibility at the adjoining edges of two different elements, i.e., a gap opens up or the material "doubles up". The usage of the extra shapes is discussed in greater detail in the User's Manual. The extra shapes are automatically deleted if nodes K and L are the same (i.e. a triangle). This case then gives the same results as a constant strain triangle.

The centroidal, integration point, and node point stresses are computed by the procedure described at the beginning of this chapter (Equation C.25)

Surface stresses may be requested for elastic isotropic materials. Even though the development given below includes some orthotropic effects, it is only valid for a few special cases of orthotropic materials. The surface stresses for plane stress applications are calculated by:

1. Computing the strain parallel to the free surface:

$$\varepsilon_1 = \frac{u_I^t - u_J^t}{L} - \alpha_x \Delta T \qquad \text{(C.52)}$$

Where: u^t = displacement parallel to the free surface

L = distance between the two surface nodes

α_x = coefficient of thermal expansion (ALPX)

ΔT = difference between average surface temperature and the reference temperature.

2. Setting the stress normal to the surface (σ_2) to the applied pressure.

3. Setting the stress in the z direction (σ_3) to 0.

4. Solving for the remaining three quantities of interest (ε_2, ε_3, σ_1) by use of the material property relationships. Specifically:

$$\sigma_1 = \varepsilon_1 E_a + \nu_{xy} \sigma_2 \tag{C.53}$$

$$\varepsilon_3 = -\nu_{xy} (\sigma_1 + \sigma_2)/E_a \tag{C.54}$$

$$\varepsilon_2 = (\sigma_2 - \nu_{xy} \varepsilon_1)/E_a \tag{C.55}$$

where: $E_a = (E_x + E_y)/2.$

E_x = Young's modulus in the x-direction (EX)

E_y = Young's modulus in the y-direction (EY)

ν_{xy} = Poisson's ratio (NUXY)

For the axisymmetric option, steps 1 and 2 above are the same. Continuing,

3. Computing the hoop strain (ε_3):

$$\varepsilon_3 = \frac{u_I^r + u_J^r}{2R} \div \frac{u_e^r}{R} - \alpha_z \Delta T \tag{C.56}$$

where: u_I^r = radial displacement of node I

u_J^r = radial displacement of node J

u_e^r = radial displacement of the midpoint of side I-J due to the applicable extra shape function

R = radius of the midpoint of side I-J

α_z = coefficient of thermal expansion in hoop direction (ALPZ)

4. Solving for the remaining three quantities of interest (ε_2, σ_1, σ_3) by use of the material property relationships. Specifically,

$$\sigma_3 = \frac{\varepsilon_3 E_z + (\nu_{yz} + \nu_{xz}\nu_{xy})\sigma_2 + \varepsilon_1 \nu_{xz} E_a}{1. - \nu_{xz} \frac{E_a}{E_z}} \tag{C.57}$$

$$\sigma_1 = \varepsilon_1 E_a + \sigma_2 \nu_{xy} + \sigma_3 \nu_{xz} \frac{E_a}{E_z} \tag{C.58}$$

$$\varepsilon_2 = \frac{\sigma_2}{E_a} - \nu_{xy} \frac{\sigma_1}{E_a} - \nu_{yz} \frac{\sigma_3}{E_z} \tag{C.59}$$

where: E_z = input quantity (EZ)

ν_{xz} = input quantity (NUXZ)

ν_{yz} = input quantity (NUYZ)

Plane strain analysis is the same as axisymmetric analysis, except that step 3 is modified so that simply,

$$\varepsilon_3 = -\alpha_z \, \Delta T \tag{C.60}$$

STIF45 - 3-D ISOPARAMETRIC SOLID ELEMENT

The element formulation includes incompatible displacement modes. A complete description of this technique is the three-dimensional extension of STIF42. Either a 3x3x3 or a 2x2x2 lattice of integration points is available for use with the numerical (Gaussian) integration procedure. For nonlinear material properties (plasticity, creep, or swelling), a 2x2x2 lattice is automatically used.

The principal stresses are calculated from the cubic equation:

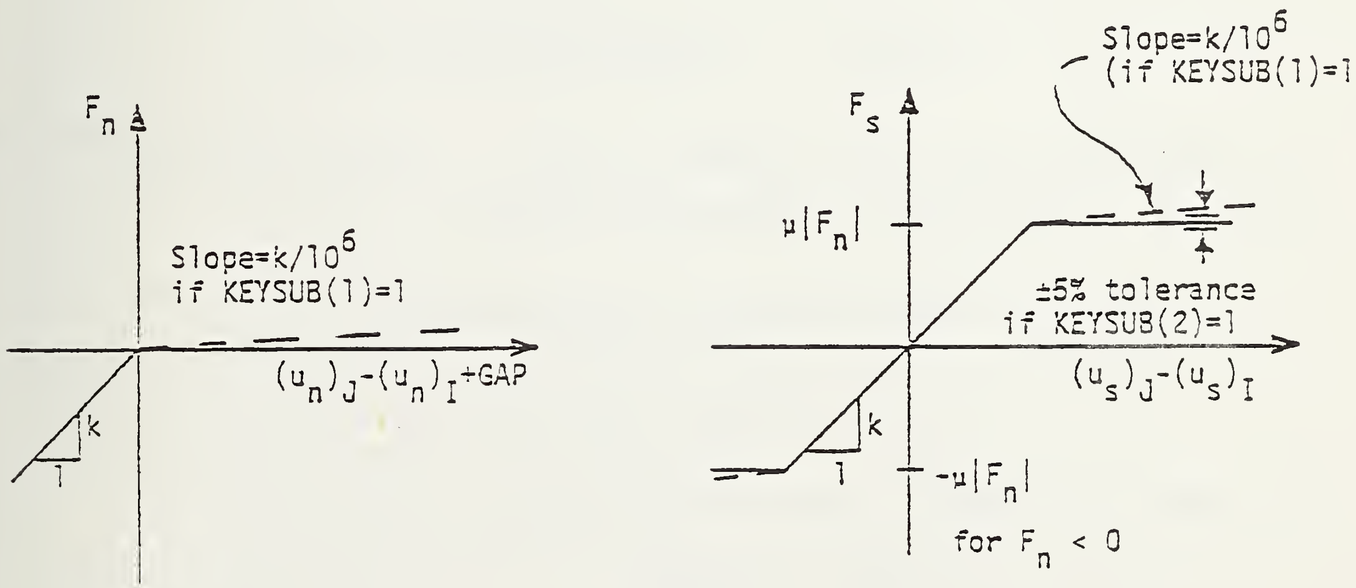
$$\begin{vmatrix} \sigma_x - \sigma & \tau_{xy} & \tau_{xz} \\ \tau_{xy} & \sigma_y - \sigma & \tau_{yz} \\ \tau_{xz} & \tau_{yz} & \sigma_2 - \sigma \end{vmatrix} = 0 \qquad \text{(C.61)}$$

The three computed values of σ are the three principal stresses.

STIF52 - 3-D INTERFACE ELEMENT

The load-deflection relationships for the interface element can be separated into the normal and tangential directions since they are basically independent. In the normal (element x) direction, when the normal force (F_n) is negative, the interface remains in contact and responds as a linear spring. As the normal force becomes positive, contact is broken and no force is transmitted (unless KEYSUB(1)=1, then a small force is supplied to prevent a portion of the structure from being isolated).

In the tangential directions, for $F_n < 0$ and the absolute value of the tangential force (F_s) less than or equal to ($\mu|F_n|$), the interface does not slide and responds as a linear spring in the tangential direction. However, for $F_n < 0$ and $F_s > \mu|F_n|$, sliding occurs. Note that F_n is a variable and if contact is broken, the tangential function degenerates to a zero slope straight line through the origin (or of slope $k/10^6$, if KEYSUB(1)=1) indicating that no (or little) tangential force is required to produce sliding. Figure C5 shows the force-deflection functions for this element.



95%.

Figure C5. Force-Deflection Relationships

STIF55 - 2-D ISOPARAMETRIC HEAT CONDUCTING SOLID ELEMENT

The temperature functions used in STIF55 are a scalar form of those developed for displacements in STIF42.

First, an element coordinate system is developed as shown in Figure C6.

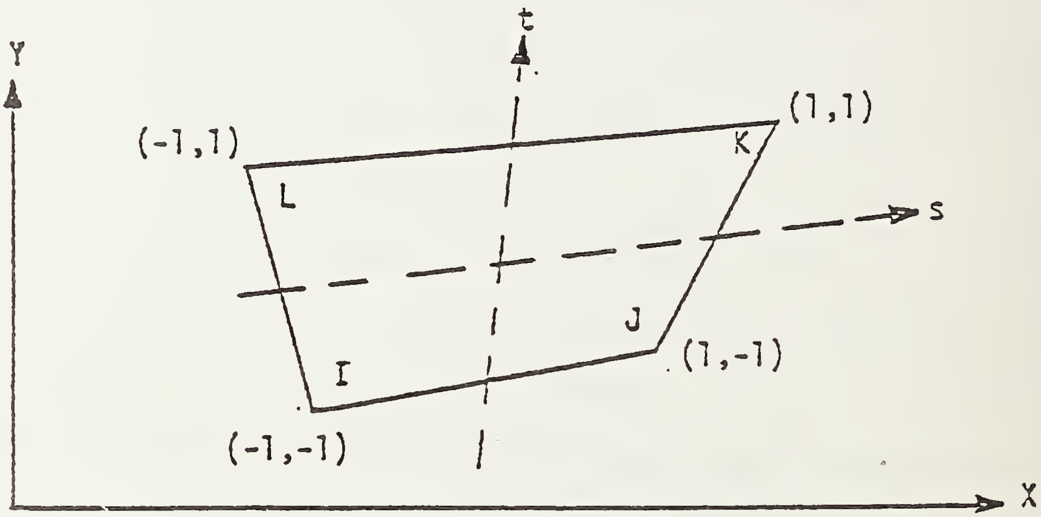


Figure C6. Element Coordinate System

It is seen that s and t vary between -1. and +1. The basic isoparametric shapes yield the following set of temperature functions:

$$T_b(s,t) = 1/4(1-s)(1-t)T_I + 1/4(1+s)(1-t)T_J + 1/4(1+s)(1+t)T_K + 1/4(1-s)(1+t)T_L \qquad (C.62)$$

The extra (and optional) shapes are defined as:

$$T_e(s,t) = (1-s^2)c_1 + (1-t^2)c_2 \qquad (C.63)$$

c_1 through c_2 may be referred to as nodeless variables. The total temperatures are then:

$$T = T_b + T_e \tag{C.64}$$

These displacement shapes are used to generate a 6 by 6 stiffness matrix. A 3 by 3 lattice of integration points is used with the numerical (Gaussian) integration procedure. This matrix is then condensed down to a 4 by 4 matrix, because there are only four nodes to connect to the rest of the structure. The condensation is analogous to that associated with superelement generation equation. The load vector is generated also with six terms and is then condensed down to four. The damping (specific heat) matrix is consistent and is also reduced down from a 6 by 6 to a 4 by 4.

APPENDIX D

COMPUTER DEFINITIONS AND COMMUNICATION LINKS

DEFINITIONS (UNIVAC 1100 COMPUTER)

The hardware organization of the 1110 (1100/40) and 1100/80 Systems differ from that of the 1106, 1108, 1100/10, and 1100/20 Systems. In some instances, different terms have been adopted for functionally similar components. In such cases, to avoid confusion and improve readability, the 1108 term has, as a general rule, been used throughout this document synonymously with the corresponding 1110 term, except where specific comments are made to the contrary. 1108-type will be used to include the 1106, 1100/10, 1100/20, and 1108. 1110-type will be used to include the 1110 and the 1100/40.

The principal corresponding terms are:

1108	1110	1100/80
CPU	CAU(plus IOAU)	CPU(plus IOU)
ACU	SPU	STU
Control Registers (ICR)	CRS	GRS

Introductory Definitions

bit — Binary digit. The fundamental unit of storage having the value 0 or 1. Bits are grouped in bytes and words to form the functional manipulative units of storage devices.

byte — A group pf adjacent bits usually operated upon as a unit; can be 6, 9, 12, or 18 bits.

buffer storage — On 1100/80 a high speed storage interface (4k to 16k).

Executive or EXEC — The 1100 Series Executive System. A program that controls the execution of other routines. The Executive is the principal interface between the user and the system as a whole. It protects against undesired interaction of users with each other or the operating system.

hardware — Physical equipment, in the form of mechanical, magnetic, electrical, or electronic devices, as opposed to software.

I/O — Input/Output. The process of transferring information between the central processor and peripheral devices. I/O devices include: magnetic tapes, magnetic disks, magnetic drums, CRTs, card readers, printers, and punches.

mnemonic — Word or term devised so as to aid the human memory. Includes acronyms, such as TTY (telety pewriter) and error mnemonics, such as PWRLOS (powerloss).

operating system	The 1100 Series Operating System. The entire set of system software available for the 1100 Series which is either a part of or operates under the Executive system. This includes the Executive system proper, compilers, utility programs, subroutine libraries, and so forth.
software	A set of computer programs including the operating system and user programs, as opposed to hardware.
system	The total 1100 Series hardware/software complex comprising an integrated information processing installation.
user	An individual or organization that consumes services provided by the system.
word	A sequence of bits or characters treated as a unit and capable of being stored in a single main storage location (a word is represented by 36 bits for the 1100 Series Systems).

Hardware Definitions

ACU	Availability Control Unit. A device used i n 1108-type Systems to isolate particular system components for maintenance or system partitioning. The ACU, in certain operating modes, can initiate autorecovery.
application	The total installation hardware configuration or a subset resulting from partitioning that configuration via hardware or software.
auxiliary storage	Supplemental storage, as opposed to main storage. It is not directly addressable by CPU(s) and is accessible only through an I/O interface. It includes magnetic tapes, flying-head magnetic drum, FASTRAND drum, disk, or unitized channel storage.
breakpoint	A feature whereby the CPU can be stopped or interrupted when a particular main storage address is read, written, or executed as an instruction.
CAU	Command/Arithmetic Unit. It is the 1110, 1100/40 equivalent of the instruction processing portion of a CPU. A CAU does not contain an input/output section, as does a CPU. Therefore, it must operate in conjunction with an IOAU in order to access peripheral subsystems.
central group	The CPUs, CAUs, IOAUs, IOUs, ACUs, SPUs, STUs, and consoles.
central site	The central group, main storage, and attached onsite peripheral equipment in a particular application.
channel	A data path for transfer of information between the central group and I/O devices.

CPU — The Central Processor Unit component on 1108 and 1100/80 Systems which executes all control and arithmetic functions. The 1108 System CPU contains an input/output section for access to peripheral devices.

CRT — Cathode-ray tube display. A television-like device that presents data in visual form.

dual channel — Two separate data paths for transfer of information between the central group and a subsystem. The sub system control unit must have dual channel capability.

IOAU — The IOAU controls all transfers of data between the peripheral devices and primary and extended storage. Transfers are initiated by CAU under program control.

IOU — The IOU controls all transfers of data between the peripheral devices and primary and extended storage. Transfers are initiated by CAU under program control.

interface — The logical path between two connected nodes.

interlock — A condition in which a peripheral unit is unable to perform an executable command until the condition is removed by the operator.

layered storage — A hardware architecture wherein different parts of main storage have different performance characteristics. On the 1110, and 1100/40, this refers to the fact that main storage consists of primary and extended storage.

line-id — Identification of the communications line to which one or more remote terminals are attached. Line-id is a unique identifier of one to six alphanumeric characters assigned by the installation.

main storage — The general-purpose high speed magnetic core, semiconductor or plated wire (1110 only) storage of the system directly addressable by the CPU, CAU, and IOAU/IOU and serving principally to contain executing programs.

mass storage — Auxiliary storage which has random access capability, as opposed to magnetic tape, for example. Includes any type of flying-head magnetic drum, FASTRAND drum units, disk, and unitized channel storage.

- word-addressable mass storage — Mass storage which is capable of being accessed in units of single words including any flying-head magnetic drum, and unitized channel storage. Word addressable mass storage may be simulated on disk.

FASTRAND-formatted mass storage	Mass storage which is accessible in units of 28 words (one sector). This may be on actual FASTRAND drum hardware, or may be simulated on other mass storage devices. The term FASTRAND in this manual refers to the format, not the hardware device, unless otherwise stated. This is the most common mass storage format.
fixed mass storage	Drum, unitized channel storage, FASTRAND drum units, and disk units declared to be fixed during the boot of the system. This storage is considered to be permanent (online).

MP — Multiprocessor. An application having two or more CPUs or CAUs.

network — All the nodes and interfaces in a system.

node — A system component.

offline — A condition in which hardware components are not under direct control of the operating system.

online — A condition in which hardware components are under direct control of the operating system.

P-register — Program address register. A CPU control register which contains the absolute main storage address of the next instruction to be executed. See Appendix A for numeric conversions.

peripheral equipment — Hardware that is distinct from the CPU, IOAU/IOU and main storage, and which provides the system with increased storage capacity, or with I/O capability.

remote site — Data terminal equipment that is time, space, or electricall distant from a central site, and capable of information exchange with the central site via communications lines.

site-id — Identification of a remote terminal. Site-id is a unique identifier of six alphanumeric characters assigned by the installation to a terminal or group of terminals.

SPU — System Partitioning Unit. A device used in the 1110 System which permits offline maintenance of units, enables the operator to logically partition the system into two or three independent systems, and can initiate a recovery sequence in the event of failure.

STU — System Transition Unit. It contains the controls and indicators for partitioning the SPERRY UNIVAC 1100/80 Systems into two independent systems. It also provides an automatic recovery feature and system power control.

subsystem	One or more peripheral units of the same type, plus a control unit which is connected to an available I/O channel. (Can be a dual subsystem).
symbiont device	Relatively slow-speed devices, such as card readers, card punches, and printers are controlled by symbionts and are used to provide direct input to and output from the system.
system conponent	The hardward units of a system. They include CPUs, IOUs, CAUs, IOAUs, primary storage, extended storage, control units, and devices and peripheral subsystems.
system drum/ disk	The mass storage unit to which the Executive is loaded. The system drum/disk is usually unit zero of the specified subsystem. The subsytem of the system drum/disk is specified during system's generation. This specification may be modified by the operator during tape bootstraps.
TTY	Teletypewriter equipment involving keyboard, printer, and sending and receiving equipment. Used primarily as a demand processing terminal.
unitized channel storage	Main storage which is treated as and accessed by peripheral I/O hardware.
UP	Unit processor. An application having a single CPU, CPU/IOU, or CAU/IOAU.

Software Definitions

backlog	The collection of runs which has been entered into the system and are held for facilities availability or unit directed time start. Backlog resides on mass storage.
batch processing	A mode in which runs are processed without any basic requirement for interactive manual data or controlled input during processing.
breakpoint	The division of symbiont-defined files into parts such that the output of completed parts may be initiated prior to run completion. This procedure allows more efficient utilization of printers and punches when large symbiont output files are involved.
checkpoint	Saves the run at a particular point in time for the purpose of subsequent restart in case of error or interruption.
deadline run	A batch run which is given certain schedule priorities to attempt run completion by a prescheduled time.

demand processing — A mode in which run processing is basically dependent on manual interaction (typically from a remote terminal) during processing. Commonly known as "time-sharing".

file — An organized collection of data, treated as a unit, and stored in such a manner as to faciliate the retrieval of each individual data item. Files are retained on auxiliary storage devices.

catalogued file	A file known to and retained by the Executive for a period of time not necessarily related to the life of a particular run, and retrievable by runs other than the run which originally created the file. In some cases, a catalogued file may be accessed simultaneously by two or more runs.
temporary file	A transient file created by, accessible to, and existing within the life of a single run (as opposed to catalogued file).

logical name — The name associated with a system component. The logical name is not required to connote the system component with which it is associated.

real time processing — A mode of operation in which the system's response to input is sufficiently fast to influence the operation being controlled. In the real time mode the program generally has exclusive use of a CPU/CAU. Generally, real time processing is under the influence of independent inputs from one or more communications devices. The real time mode may be entered from either batch or demand mode.

restart — Resumption of processing a run from a checkpoint rather than from the beginning of the run.

run — A group of tasks prescribed as a unit of work for the system. A @RUN control statement must be the first card or image of a run. A @FIN control statement is the last image.

run-id — Identifies a run to the Executive. Run-id may consist of one to six alphanumeric characters and is specified on the @RUN control statement. If the specified run duplicates a run-id already in the system, the Executive modifies the newly submitted run-id to make it unique. When the run-id is modified, both the original and the modified run-ids are output to the operator console.

swapping — The 1100 Series Operating System's method of moving low priority runs from main storage to mass storage in order to provide space to load higher priority runs into main storage for execution.

symbiont A complex of Executive routines providing the user interface with symbiont devices. Symbionts buffer the output so that symbiont devices can handle the high speed output which the cemtral processor provides. This allows system processing to proceed at the higher internal and mass storage speeds rather than at the relatively slow speed of symbiont devices .

TSS Terminal Security System

System Definitions

bootstrap Act of loading (booting) the Executive into main storage along with certain other initialization functions which vary depending on the type of bootstrap performed. Bootstrap is used synonymously with boot.

initial bootstrap The method wherein the operating sy stem is read from the boot tape and copied onto mass storage devices. At the conclusion of the initialization, the Executive control routines, called the resident Executive, are read into main storage and are given control.

recovery bootstrap The method wherein the Executive control routines are read from tape, disk, or drum and copied into main storage.

autorecovery A recovery bootstrap of the system taken when a system malfunction or error is detected. The recovery may be system initialed (programmed recovery) or ACU/SPU/STU initiated. Operator intervention is not needed for either type.

panic dump The process of documenting portions of main and mass storage for future analysis. Panic dumps are usually initiated by the operator or the Executive following a system error.

system generation The process of tailoring the operating system to the particular hardware configuration and software requirements of a site. The end result of a system generation is a tape that contains a copy of the operating system in a form suitable for loading into the computer system (i.e., a boot tape).

zero stop A CPU/CAU stop initiated by the Executive due to either software or hardware detected faults.

COMMUNICATION LINKS

The transmission and reception of data to and from a computer require a highly reliable electronic conversion process in most instances. In general, data are generated and processed by both terminals and computers in coded formats utilizing patterns of binary bits. Transmission of data over communication lines requires a conversion of data from an electromechanical or magnetic storage format to electrical communication signals. These signals represent tones that are audible only when used to drive a suitable speaker-like or diaphragm device such as a telephone receiver. On receipt, the signals or tones are reconverted to equivalent electrical energy to rerecord the data mechanically or magnetically.

The devices that perform this conversion process at both the sending and receiving ends of a communication line are known as either modems or Data Sets. (The word "modem" is an acronym for the function "modulate-demodulate." Modulation is the conversion of impulses to tones; demodulation is the reverse). Data Sets are a specific type of modem installed by Bell System companies. The modem, in effect, is the telephone station through which a terminal talks to the timesharing computer. In most cases, Data Sets include telephone instruments and dials.

The actual connection of terminals to modems is accomplished in either of two ways:

1. The terminal can be "hard-wired" to the modem. This indicates that the wiring of the terminal is connected directly into the transmitter/receiver unit.

2. The modem can incorporate an audio coupler. With this approach, the connection is established between the terminal location and the computer on an ordinary voice-grade dial telephone. The telephone handset is inserted into the audio coupler of the modem, which then generates or reads tones into or from the telephone instrument.

The hard-wire installation is more reliable, of higher quality, and of greater permanence. However, this approach requires professional installation, represents a longer-term commitment, and is less flexible.

By comparison, audio coupling is more subject to line interference but far more flexible. With this technique, timesharing service can be established or discontinued at any point where the user has a telephone instrument.

Timesharing transmissions can be carried over many different kinds of communication lines. In general, line costs are directly related to the transmission capacity and length of a given line.

The least expensive, lowest-capacity transmission line is known as a half-duplex circuit. This is simply a circuit with two wires - one signal line and a return, or ground - between two points. With a half-duplex or two-wire circuit, data can be transmitted in only one direction at a time. Thus a terminal cannot be receiving data from the computer while the operator is sending data. This type of communication link has been used primarily for telegraphic service.

The next step up is to use a four-wire, or full-duplex, circuit. This is the type of connection normally established for telephone conversions. Most timesharing services today use full-duplex circuits. These can be acquired either through dial service or on a leased-line basis. (With leased-line service, a full-duplex line is rented on a regular basis from telephone common carriers.) In general, a full-duplex line has the capacity to transmit or receive at a rate of up to 2400 baud, or bits of data per second. This is equivalent to approximately 240 characters per second.

Consideration of this line capacity gives further dimension to earlier discussions of terminal speed and automated transmission from off-line storage media. Recorded data can be transmitted at speeds of up to 240 characters per second. However, even under automatic operation, printing terminals are limited to 30 characters per second-and a typist entering data directly from a keyboard is effectively limited to seven or eight characters per second.

Where data transmission requirements are greater, additional lines can be added. In general, communication lines with capacities greater than full-duplex are known as broadband service. Transmission capabilities are directly proportional to the lines available. Thus four lines would make a transmission rate of 4800 baud available, eight lines would carry 9600 baud, and so on. Services regularly available from telephone carriers extend to 32 lines. However, users of timesharing utility services will rarely require or encounter services involving more than full-duplex lines.

In some cases, however, timesharing utilities do use a technique known as multiplexing to concentrate transmission from a number of users over the same telephone lines. Multiplexors are satellite communication processors. (Minicomputers are often used for multiplexing.) A large number of timesharing users, sometimes as many as 132, can be linked to a single multiplexing point. Their transmissions are then carried from the multiplexor to the central computer over either full-duplex or broadband lines. Typically, a multiplexor will be set up in a city remote from the central computer. For example, many timesharing companies operate computers in New York. These organizations then establish multiplexing points in major cities such as Chicago and Los Angeles, where users can link into the national timesharing network through local telephone calls.

*U.S. GOVERNMENT PRINTING OFFICE : 19800-328-231/6531

FEDERALLY COORDINATED PROGRAM (FCP) OF HIGHWAY RESEARCH AND DEVELOPMENT

The Offices of Research and Development (R&D) of the Federal Highway Administration (FHWA) are responsible for a broad program of staff and contract research and development and a Federal-aid program, conducted by or through the State highway transportation agencies, that includes the Highway Planning and Research (HP&R) program and the National Cooperative Highway Research Program (NCHRP) managed by the Transportation Research Board. The FCP is a carefully selected group of projects that uses research and development resources to obtain timely solutions to urgent national highway engineering problems.*

The diagonal double stripe on the cover of this report represents a highway and is color-coded to identify the FCP category that the report falls under. A red stripe is used for category 1, dark blue for category 2, light blue for category 3, brown for category 4, gray for category 5, green for categories 6 and 7, and an orange stripe identifies category 0.

FCP Category Descriptions

1. Improved Highway Design and Operation for Safety

Safety R&D addresses problems associated with the responsibilities of the FHWA under the Highway Safety Act and includes investigation of appropriate design standards, roadside hardware, signing, and physical and scientific data for the formulation of improved safety regulations.

2. Reduction of Traffic Congestion, and Improved Operational Efficiency

Traffic R&D is concerned with increasing the operational efficiency of existing highways by advancing technology, by improving designs for existing as well as new facilities, and by balancing the demand-capacity relationship through traffic management techniques such as bus and carpool preferential treatment, motorist information, and rerouting of traffic.

3. Environmental Considerations in Highway Design, Location, Construction, and Operation

Environmental R&D is directed toward identifying and evaluating highway elements that affect the quality of the human environment. The goals are reduction of adverse highway and traffic impacts, and protection and enhancement of the environment.

4. Improved Materials Utilization and Durability

Materials R&D is concerned with expanding the knowledge and technology of materials properties, using available natural materials, improving structural foundation materials, recycling highway materials, converting industrial wastes into useful highway products, developing extender or substitute materials for those in short supply, and developing more rapid and reliable testing procedures. The goals are lower highway construction costs and extended maintenance-free operation.

5. Improved Design to Reduce Costs, Extend Life Expectancy, and Insure Structural Safety

Structural R&D is concerned with furthering the latest technological advances in structural and hydraulic designs, fabrication processes, and construction techniques to provide safe, efficient highways at reasonable costs.

6. Improved Technology for Highway Construction

This category is concerned with the research, development, and implementation of highway construction technology to increase productivity, reduce energy consumption, conserve dwindling resources, and reduce costs while improving the quality and methods of construction.

7. Improved Technology for Highway Maintenance

This category addresses problems in preserving the Nation's highways and includes activities in physical maintenance, traffic services, management, and equipment. The goal is to maximize operational efficiency and safety to the traveling public while conserving resources.

0. Other New Studies

This category, not included in the seven-volume official statement of the FCP, is concerned with HP&R and NCHRP studies not specifically related to FCP projects. These studies involve R&D support of other FHWA program office research.

* The complete seven-volume official statement of the FCP is available from the National Technical Information Service, Springfield, Va. 22161. Single copies of the introductory volume are available without charge from Program Analysis (HRD-3), Offices of Research and Development, Federal Highway Administration, Washington, D.C. 20590.