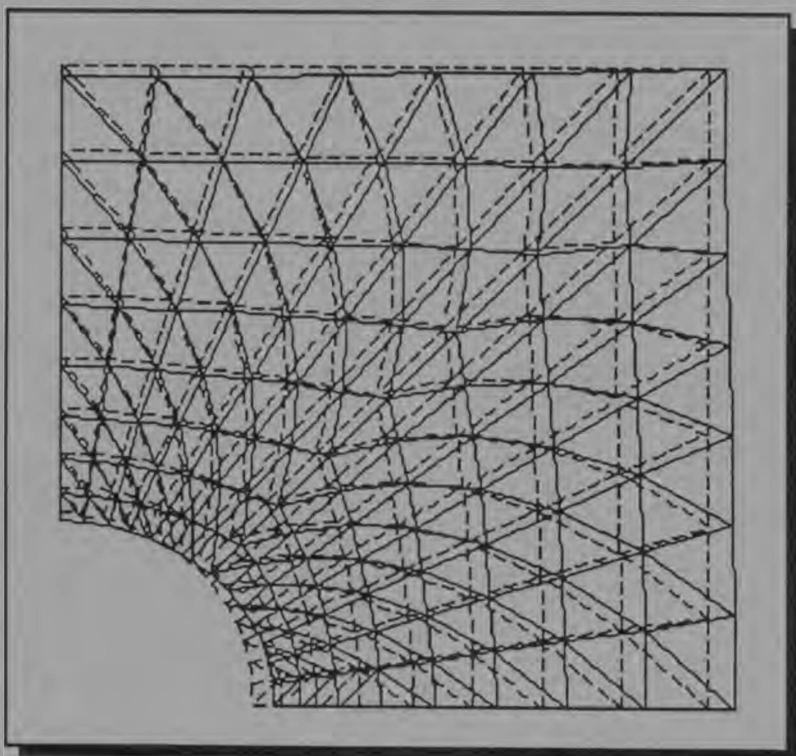
PC-Elastic™

Instructional Finite Element Analysis
for Solving Elasticity Problems
With the IBM-PC®



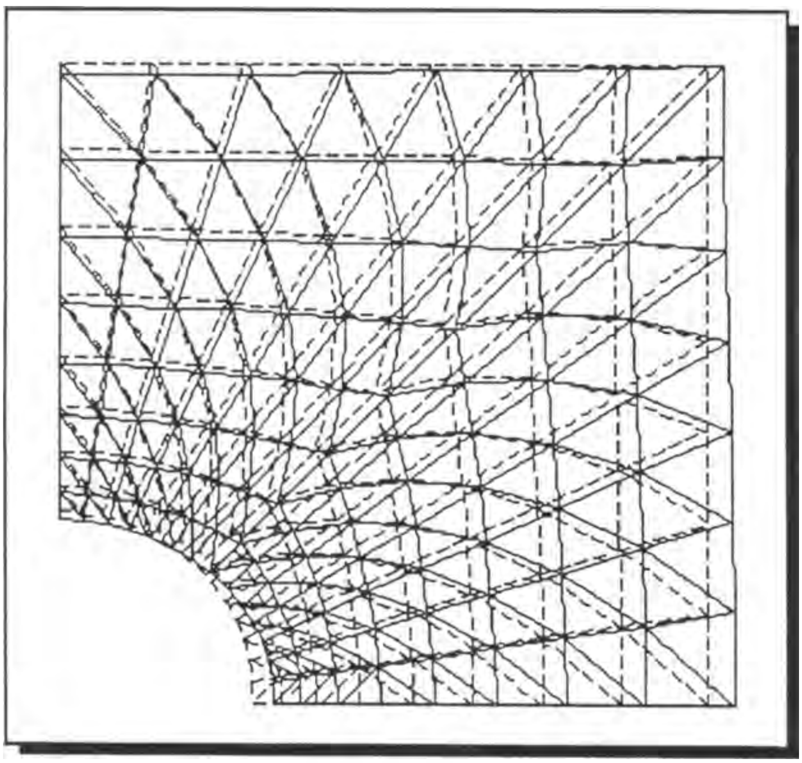
J. R. Cooke
D.C. Davis
E.T. Sobel
R.S. Gates
R.N. Perl

Cooke Publications, Ltd. • Ithaca, NY

Prog: shows nodes
Plot - F1
<> gives mesh

PC-Elastic™

Instructional Finite Element Analysis
for Solving Elasticity Problems
With the IBM-PC®



J. R. Cooke
D.C. Davis
E.T. Sobel
R.S. Gates
R.N. Perl

Before you begin

Versions: The PC-Elastic (PCE) student version, which is limited to 300 degrees of freedom, includes an applications disk with one or more demo projects and this manual. The professional version has the same functional capabilities but handles larger problems, up to the limits of RAM.

Updates: Return a copy of the Registration card from the Appendix to have your name placed on our mailing list for notification of updates and related products.

Hardware: PC-Elastic should normally be used with an IBM-PC (or compatible) with graphics output (CGA, EGA, etc.), at least 512K of RAM, and a math coprocessor (e.g., 8087) with at least two disk drives. PCE works best with generous memory and disk storage. This configuration is satisfactory for instructional problems.

The PCE code is large and requires the use of overlays to function properly. This size necessitated an extended testing period. The program is distributed on a double-sided, double density disk.

System Software: PC-Elastic functions properly with PC DOS 2.0 and later versions. DOS is not supplied with PCE.

Protecting your investment: ***Before using your PCE distribution disk***, place tape over the write-protect tab on the diskette. Then make a working copy and store the original in a safe place. For your convenience we have NOT copy-protected the disk. You may make an archival copy using the standard copying process, and you may transfer the program to your hard disk. Unless explicitly covered by a written contract, PCE is licensed as a single-user product. See the license agreement. We have spent hundreds of hours in the development of PCE, so we appreciate your assistance in protecting our investment!

After exploring the demo example (IN DEMO MODE) remove it from the working copy. *All data files related to a single example must be in the same subdirectory.* The master file contains status information about the other files.

If you have problems: 1) Re-read the documentation. 2) Read the supplements to the documentation supplied with PCE. This is where we document extensions to the software or manual and provide hints to clarify frequently asked questions. 3) Make a reasonable effort to resolve the problem. 4) Then, please write or call. Be sure to provide enough background (equipment, system software, PCE version number and serial number displayed at logon) for us to respond. Provide sufficient detail for us to replicate the problem situation. In the case of site licenses for classroom use, all questions should be routed through the person who obtained the site license. The license fee does not include support for student consultation. No support will be provided for stolen copies!

Getting Started: Proceed only after making your backup and storing the distribution disk. Although an experienced PC user probably could use many of the program features without studying this manual, reading this manual should make your learning experience more enjoyable and productive. You might otherwise overlook some of the nifty features. In addition, the subject addressed by PCE is likely to be more complex than most applications you have encountered.

This manual was prepared using FullWrite Professional, MathWriter 1.4, and FullPaint. The typesetting was produced using a LaserWriter Plus with a Macintosh II.

DISCLAIMER OF WARRANTY

COOKE PUBLICATIONS HEREBY DISCLAIMS ALL WARRANTIES WITH REGARD TO THIS SOFTWARE INCLUDING WARRANTIES OF MERCHANTABILITY AND FITNESS FOR ANY PARTICULAR PURPOSE (WHETHER OR NOT COOKE PUBLICATIONS HAS BEEN ADVISED OF SUCH PURPOSE). THIS SOFTWARE IS NOT INTENDED BY EITHER PARTY TO BE "CONSUMER GOODS" UNDER ANY STATE OR FEDERAL LAW.

NEITHER COOKE PUBLICATIONS NOR ANY INDIVIDUAL ASSOCIATED WITH THE DESIGN OR PRODUCTION OF THE SYSTEM SHALL HAVE ANY LIABILITY OR RESPONSIBILITY FOR DAMAGES OF ANY KIND, INCLUDING SPECIAL, DIRECT, INDIRECT, INCIDENTAL, OR CONSEQUENTIAL, ARISING OUT OF OR RESULTING FROM THE USE OF THE SYSTEM OR FROM ITS PERFORMANCE OR FAILURE TO PERFORM ANY FUNCTION WHETHER OR NOT COOKE PUBLICATIONS OR ANY SUCH INDIVIDUAL HAS BEEN ADVISED OF SUCH DAMAGES OR POSSIBILITY OF DAMAGES. USERS RELY UPON THE SYSTEM AND ITS RESULTS SOLELY AT THEIR OWN RISK.

Limited Warranty: Cooke Publications warrants that the manuals and the disk(s) on which the system is recorded are free from defects in materials and workmanship under normal use and service for a period of thirty (30) days from the date of purchase. Cooke Publications' entire liability under this warranty shall be limited to replacement of the defective manual or disk upon its return to Cooke Publications along with a copy of the receipt. If the failure of a disk resulted from accident, abuse, or misapplication of the product, then Cooke Publications shall have no responsibility to replace the disk under this warranty.

Cooke Publications, Ltd., P.O. Box 4448, Ithaca, New York 14852

ISBN 0-940119-11-0

Preface

The finite element method (FEM) is rapidly becoming one of the premier computational techniques of science. Consequently, university curricula in engineering and mathematics should broaden their offerings to make this powerful and unifying tool accessible to undergraduates, not just to the traditional graduate student audience.

FEM instruction which relies upon textbook-only presentation can allow computational details to obscure both the intellectual content and the generality of the method. Because of its procedural complexity the FEM becomes an attractive computational technique only if used with a computer. Therefore, interactive, visually oriented, instructional software is needed to supplement textbooks.

Two major obstacles to widespread use of the FEM in the undergraduate curriculum have been addressed by PC-Elastic™. First, little finite element software has been developed specifically for the novice user. Commercial FEM code emphasizes generality, rather than ease of use by the novice. Second, the equipment and support costs associated with high-end instructional graphics labs have been an obstacle. However, the popularization of microcomputers with graphics means that this larger audience can be served more economically now. "Instructional Software Makes The Finite Element Method Accessible" in the September 1988 issue of *Academic Computing* (pp 34, 35, 54, 56) offers additional commentary.

PC-Elastic has evolved over several years with several different graphics drivers. Designing, writing, and testing more than five hundred pages of Pascal source code and this documentation have required hundreds of hours. We hope this effort substantially reduces the effort and learning time required by new users of this important technique.

Several people have helped make PC-Elastic possible. Professor L.J. Segerlind's *Applied Finite Element Analysis* provided inspiration. The constructive comments of Professor J. F. Booker and his students provided classroom reality checks on this and the companion program MacPoisson™. We also thank Dr. Kenneth King for his encouragement while he was serving as Vice Provost for Computing at Cornell University.

JRC, Ithaca
DCD, Pullman
ETS, Ithaca
RSG, Lexington
RNP, Syracuse.. May 1989

Credits

PC-Elastic™

Concepts...J. R. Cooke, D.C. Davis, E.T. Sobel
Programming................................E.T. Sobel, R. Perl, P. Hummel, R.S. Gates
Manual...J.R. Cooke, D.C. Davis
Performance Testing..R. Perl

Editor...Nancy B. Cooke
Graphic Design..Susan MacKay
Typesetting..J. Robert Cooke
Proofreading...Betty Czarniecki

Chapter 1

Finite Element Method Overview

Steps in the problem solving process

Finite Element Analysis is a numerical method for solving problems by breaking the physical space into discrete "elements" for which the approximate solution is known. The composite of these finite elements is used to form the global solution. Specifically, PC-Elastic (**PCE**) solves problems governed by the biharmonic equation in 2 dimensions (planar) and 3 dimensions (problems having symmetry about an axis).

The method of solution can be broken into three main steps: first, you generate a mesh to describe the geometry of the problem. This corresponds to the discretization of space mentioned above. Next, you ascribe constraints to the problem. These constraints include the physical properties, as well as loads (distributed and line), fixed displacement components, and surface stresses (normal, tangential and in coordinate directions). Finally, PCE solves the problem, which involves automatically forming and solving a set of simultaneous linear algebraic equations dictated by the first two steps. Output includes both tabular and graphical displays. PCE provides a visual framework for both input and output.

The steps require a tremendous amount of bookkeeping in order to track the position of the mesh elements, the relationships between elements, the properties of different elements, and the boundary conditions at nodes and elements. *The power of PC-Elastic lies in its ability to free you from these bookkeeping tasks, thus allowing you to concentrate on the problem being modeled.*

The six selections on PC-Elastic's main menu follow, along with a brief description of the function of each selection. These steps are illustrated using elementary examples. This chapter outlines both an axisymmetric and a planar problem. These examples are considered in detail in Chapters 2 and 3, respectively. Both problems are formulated using two mesh generating regions.

The treatment in Chapter 1 is intentionally brief in order to not obscure the "big picture". We suggest that you read this chapter first to develop an overview. Chapter 2 presents brief step-by-step instructions as a guided tour. Chapter 3 is organized as a detailed reference for the commands. These problems are included on the distribution disk.

A. Column on tapered footing subjected to axial loading

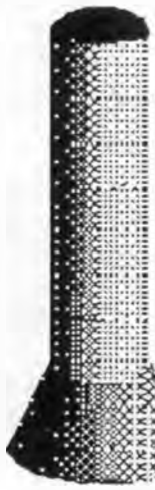


Fig 1.1 Column problem

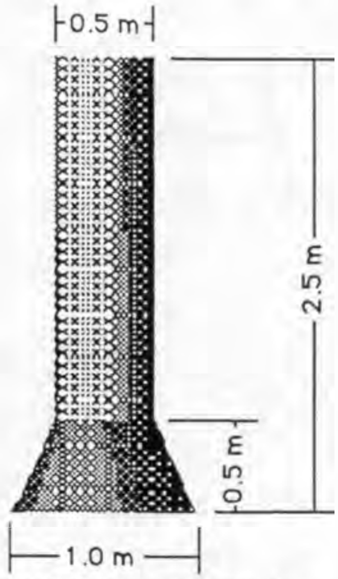


Fig 1.2 Column schematic

Figure 1.1 depicts a circular column resting on a tapered footing. The dimensions, properties, and boundary conditions are shown in Figure 1.2, which also shows how the axial symmetry is being used. The column is represented as a rectangle rotated about the z-axis; the footing is represented as a trapezoid rotated about the z-axis. Assume that the column and footing are different, homogeneous materials. Find the displaced shape of the column and footing as well as stresses and strains.

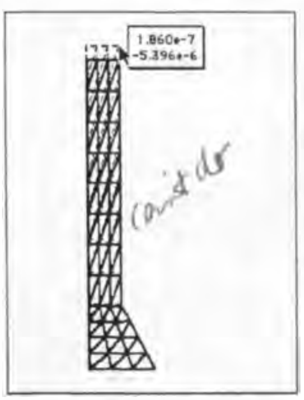


Fig 1.3 Displacement of the column and footing

Fig 1.3 displays the solution you seek. The displacement at all interior points is uniquely determined; the solution must satisfy the biharmonic equation and the conditions you specify on the boundaries.

The column becomes shorter due to the vertical downward loading; there is a smaller radial displacement. (Refer to Chapter 2 for a step-by-step guide to the solution.)

A1. GEOMETRY - Overall body definition: the problem space is defined and subdivided for later processing.

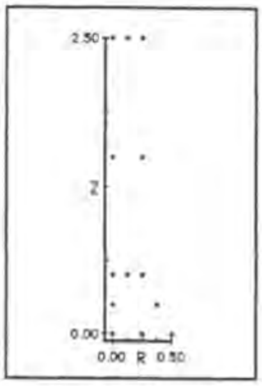


Fig 1.4 Points

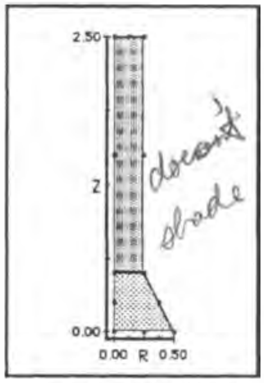


Fig 1.5 Regions

The geometry is defined using points, lines, and arcs (Fig 1.4). The problem, therefore, looks like an unfinished "connect the dots" game prior to region definition, described next. Several tools aid in the rapid generation of points on lines and arcs.

To utilize the automatic mesh generator, you must define regions (Fig 1.5) meeting specific criteria. The fundamental region is the curvilinear quadrilateral, a shape defined by four vertices with each of the four sides having one intermediate point to which a quadratic curve is fitted, for a total of eight defining points. Complex problems have borders which cannot be described by only one curvilinear quadrilateral. In this case, you break the total space into multiple regions. The actual region generation is done by selecting the predefined points in counterclockwise order, beginning with a vertex. Once eight points are selected, PC-Elastic draws and shades the polygon region.

A2. MESH - Mesh generation: the automatic mesh generator takes each properly defined region and breaks it into triangular subregions referred to as "elements".

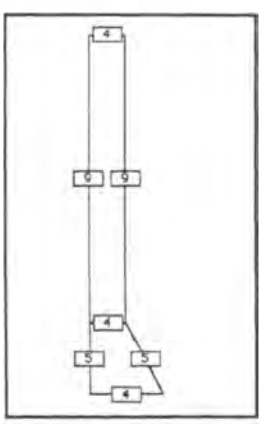


Fig 1.6 Nodes/side

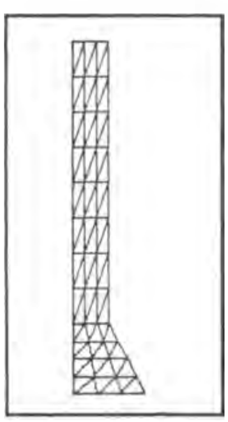


Fig 1.7 Mesh

Each element vertex is known as a "node". The user specifies the number of nodes to be placed on each side of a mesh generating region (Fig 1.6), with the constraint that opposite sides of a region have the same number of nodes and that regions sharing a common boundary have the same number of nodes on the common boundary.

For properly defined regions PC-Elastic enforces these constraints automatically, creates the mesh (Fig 1.7), and also keeps track of the coordinates of each of the generated nodes. Facilities to modify the mesh in other ways (i.e., slightly move a node, reorient a diagonal, and subdivide elements) are available. When the mesh is satisfactory, the program goes through several calculations to identify unique lines and the connectivity of line segments, as well as to minimize the bandwidth of the global "stiffness" matrix. This step reduces both memory requirements and computational time.

A3. PROPERTIES - Define properties and boundary conditions:

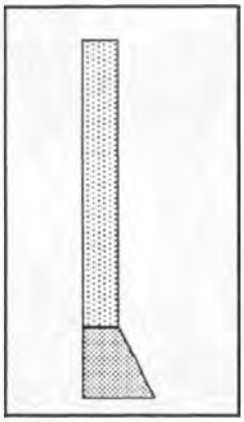


Fig 1.8
Properties

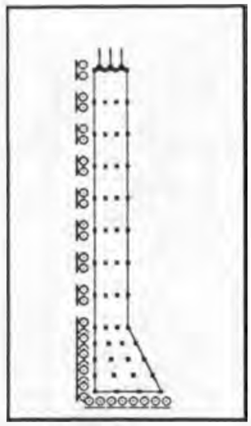


Fig 1.9
Constraints

As mentioned earlier, the solution of a PC-Elastic problem involves approximating the solution over a large number of small, connected elements. The material properties of each element are, therefore, crucial to the outcome of the problem. From the properties menu, choose the material properties section to set the element material properties by element, region, or whole body (Fig 1.8). Each element can contain a distributed body force.

Next, prescribe the boundary conditions (Fig 1.9). You can set fixed nodal displacement components, nodal forces, as well as stresses at boundaries. ***Responsibility for formulating a meaningful, well-posed problem rests with you.***

A4. SOLVE - Solution of the system of equations:

If you have completed the previous steps properly, PCE forms and solves the equations which describe the physical situation. PCE uses the Gaussian elimination technique to solve the system of equations, thereby, getting the resulting nodal displacements. Once PCE finds the nodal displacement components, you can direct the program to compute element and nodal stresses and strains.

A 5. PLOT - Plot results:

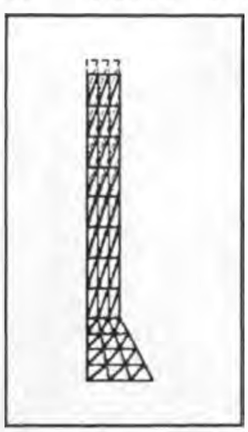


Fig 1.10
Displacement plots

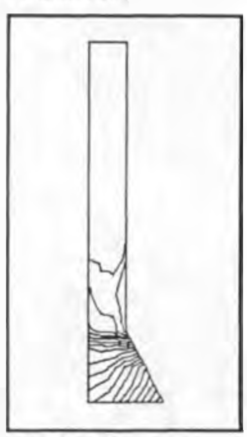


Fig 1.11
Contour plots

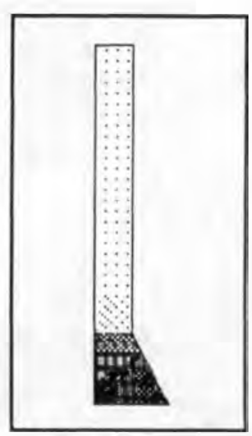


Fig 1.12
Shaded plots

PC-Elastic produces displacement plots (Fig 1.10), contour plots of nodal values, and shaded plots for element values (Fig 1.12). You can control the resolution to focus on a particular section of the problem.

You can produce both diagnostic and presentation quality plots.

PCE provides labeling of nodes, elements, contours, average values, and text. Zooming capabilities allow you to refine the plots as needed.

A6. LIBRARY - Project creation and data examination:

Node	R	Z
1	8.88474047671870e-8	0.00000000000000e+0
2	8.59403350719807e-8	0.00000000000000e+0
3	9.24819311001363e-8	-3.27826322285533e-8
4	5.28840155101649e-8	0.00000000000000e+0
5	7.89693066297271e-8	-7.46732216726424e-8
6	8.70722406786912e-8	-1.08048115767492e-7
7	0.00000000000000e+0	0.00000000000000e+0
8	4.47337572456687e-8	-9.96883506622350e-8
9	7.08897140138613e-8	-1.73178350383073e-7
10	8.82290884879920e-8	-2.31226819501914e-7
11	0.00000000000000e+0	-1.09731417847710e-7
12	3.74424004018065e-8	-2.12065466928361e-7
13	6.25170188028887e-8	-3.07383508142780e-7
14	1.13372228007585e-7	-4.44515151364267e-7
15	0.00000000000000e+0	-3.25829835191168e-7
16	3.12680668579877e-8	-3.41175083636575e-7
17	6.84523325082305e-8	-4.77946837562609e-7
18	1.81149381672669e-7	-1.05967920102389e-6
19	0.00000000000000e+0	-3.53363758170083e-7
20	3.00137682697716e-8	-4.96622133632294e-7
21	1.20816446153052e-7	-1.05844988432859e-6

Fig 1.13 Tabular output

You can examine the tabular output from within PCE (Fig 1.13), as well as with a word processor. With the output data you can compute other interesting properties.

Use the Library to examine the input and the intermediate calculation steps. The library also enables you to generate input files without graphical support (in order to study the details of the process) or to produce a specialized custom mesh.

The plate example which follows parallels the column example just described, but illustrates a planar problem with a curved boundary. To the extent possible, the text repeats the previous discussion. Alternatively, you can skip the plate example and proceed to the step-by-step tutorial in Chapter 2.

B. Flat plate with hole and tensile loading

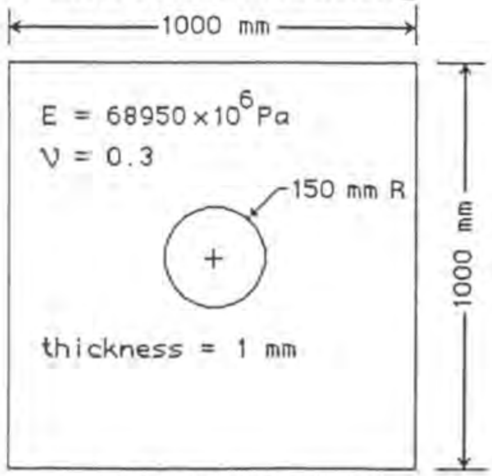


Fig 1.14 Flat plate with hole

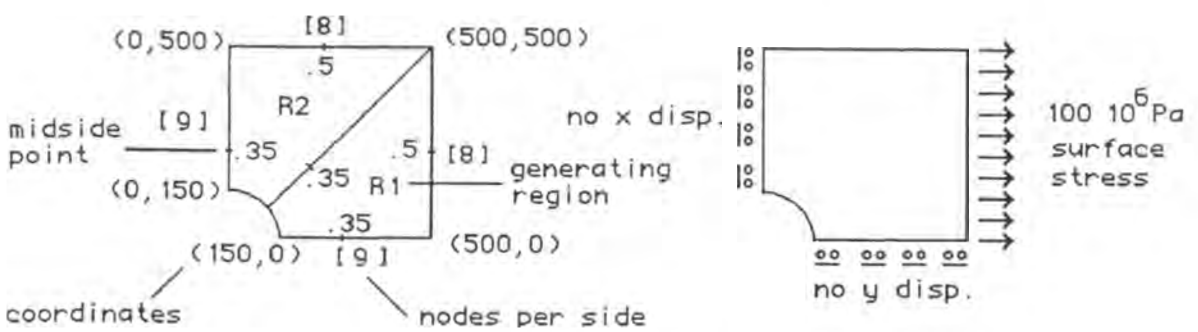


Fig 1.15 Schematic of plate

Figure 1.14 depicts a quadrant of flat plate with a circular hole. Since it is symmetrical, you need consider only one fourth. Rollers which allow displacements in only one direction replace the lines of symmetry. The hole and upper surface are traction free, while the right side has a surface load. The dimensions, properties, and boundary conditions are shown in Figure 1.15. You are to find the displacement and stresses.

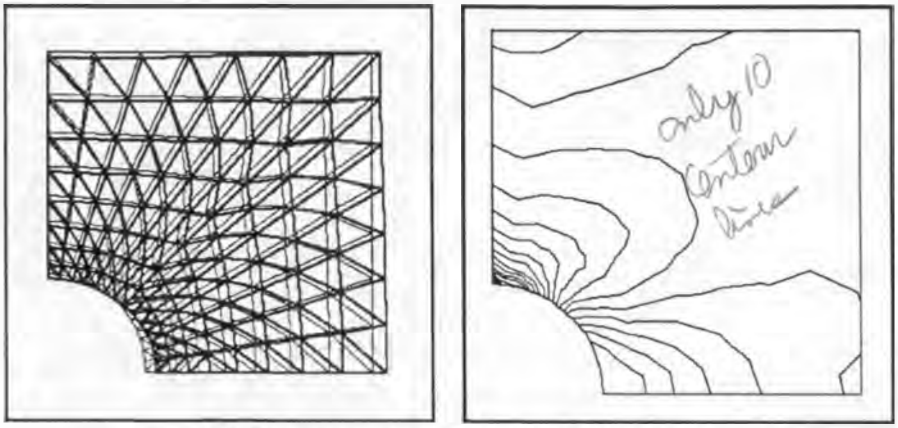


Fig 1.16 Displacement and constant stress lines

The displacement at all interior points is uniquely determined; the solution must satisfy the biharmonic equation and the conditions you specify on the boundaries.

The plate elongates in the x direction and becomes shorter in the y direction; the hole elongates in the loading direction and shrinks in the other direction (Fig 1.16). See Chapter 3 for a step-by-step description.

B1. GEOMETRY - Overall body definition: the problem space is defined and subdivided for later processing. * don't shade

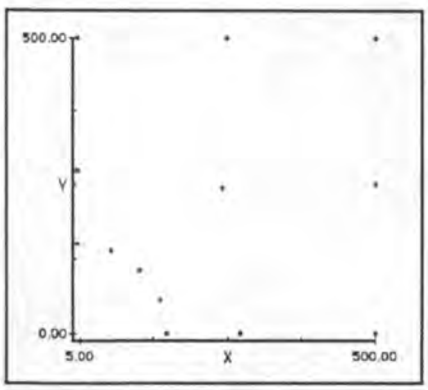


Fig 1.17 Points

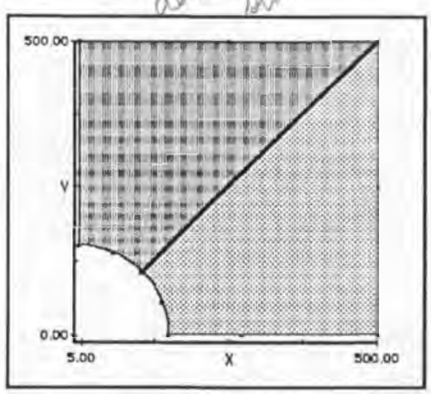


Fig 1.18 Regions

You input the geometry as points, lines, and arcs. The problem (Fig 1.17), therefore, looks like an unfinished "connect the dots" game.

In order to utilize the automatic mesh generator, you must define regions (Fig 1.18) meeting specific criteria. The fundamental region is the curvilinear quadrilateral, a shape defined by four vertices with each of the four sides having one intermediate point to which PCE fits a quadratic curve, for a total of eight defining points. Complex problems have borders which cannot be described by only one curvilinear quadrilateral. In this case, once you have entered all the points defining the entire problem space, you break the total space into multiple regions. You do the actual region generation by selecting the predefined points in counterclockwise order, beginning with a vertex. Once you

* will shade elements or elements in region if choose those two options

have selected eight points, PC-Elastic draws and shades the polygon region.

B2. MESH - Mesh generation: the automatic mesh generator takes each properly defined region and breaks it into triangular subregions referred to as "elements".

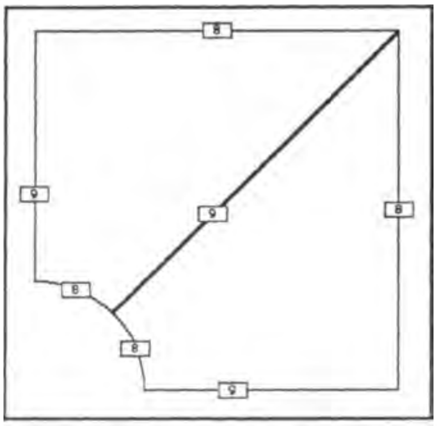


Fig 1.19 Nodes per side

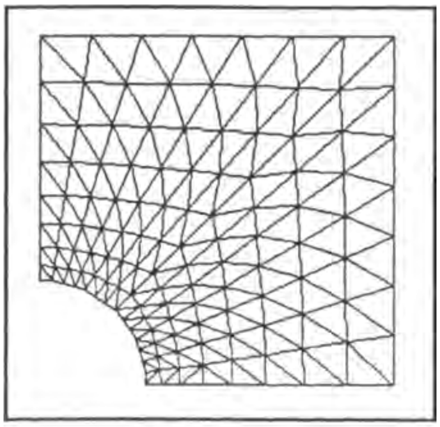


Fig 1.20 Mesh

Each element vertex is known as a "node". You specify the number of nodes to be used in each mesh generating region (Fig 1.19), with the constraint that opposite sides of a region have the same number of nodes and that regions sharing a common boundary have the same number of nodes on the boundary.

For properly defined regions, PC-Elastic enforces these constraints automatically, creates the mesh (Fig 1.20), and also keeps track of the coordinates of each of the generated nodes. Facilities to modify the mesh in other ways (i.e., move a node slightly, reorient a diagonal, and subdivide elements) are available. When the mesh is satisfactory, the program goes through several calculations to identify unique lines and the connectivity of line segments, as well as to minimize the bandwidth of the global "stiffness" matrix. This step reduces memory requirements and reduces computational time.

B3. PROPERTIES - Assign properties and boundary conditions:

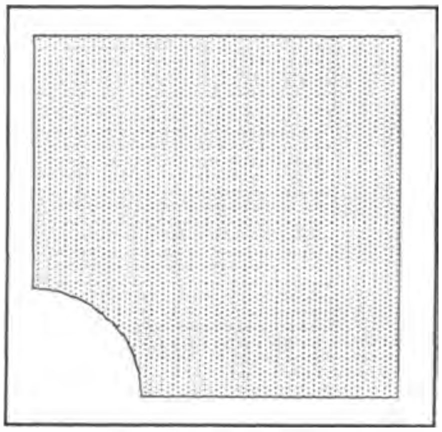


Fig 1.21 Properties

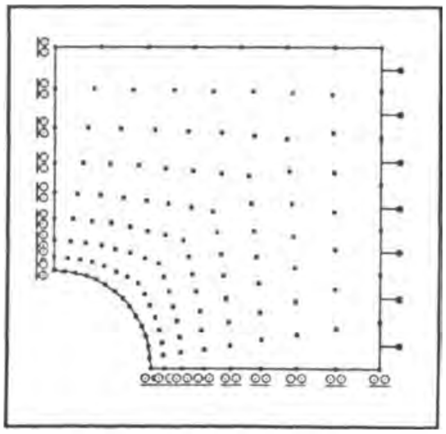


Fig 1.22 Constraints

As mentioned earlier, the solution of a PC-Elastic problem involves approximating the solution over a large number of small, connected elements. The material properties of each element are, therefore, crucial to the outcome of the problem. From the properties menu, you choose the material properties section to set the element material properties by element, region, or whole body (Fig 1.21). Each element can include a distributed body force.

Next, you prescribe the boundary conditions (Fig 1.22). You can prescribe fixed nodal displacement components, along with nodal forces, as well as surface stress on boundaries (normal and tangential or along coordinate directions). ***Responsibility for formulating a meaningful, well-posed problem rests with you.***

B4. SOLVE - Solution of a system of equations:

If you have completed the previous steps properly, PCE forms and solves the equations which describe the physical situation being modeled. This part of the program uses the Gaussian elimination technique to solve the system of equations, thereby, getting the resulting nodal displacement components. Once PCE solves two displacement components at each node, you can direct the program to compute element and nodal stresses and strains.

B5. PLOT - Plotting results:

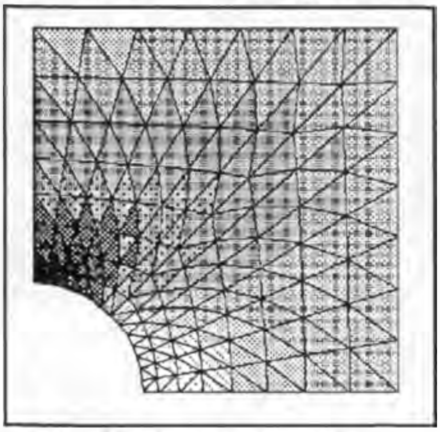


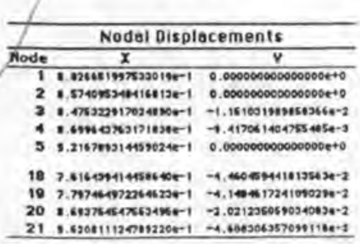
Fig 1.23 Element stress plot

PC-Elastic produces displacement plots (Fig 1.16), contour plots of nodal values (Fig 1.16), and shaded plots for element values (Fig 1.23). You can control the resolution to focus on a particular section of the problem.

You can produce both diagnostic and presentation quality plots. PCE provides labeling of nodes, elements, contours, average values, in addition to text. Zooming capabilities allow you to refine the plots as needed.

B6. LIBRARY - Create projects and examine data:

File Management



Nodal Displacements

Node	X	Y
1	8.826681997533019e-1	0.0000000000000000e+0
2	8.574095348416813e-1	0.0000000000000000e+0
3	8.476322917024830e-1	-1.161001989858366e-2
4	8.699643763171836e-1	-9.417061404755485e-3
5	5.216789314459024e-1	0.0000000000000000e+0
18	7.616439414458640e-1	-4.460459441813563e-2
19	7.797464972264623e-1	-4.148461724109029e-2
20	8.692764547563496e-1	-3.021236059034083e-2
21	9.620811124789220e-1	-4.608306357099118e-3

Fig 1.24 Tabular output

You can examine the tabular output from within PCE (Fig 1.24), as well as with a word processor. With the output data you can compute other interesting properties.

Use the Library to examine the input and the intermediate calculation steps. The library also enables you to generate input files without graphical support (in order to study the details of the process) or to produce a specialized custom mesh.

Chapter 2

PC-Elastic™ User's Guide

PC Basics

This section on PC Basics contains only background information and can be skipped by the experienced user. In the manual we assume that you are already familiar with the PC computer and application programs such as a word processor. As a result of our following the PC protocols, we hope that you can concentrate more comfortably upon the content of the finite element method, rather than upon the vehicle used to present this powerful technique.

The program provides an immediate visual indication that it is performing your command. Computation-intensive commands are accompanied by banners or other visual indicators of the progress of your analysis.

We have provided multiple paths through the analysis so that you may proceed in the most comfortable and intuitive manner possible, given the fundamental constraints of the algorithms. In addition, we present the most probable next step as the default, indicated by a bracketed value in the prompt message.

The PC supports extensive use of graphics which, we think, makes it an excellent instructional vehicle for the finite element method. The problem formulation phase, if done without graphics support, requires the tedious and error-prone preparation of text files of tabular data. Similarly, interpretation of the massive tabular results is difficult and less comprehensible without graphics.

Let's briefly review some of the most important PC topics which are used in the discussion which follows. PCE runs on the IBM-PC series of microcomputers (IBM-PC, IBM-PC/XT, IBM-PC/AT and PS/2 series) with 512K or more of memory and the color graphics (CGA) and enhanced graphics (EGA) adapter. A driver for the Hercules graphics card is also provided. VGA also

An input routine serves as the heart of the user interface by providing error checking and access to text and graphics screens. Most inputs (except single keystroke commands for graphics) use this input routine. The input routine is in use whenever you see a request for input in the form:

prompt [default] ?

where *prompt* is a descriptive request, and

default enclosed in brackets is a default value assigned if you press Enter.
? Enter displays a help message number
< Enter returns to a previous step (presently disabled.) these work where they are useful
> Enter branches forward (presently disabled.)
: Enter (in Geometry) allows you to select a previous point with the cursor
^Q aborts program (~~presently disabled.~~) or Escape
^C aborts program after a warning (~~presently disabled.~~)
^T displays the text screen
^G displays the split graphics screen only F1
^V displays the full graphics screen
Shift PrtSc prints the displayed screen (text or graphics) if you executed GRAPHICS.COM

Notes:

Enter denotes the "return" key
^ denotes that you must press and hold the control key while you type the next character.

If the foregoing discussion was not a review, we recommend that you take a few moments to review your PC owner's manual and to practice using your word processor.

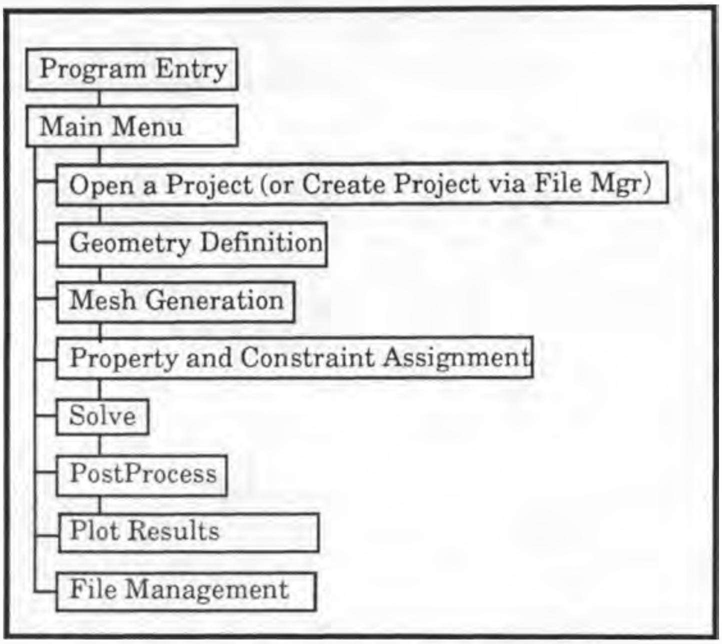


Fig 2.1 The structure of PC-Elastic

Fig 2.1 depicts the organization of the major parts or modules of PCE. In the following two examples you explore each of the modules.

Illustrative Examples

In this chapter we present in tutorial format the step-by-step details for the column example presented in the Chapter 1 overview. Chapter 3 presents a complete description of all PCE commands, including those mentioned in this chapter, and provides complete procedural details using the flat plate example.

PC-Elastic can solve classical elasticity problems governed by the biharmonic equation. Because we intend this to be a first introduction to the FEM, we provide a quick tour of the procedural details.

Typographic Convention:
• A bullet (•) denotes a step to be executed.

Column on tapered footing subjected to axial loading

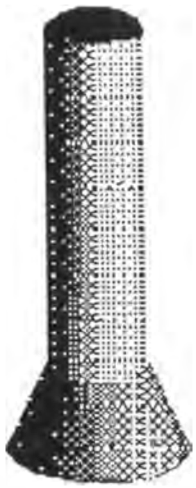


Fig 2.2 Cylinder

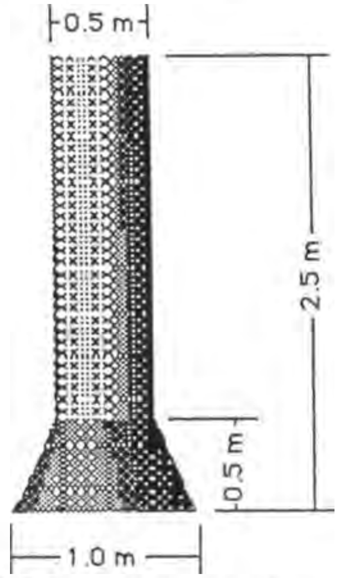


Fig 2.3 Cylinder in cross-section view

Figure 2.2 depicts a circular column resting on a tapered footing. The dimensions, properties, and boundary conditions are shown in Figure 2.3. We will make use of the axial symmetry. The column is represented as a rectangle rotated about the z-axis; the footing is represented as a trapezoid rotated about the z-axis. We assume that the column and footing are different, homogeneous materials. You are to find the displaced shape of the column and footing as well as stresses and strains. You could also specify point and body forces.

1. Start your computer and activate PCE.

• Perform the usual startup process.

• Select the subdirectory which contains PC-Elastic. (For example, if PCE is in a subdirectory named PCE, enter the command cd PCE <enter>.)

• Type PCE <enter> to activate the program. The title screen appears (Fig 2.4).

```
COOKE PUBLICATIONS                              SINGLE-USER LICENSE
                                                serial #:

                           PC-ELASTIC

                          VERSION 1.5

                     J.R.COOKE, D.C.DAVIS,
                E.T.SOBEL, R.S.GATES, R.PERL
                   Additional Programming
                          P.HUMMEL

                    Copyright 1986-1989
                 COOKE PUBLICATIONS, LTD.
                    All Rights Reserved

Student Version-300 DOF  Press F1 for additional information
        ENTER
Press <RETURN>[]_
```

Fig 2.4 Program credits and copyright notice

• Press F1 for additional information about the program.

```
Please send comments on PC-Elastic to:

                          J. Robert Cooke
                      Cooke Publications, Ltd.
                            PO Box 4448
                         Ithaca, NY 14852

If you have solved problems which you are willing to have described to
other PCE users, please send them to the above address.

Version: 1.5
Date: January 6,1989

       ENTER
Press <RETURN> to continue...
```

Fig 2.5 About PCE screen

• Press <return> to return to the title screen.

• Press <return> again to display the main menu which provides access to all segments of the program.

```
PC-ELASTIC                                              PROGRAM MENU

1. GEOMETRY
   Define Problem Geometry
2. MESH
   Generate & Modify Mesh
3. PREPROCESS
   Define Properties and Boundary Conditions
4. Solve
   Assemble Equations and Solve for Nodal Displacements
5. POSTPROCESS
   Solve for Element Strains and Stresses and Nodal Stresses
6. PLOT
   Plot Grid, Boundary Conditions, and Problem Results.
7. FILE MANAGEMENT
   Create or Select Project

0. STOP
                                  Enter
 ... Select by number <RETURN>[1]_
```

Fig 2.6 The main menu

You can begin problem formulation when the main menu appears. The phases of problem formulation, solution, and output are represented in the main menu by the steps which you execute sequentially. See Fig 2.1 also.

2. Select a project.

• Select 1 (Geometry) by pressing <Enter>.

• Supply a response to "Enter path for application [C:\PCE\]?" and press <Enter>. {If the data and PCE are in the same subdirectory, just press <Enter>.}

```
FILER   (7.50)                              APPLICATION SELECTION


The following applications are available:

1:  COLUMN                          2: PLATE

Choose an application number:[1]>_
```

Fig 2.7 Select a project

• Select the column demo (Column) by number and press <Enter>.

Note: You must keep all of the PCE files pertinent to a project in the same subdirectory.

If no applications were present, PCE routes you to the File Management module.

```
FILER  (7.50)                                    APPLICATION SELECTION

Application is: COLUMN

User Interaction Mode:

1. Maximal (no defaults)
2. Intermediate (some defaults)
3. Minimal (free run)
4. Demo with previous data

... Select by number <RETURN>[2]
```

Fig 2.8 Select user-interaction level

Note: This choice determines the level of detail available to you. Use 4 if this is your first encounter. Use 3 if you are an experienced user. Use 1 if you are exploring the algorithms in detail.

- Choose 4 to suppress all file saving.
- Press <Enter>. IMPORTANT.

> Important: In Demo Mode PCE suppresses all file saving; this allows you to explore various options at each stage of the analysis without corrupting the data files. *If you fail to select Demo Mode at this point, you may not be able to follow the instructions in the remainder of the following exercise.*
>
> Conversely, if you are in Demo Mode, you cannot solve an original project.

```
                         PC-ELASTIC

                      GEOMETRY (1.00)

ABSTRACT:

Discretization begins with the creation of curvilinear QUADRILATERAL
subregions which together describe the overall geometry.

Subsequently PCE will subdivide these input regions into a mesh of 3-
node triangles.

Press <RETURN>[]
         ENTER
```

Fig 2.9 The Geometry menu

Next you must specify the type of element. In this case, you choose from a two-dimensional triangle and a triangular annulus for axisymmetric problems.

```
 GEOMETRY (1.05)                                          MAIN MENU
                                   INITIALIZE COORDIATE TYPE

ELEMENT TYPE?

1. Two-Dimensional 3-Node Triangle
2. Axisymmetric 3-Node Triangle
                              ENTER
... Select by number <RETURN>[2]
```

Fig 2.10 The element type

Note: The default reflects the selected application.

• Press <Enter>.

Planar uses Cartesian coordinates (x,y) and assumes no change in the z-direction. Axisymmetric uses cylindrical coordinates (r,z) and assumes no change in geometry or constraints with respect to angular position about the z axis of symmetry.

```
GEOMETRY (1.05)                                                    MAIN MENU

--- POINT COORDINATE ENTRY ---
1. Input with Graphics
2. Input without Graphics

--- DATA REVIEW ---
3. List Input Data
4. Plot Input Data

0. None of the Above
                            Enter
... Select by number <RETURN>[1]
```

Fig 2.11 The Geometry menu

> **You** must assure a consistent set of units throughout the project!

Note: Since you are in Demo Mode, any changes you make will affect only this module, i.e., Geometry. You can explore without disturbing the tutorial.

3. Define mesh generating regions.

The geometry creation occurs in two phases: phase 1 entails creating an outline of the problem (points representing endpoints and midpoints of line segments comprising the problem). Phase 2 involves selecting these points in an appropriate manner to create regions. See the section on region creation in Chapter 3 of this manual for details. This process must anticipate the next mesh requirements, perhaps the most subjective and difficult step in PCE.

```
GEOMETRY (1.0)                                   DATA ENTRY WITH GRAPHICS

Is data stored in a disk file [Y]?
```

Fig 2.12 Retrieve existing data?

- Use the default—Y—since you are using a demo file.
- Observe the description of the data being retrieved and press <RETURN>.
- Accept the default axes endpoints by pressing <Enter> four times.

Note: If you had selected the plate example, the axes would have been called X and Y for the Cartesian coordinates [(0,500), (0,500)].

GEOMETRY (1.10) DATA ENTRY WITH GRAPHICS

-- SET AXIS LIMITS --

Minimum R [0.00]>
Maximum R [0.50]>

Minimum Z [0.00]>
Maximum Z [2.50]>

Fig 2.13 The axis endpoints

The eight points used to define a mesh generating region for the demo are immediately displayed.

GEOMETRY (1.20) COORDINATE ENTRY

INSTRUCTIONS:

Enter the EIGHT points which define each quadrilateral region. There must be FOUR vertices and ONE point between each set of TWO vertices.

Adjacent regions share the THREE points along the common side.

As a degenerate case, the sides may be straight lines or TWO connected straight lines to form a single straight side of a triangular region.

Press <~~RETURN~~ ENTER> to continue...

Fig 2.14 Geometry instructions

GEOMETRY (1.20) COORDINATE ENTRY

ENTER QUADRILATERAL POINTS-
OPTIONS: ENTER, DELETE, QUIT, BRANCH
(E D Q < >) [E]

Fig 2.15 The region generation points

For reference purposes, here are the tools you would use to create regions:

Note: In graphics mode, do not press <Enter>, the commands require only a keystroke.

Press "E" to gain access to point entry.

Press "D" to delete points you select using the cursor and press "D" to delete.

Press "Q" to quit entry of points.

"<" and ">" allow you to branch backward and forward, skipping the pending input. ~~[Note: These have not been reactivated yet in this version.]~~

If you press "E", you can add single points (P), points on a line (L), points on an arc (A), as well as quit (Q) and branching.

P : Supply the two coordinates of a point.
L : Supply the starting point coordinates, ending point coordinates, the number of intermediate colinear points you want to generate by interpolation, whether the points are to be evenly spaced, and if not, their fractional distance from the starting point.
A : Supply the coordinates of the center of the arc, the coordinates of the starting point of the circular arc, the total angle of arc in counterclockwise degrees, the number of intermediate points you want to generate by interpolation if evenly spaced, and if not, the fractional distance from the starting point.
Q : Select after you have placed all the generating points.

• Select "Q" to leave this section.

GEOMETRY (1.25) REGION DEFINITIONS

INSTRUCTIONS:

Each region is defined by a COUNTERCLOCKWISE sequence of 8 points.
Use arrows to move the cursor to the desired point. Then select the
point by pressing <S>. The first point must be a VERTEX.

Each region will be numbered at its centroid when completed.

Enter
Press <~~RETURN~~> to continue...

Fig 2.16 Instructions for forming regions

Briefly: All regions are comprised of four sides, each defined by two endpoints and one intermediate point, to which you fit a quadratic curve. If the midpoint is collinear with the endpoints, PCE creates a straight line. The total number of points in a region is, therefore, eight. To select a region, move the cursor to a ***vertex*** *of* *your proposed region. Select the vertex, move the cursor to the closest counterclock-wise point in the region, and select again. Repeat this until you have selected all eight points.*

PC-Elastic now slowly draws the region, verifying your choice. Notice in Fig 2.16 that if you use multiple regions, all three points on a common side must be shared by adjacent regions to assure a common boundary!

Note: Number the regions consecutively.

DEFINE REGION NUMBER-
REGION = [1]>

- Press <Enter> to accept the default of 1.

- Connect the points to define the regions.

Note: The order of points located by pressing the arrow keys is probably not in the sequence to form the region(s) in counterclockwise order. Remember to select eight in counterclockwise order starting at a vertex.

After you have selected exactly eight:

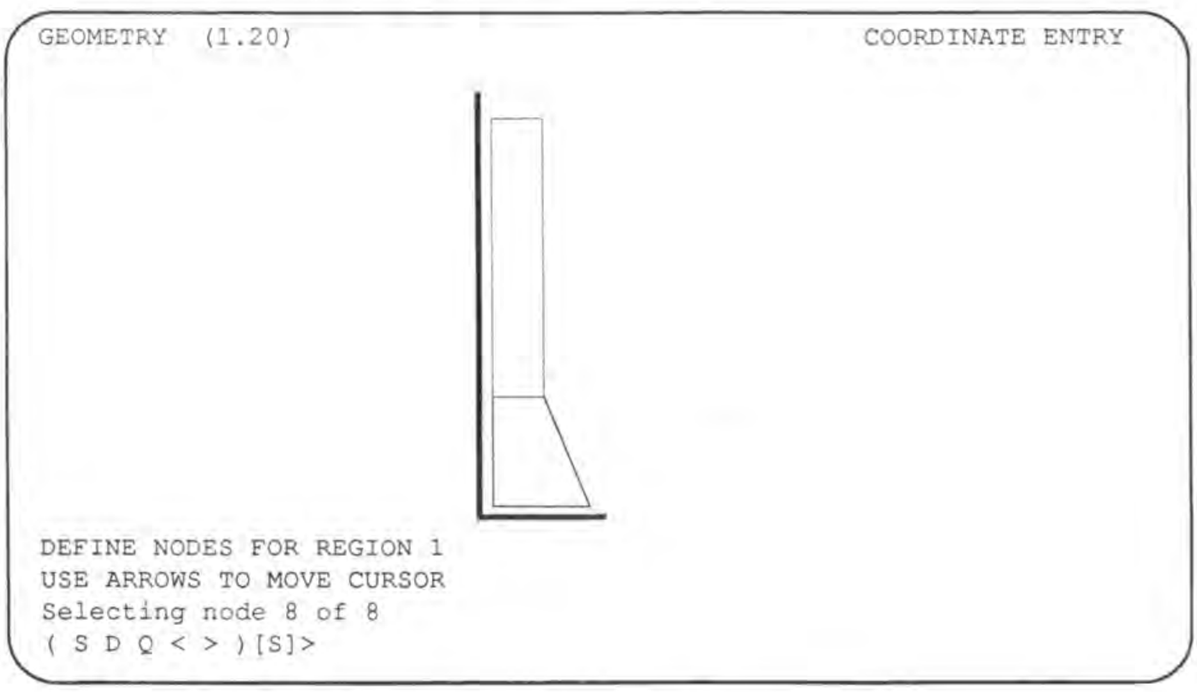


Fig 2.17 Verify counterclockwise region formation

- Press "Q" to quit selection.

Watch the region being formed by PCE to assure that you have defined the region in a counterclockwise order.

Is this OK?
Are the 8 nodes defined in a COUNTERCLOCKWISE sequence? [N]>

- Press "Y" and <Enter> if you traced the region in a counterclockwise order.

- DEFINE ANOTHER REGION [N]?

Type "N" after you have defined the second (and last) region.

PCE notifies you of the number of regions you have defined.

When you have completed exploring this module:

- Press <Enter> twice to return to the Geometry menu.

> Note: To avoid corrupting your files, never simply turn the power off in the midst of a project.

If you were not in Demo Mode, PCE would ~~ask you if you wanted to~~ have saved the data. to the DISK

From Fig 2.11

- Select items 3 and 4 to review your results.

- Select the last item of the menu to leave this module.

The final module menu allows you:

1. to proceed to the next logical step ("Mesh")
2. to return to the main menu
3. to change your mind and remain in this module

0. to terminate the session altogether

- Select 1 and press <Enter>.

4. Create the mesh.

You can enter the mesh module from the main menu (Fig 2.6) or as a continuation from the Geometry module above.

PC-ELASTIC

MESH (2.00)

ABSTRACT:

A mesh of 3-node triangular elements is generated for the problem domain. Next, nodes are renumbered to reduce the bandwidth, and unique lines and nodes on boundaries are identified automatically.

Press ~~<RETURN>~~ <ENTER> to continue

Fig 2.18 Mesh module entry

The mesh module has two phases. The automatic mesh generator 1) takes each properly defined region and breaks it into triangular areas referred to as "elements" (Generate Mesh) and 2) automatically renumbers the element vertices, referred to as nodes, to improve computational efficiency (Other Calculations).

- Press <Enter>.

4.1 Generate a mesh.

MESH (1.05) MAIN MENU

-- MESH DEFINITION --
1. Do complete Mesh Generation (2-4)

2. Define Mesh Elements & Nodes
3. Renumber Nodes to Reduce Bandwidth
4. Sort Unique Lines & Boundary Nodes

-- MESH EXAMINATION --
5. List Element Nodal Coordinates
6. List Line & Boundary Nodes
7. Plot Entire Mesh

0. None of the Above

... Select by number <~~RETURN~~ ENTER> [1]

Fig 2.19 Mesh module menu

Your first task is to specify the mesh refinement appropriate to this problem. In general, a mesh with a larger number of smaller elements produces better numerical results than a mesh with fewer, but larger, elements. On the other hand, a larger number of elements leads to a larger number of unknowns and, therefore, to a larger number of equations. This, in turn, requires more computer memory and requires increased computational time. The verification studies in Chapter 3 provide some guidance on this important, subjective process. In more complicated problems you reduce the element size in the vicinity of greatest anticipated change in the dependent variable, i.e., nodal displacement. Conversely, you can make the elements larger where the expected change is smaller.

Selection of menu option 1 causes PCE to execute options 2, 3, and 4 without further intervention. In this case you want to examine the steps only superficially.

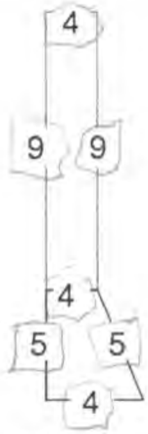
• Select 1 and press <Enter>.

```
MESH      (2.77)                                   DEFINE NODES PER SIDE

INSTRUCTIONS:

The number of triangular element nodes on each quadrilateral region
side (including both end nodes) must be specified.  The minimum number
is TWO, which includes the endpoints.  Use arrows to move the cursor
to the desired side; select by pressing <S>.

Because elements must be connected at their nodes, opposite sides of a
region must have the same number of element nodes.  Therefore,
adjacent regions and their opposite sides are matched automatically.

Enter the number of nodes for each marked side.  Change any entries by
reselecting the side.

Press <RETURN> to continue...
```

Fig 2.20 Mesh module instructions

• The number of nodes along a side of the mesh generating region has already been specified.

```
MESH     (2.27)                                         DEFINE NODES PER SIDE
```




```
DEFINE NUMBER OF NODES PER SIDE-
  OPTIONS: SELECT SIDE, QUIT, BRANCH
( S Q < > ) [S]>
```

Fig 2.21 Nodes per side

- Press "Q" to resume.

PCE calculates the number of degrees of freedom for the mesh and asks:

"Your problem has 104 Degrees of Freedom.
Is this acceptable? [Y]".

- Press <Enter>.

Note: The student version is arbitrarily limited to 300 DOF. A "N" response would return you to the nodes per side screen.

```
RUN OPTIONS:
1. W/O USER INTERACTION
    (USING ALL DEFAULTS)
2. WITH USER INTERACTION

...Select by number <RETURN>[1]>
```

- Press <Enter>.

For this demo complete the calculations for node coordinates and draw the mesh without interruption. PCE reduces the bandwidth, finds unique plot lines, and identifies boundaries. PCE displays the number of elements (72), number of nodes (529), and bandwidth (42 reduced to 81). 7

PCE generates automatically the mesh for each of the mesh generating regions, using a scale to make the largest possible image.

Finally, PCE creates a composite mesh using a common scale.

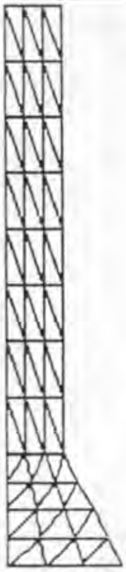


Fig 2.22 Composite mesh

Note: The number of elements is the number of triangles displayed. The number of nodes is the number of unique vertex points. Recall that PCE calculates values of the two dependent variables at each of these vertices. Remember that you will assign some of these values as boundary conditions.

4.2 Other calculations.

While still operating with minimal user interaction, PCE next attempts to reduce the bandwidth. See the chapter on computational details for an explanation of bandwidth. For present purposes just note that a small bandwidth is preferable because less RAM and disk storage are needed, and the computation time is less. Bandwidth depends upon the project geometry and, therefore, the mesh configuration. Bandwidth also reflects the node numbering scheme used. In the next section PCE provides an algorithm to attempt various renumbering schemes automatically.

The bandwidth reduction option uses the Collins algorithm to renumber the nodes, to reduce the bandwidth, or to use the existing node numbering if no reduction is feasible. Bandwidth reduction is especially useful with multiple regions and mesh refinement situations. See Chapter 4 on computational details for a discussion of this topic.

If PCE cannot reduce the bandwidth, no change is made. Otherwise, PCE uses the new numbering.

In order to identify the boundaries and to avoid redundant plotting of common boundaries of interior triangles, PCE next examines the mesh lines in the next step.

Elements are joined only at their nodes. Therefore, when you specify the number of nodes you want to use on a side of a mesh generating region, PCE enforces the constraint that opposite sides of a region have the same number of nodes and that regions sharing a common boundary have the same number of nodes on the boundary. PCE calculates the nodal coordinates and element connectivity.

PCE automatically goes through several calculations to remove duplicate plot lines (e.g., a side common to two adjacent elements), to find boundary nodes and elements, and to determine the connectivity of elements.

When PCE has completed these steps, you are once again presented the Mesh main menu (Fig 2.19).

Note: Had you elected to examine the individual steps (2, 3, & 4 of the Mesh main menu), you could have examined the processes in much greater detail. In fact, you can examine those steps now or later. In addition to reviewing the intermediate *steps, you also can modify the automatically generated mesh—adding nodes by subdividing elements, changing the choice of the diagonal used to subdivide rectangles, and repositioning selected nodes.*

You can list the numerical data associated with the mesh—coordinates of nodes, node numbers of individual elements, and boundaries. You can visually review the node and element numbering patterns on a plot of the generated mesh.

From the Mesh main menu you can move to the next module.

• Select 0 to leave Mesh generation.

The exit menu allows you to:

```
1. Proceed to the next module, "PREPROCESS"
2. Return to the Program Menu
3. Change your mind and stay in MESH

0. Terminate PCE
```

If you were in Demo Mode, PCE would not save the data files. See the File Management module in the Reference chapter and the Appendix for a description of these files. *Note:* You would select "0" to terminate the session.

> Note: Never simply turn the power off in the midst of a project.
> Always exit via the Quit command of the File menu.

However, to continue with the next module:

• Select 1 by pressing <Enter>.

5. Assign properties and boundary conditions.

PC-ELASTIC

PREPROCESS (3.00)

ABSTRACT:

Specify MATERIAL PROPERTIES and BOUNDARY CONDITIONS, in that order. The required properties are the ELASTIC MODULUS, POISSON'S RATIO, THICKNESS, TEMPERATURE, and THERMAL EXPANSION COEFFICIENT for each element. Boundary conditions include POINT FORCE, FIXED NODAL DISPLACEMENT, and SURFACE STRESS.

Press ~~<RETURN>[]~~ <ENTER> to continue ...

Fig 2.23 Preprocess abstract

• Press <Enter>.

PCE displays the problem keyword and description.

• Press <Enter>.

need Return

PCE retrieves the data required for this module that you created in the earlier steps and displays the Preprocess menu (Fig 2.24).

• Press <Enter>

PREPROCESS (3.00) MAIN MENU

--- DATA INPUT ---
1. Input Material Properties and Thickness
2. Input ~~Potential, Flux, and~~ Source Boundary Conditions
Force, Displacement, Stress Boundary Conditions
--- DATA EXAMINATION ---
3. List Properties
4. Plot Properties
5. List Input Boundary Conditions
6. List Equivalent Boundary Conditions
7. Plot Input Boundary Conditions
8. Plot Equivalent Boundary Conditions

0. None of the Above

... Select by number <RETURN>[1]

Fig 2.24 Preprocess module main menu

You can enter the properties module from the main menu (Fig 2.6) or as a continuation from the previous mesh module.

This module handles two major tasks: 1) entering and editing element properties and 2) entering and editing conditions at boundary and interior nodes. Distributed body conditions use the same input structure.

5.1 Properties.

• Select 1 and press <Enter> to enter the boundary conditions.

PCE finds the element centroids and the boundary nodes.

"Are properties in a disk file [N]?" Y

• ~~Type Y and~~ press <Enter> to retrieve the necessary demo data files.

PROPERTY EDITING OPTIONS: (FIG 2.25) allows you to return to modify further or examine properties.

- Select 0 and press <Enter>.

PCE asks "Do you wish to proceed to definition of BOUNDARY CONDITIONS [Y]?>"

- Press <Enter> to continue.

```
PREPROCESS (3.18)                                        EDIT PROPERTIES

PROPERTY EDITING OPTIONS:

1. ELASTIC MODULUS
2. POISSON'S RATIO
3. TEMPERATURE
4. THERMAL EXPANSION COEFFICIENT

0. NONE

... Select by number <RETURN>[1]
```

Fig 2.25 Preprocess module main menu

For axisymmetric problems, the following questions about thickness do not appear. *NOTE: For two-dimensional problems, you must specify the thickness. Each element can have a different thickness.*

If you had selected variation by input region or by element, then you would be required to supply that additional information (Figs 2.26 and 2.27).

```
PREPROCESS (3.18)                                        EDIT PROPERTIES

                        PROPERTY: ELASTIC MODULUS

ELASTIC MODULUS IS UNIFORM BY:

1. ENTIRE BODY
2. INPUT REGION
3. ELEMENT

... Select by number <RETURN>[1]
```

Fig 2.26 Edit menu

move to after properties - a little confusing

```
PREPROCESS (3.18)                                        EDIT PROPERTIES

                    PROPERTY: ELASTIC MODULUS

ELASTIC MODULUS is uniform in the body.

What is the ELASTIC MODULUS value [18200000.00]>
```

Fig 2.27 Edit value

5.2 Boundary conditions.

PCE asks "`Are BOUNDARY CONDITIONS in a disk file [N]?`" Y

• Type Y and press <Enter> to retrieve the demo data files.

```
PREPROCESS (3.20)                         ENTRY OF BOUNDARY CONDITIONS

INSTRUCTIONS:

Boundary conditions may be DEFINED or DELETED at a node or for a range
of nodes and for each component direction.

First define the details of the B.C.; then specify the range.

Ranges are COUNTERCLOCKWISE (viewed from the body interior) from
starting point to ending point.

Press <RETURN> to continue...
```

Fig 2.28 Boundary conditions instructions

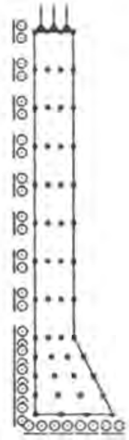
Press <Enter> for the next screen (Fig 2.29).

Then use the initial letter of the command.

E: enter boundary conditions mode. The boundary condition choices are as follows:

`1. POINT FORCE   2. DISPLACEMENT  3. SURFACE STRESS 4. BODY FORCE`

PREPROCESS (3.20) INPUT BOUNDARY CONDITIONS



DISPLACEMENT POINT FORCE SURFACE STRESS
DEFINE BOUNDARY CONDITIONS-
OPTIONS: ENTER, DELETE, QUIT, BRANCH
(E D Q < > [E]>

Fig 2.29 Boundary conditions

If you select 1, you can assign values for point force at interior nodes or on the boundaries. First assign a coordinate direction, then assign the value, and then specify the nodal points—single points or a contiguous range of points. For a range of points, remember to observe the counterclockwise convention.

If you select 2, you can assign values for displacement at interior nodes or on the boundaries. First assign a coordinate direction, then assign the value, and then specify the nodal points—single points or a contiguous range of points. For a range of points, remember to observe the counterclockwise convention.

If you select 3, you can assign values for surface stress in a coordinate direction or in the normal and tangential directions, then assign the value, and then specify the nodal points—single points or a continuous range of points. For a range of points, remember to observe the counterclockwise convention.

If you select 4, you can assign body force for a coordinate direction, then assign the value, and then specify the nodal points—single points or a contiguous range of points. For a range of points, remember to observe the counterclockwise convention.

- Select "Q" as often as necessary to quit.
- Indicate whether additional changes are required before leaving this module.
- Press <Enter> to see a summary for this section.
- Press <Enter> again to return to the Preprocess main menu (Fig 2.24).

• Select 0 and press <enter> to leave this module.

```
PREPROCESS (3.09)                                                  EXIT


1. Proceed to 'SOLVE'
2. Exit to Program Menu
3. Don't Exit; Remain in 'PREP'

0. Stop

... Select by number <RETURN> [1]>
```

Fig 2.30 Preprocess exit menu

• Select 1 (Fig 2.30) to advance to the SOLVE module.

6. Form and solve the equations.

You can enter the solve module from the main menu (Fig 2.6) or as a continuation from the properties module (Fig 2.30).

```
                                PC-ELASTIC

                               SOLVE (4.00)

ABSTRACT:

Finite element equations are defined for each element, assembled into
a global matrix equation, modified by boundary conditions, and solved
for nodal displacements.

Press <RETURN>[] to continue...
```

Fig 2.31 Solve module abstract

• Press <Enter> to continue.

After PCE reminds you of the project selected:

• Press <Enter> to continue and PCE retrieves the required data files.

```
SOLVE (4.05)                                          MAIN MENU

-- FORM AND SOLVE ALL EQUATIONS --
1. Do Complete Problem Solution (Menu Options 2-5)

-- FORM SYSTEM EQUATIONS --
2. Assemble Global Force and Stiffness Matrices
3. Combine External Force Boundary Conditions
4. Apply Displacement Boundary Conditions

-- SOLVE SYSTEM OF EQUATIONS --
5. Solve for Nodal Displacements

-- LIST RESULTS --
6. List Nodal Displacements

0. None of the Above

... Select by number <RETURN>[1]
```

Fig 2.32 Solve main menu

For this demo:

• Select 1 and press <Enter> to complete the formulation of the global system of equations, apply the boundary conditions, and find the nodal values.

```
SOLVE (4.05)                       COMPLETE PROBLEM FORMULATION

OPTIONS:
1. Run WITHOUT user interaction (using all defaults)
2. Run WITH user interaction

... Select by number <RETURN> [1]>
```

Fig 2.33 Solve interaction options

• Press <Enter> to run without intervention.

Later you can examine the intermediate calculations and list the numerical results.

To leave this module (Fig 2.32):

• Select 0 and press <Enter>.

```
SOLVE  (4.00)                                                    EXIT

1. Proceed to 'POSTPROCESS'
2. Exit to Program Menu
3. Don't Exit; Remain in 'SOLVE'

0. Stop
                               ENTER
... Select by number <RETURN> [1]>
```

Fig 2.34 Solve exit menu

• Select 1 and press <Enter>.

```
                                 PC-ELASTIC

                             POSTPROCESS (5.00)

ABSTRACT:

Node displacements determined in 'SOLVE' are used to define element
strains, element stresses, and node stresses.  Stresses are defined in
coordinate and principal directions.

In addition node reactions and new node positions are determined.
          ENTER
Press <RETURN> to continue...
```

Fig 2.35 Postprocess abstract

• Press <Enter> to continue. PCE again reminds you of the project selected.

• Press <Enter> to continue. PCE retrieves the required data files prepared by previous modules.

```
POSTPROCESS  (5.00)                                                MAIN MENU

-- SOLVE ALL EQUATIONS --
1. Do all calculations (Menu Options 2-6)

-- SOLVE FOR STRAINS AND STRESSES --
2. Element Strains and Stresses
3. Element Principal Stresses
4. Nodal Principal Stresses

-- OTHER CALCULATIONS --
5. New Node Positions
6. Node Reactions

-- LIST RESULTS --
7. List Results

0. None of the Above

... Select by number <RETURN>[1]
```

Fig 2.36 Postprocess main menu

As with the Solve menu, the first menu item allows you to complete all steps with one selection. Later, you will use options 2-6 to examine several levels of the intermediate steps in the calculations. For now:

• Select 1 and press <Enter>.

Then elect to "`Run WITHOUT user interaction (using all defaults)`".

• Select 1 and press <Enter>.

The calculations proceed without intervention—PCE reads certain files, and creates and saves others. PCE returns you to the Postprocess main menu (Fig 2.36).

As with the solve module, examine the numerical results from within this module—now or later. In the next module the computed results will be presented in graphical form.

• Select 0 and press <Enter>.

```
POSTPROCESS (5.00)                                                  EXIT


1. Proceed to 'PLOT'
2. Exit to Program Menu
3. Don't Exit; Remain in 'POSTPROCESS'

0. Stop

... Select by number <RETURN> [1]>
```

Fig 2.37 Exit Postprocess

- Press <Enter> to advance to PLOT, the default.

7. Plot.

You can enter the plot module from the main menu (Fig 2.6) or as a continuation from the postprocess module (Fig 2.35).

```
                              PC-ELASTIC

                              PLOT (6.00)

ABSTRACT:

Prepare finished plots of MESH, BOUNDARY CONDITIONS, or RESULTS.  Any
plot may be labeled and printed.  Zones may be selected to enlarge
parts of interest.

Press <RETURN>[]
```

Fig 2.38 Plot module abstract

- Press <Enter> to continue.

has name, serial number
and year on top

After PCE displays a reminder of the project selected:

• Press <Enter> to continue. PCE retrieves the required data files.

```
PLOT   (6.06)                                                   MAIN MENU

-- PLOT FORMULATION --
1. Plot Generated Mesh
2. Plot Boundary Conditions


-- PLOT RESULTS --
3. Plot Nodal Displacements
4. Plot Element Strains
5. Plot Element Stresses
6. Plot Nodal Stresses

0. None of the Above

... Select by number <RETURN>[1]
```

Fig 2.39 Plot main menu

Examine each of the plot options. In this first glimpse of the plots, ignore the advanced features such as labeling and zooming.

7.1 Generated mesh.

• Select 1 and press <Enter> to plot the mesh.

Plot Generated Mesh

```
PLOT   (6.01)                                                   EXIT

Whole body will be plotted; then ZONE plotting is offered.  After each
plot you will choose from the following labeling options:

 NO LABELS
 LABEL ELEMENTS
 LABEL NODES
 LABEL CONTOUR LINES (ONLY FOR MENU OPTION - NODE STRESSES)
 ADD ANY LABEL   LABEL ELEMENT VALUES

Note: Labels falling on lines or shaded regions will appear fuzzy.  To
clear this up, choose the redrawing option.

Press <ENTER> to continue...
```

Fig 2.40 Plot options

• Press <Enter>.

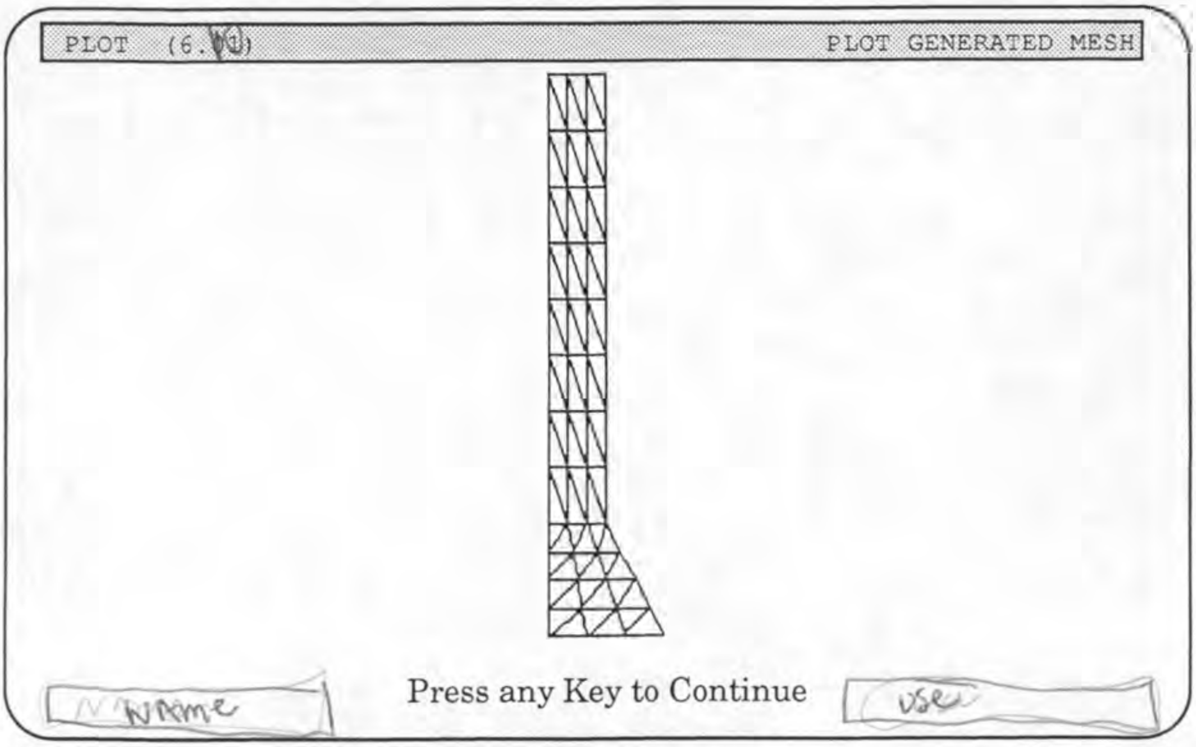


Fig 2.41 Mesh

- Press any key to continue.

Let's not examine the labeling capabilities in this quick overview.

- Press <Enter> to accept the "N" default.

```
Label this plot [N}?
```

Likewise, let's not examine the zoom capability yet.

- Press <Enter> to accept the default and return to the plot menu (Fig 2.39).

```
PLOT DESTINATION OPTIONS:
1. Proceed with zone plotting

0. End this plot

...Select by number <Enter>[0]
```

Fig 2.42 Plot windows

7.2 Boundary conditions.

The *current* default at the Plot menu is now 2.

- Type 2 and press <Enter>.

PCE retrieves the required data files and presents the introductory list of options (Fig 2.40).

• Press <Enter> to resume.

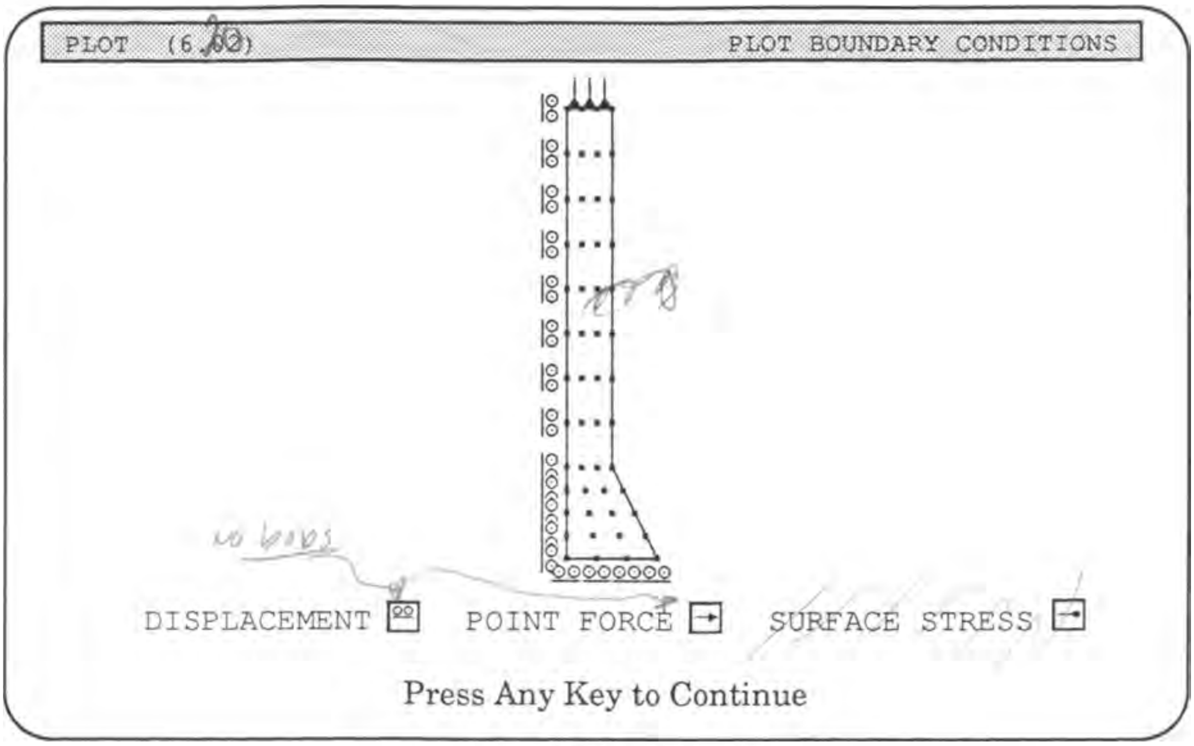


Fig 2.43 Boundary conditions

Again, let's not examine the labeling and zoom capabilities yet.

• Press <Enter> twice (Fig 2.43) to skip the advanced features and to return to the Plot main menu (Fig 2.39).

7.3 Nodal displacements.

Option 3 produces a plot of the principal results—the displaced mesh.

• Type 3 and press <Enter>.

Again, use the default.

• Accept the default by pressing <Enter>.

```
Mesh Options:

1. Entire Mesh
2. Only boundary lines

0. None of these

...Select by number <Enter>[1]
```

Fig 2.44a Plot specifications

In this window you can accept the default values again.

• Press <Enter> to accept the default values.

```
Line Type Options:

1. Original: dashed
   Displaced: solid
2. Original: solid
   Displaced: dashed

0. None of these

... Select by number <RETURN>[1]>
```

Fig 2.44b Plot specifications

A reminder of the plot options (Fig 2.42) appears.

• Press <Enter> to resume.

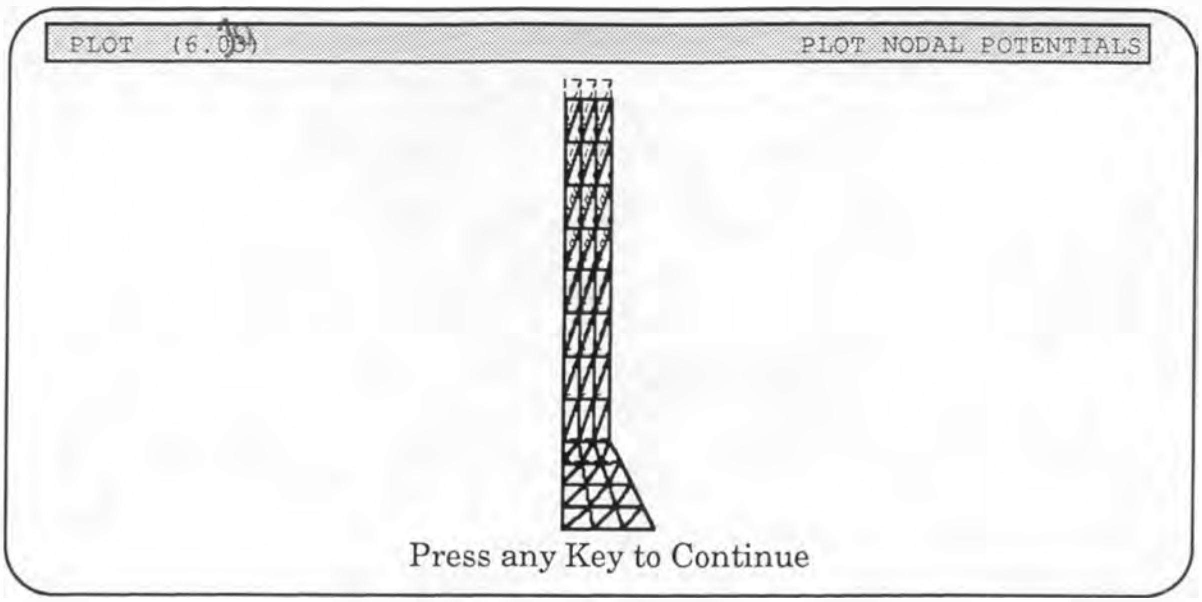


Fig 2.45 Nodal displacements

• Press <Enter> to continue.

Again, let's not examine the labeling and zoom capabilities.

• Press <Enter> twice (Fig 2.42) to skip the advanced features and to return to the Plot main menu (Fig 2.39).

7.4 Element strains.

A plot of the strain for each element provides an easily visualized characterization of the conditions. Shading density conveys a feel for the variation throughout the body.

• Type 4 and press <Enter>.

• Type 3 and press <Enter> to plot the z-component of element strain.

PCE displays the data minimum and maximum and the default increment size.

• Press <Enter> to accept the default.

```
Type: COORDINATE-DIRECTION

Component Options:
1. R-component
2. THETA-component
3. Z-component
4. R-Z shear

0. None of these

... Select by number <RETURN>[1]>
```

Fig 2.46 Element strain component

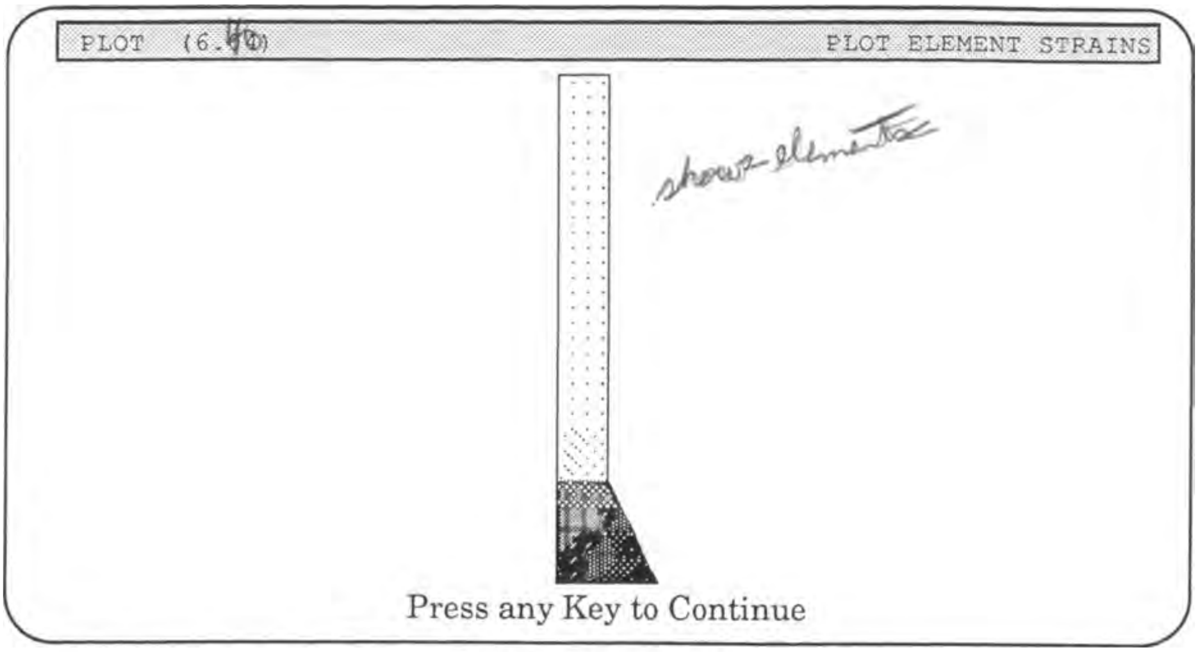


Fig 2.47 Element strain

• Press <Enter> to continue.

Again, let's not examine the labeling and zoom capabilities. ??

• Press <Enter> twice (Fig 2.42) to skip the advanced features and to return to the component options (Fig 2.46).

• Type 0 and press <Enter> to return to the plot menu (Fig 2.39).

7.5 Element stresses.

PCE earlier computed the element stresses, and you can display the results as a shaded plot of constant stress components—r,theta, z, r-z components.

• Type 5 and press <Enter>.

The component options are the same as for element strains (Fig 2.46).

Select the z-component (Fig 2.46).

• Type 3 and press <Enter>.

In this window you can accept the default values or request a window in which you can specify the values.

• Press <Enter> to accept the default values.

A reminder of the plot options (Fig 2.42) appears.

```
Specify plot range and increment...
Data minimum: -1.10814904305331E+0002
Data maximum: -1.32041129020181E+0001

Default is full range with 8 increments
of 1.22013489254141+0001

Is this OK[Y]?
```

Fig 2.48 Plot specifications

• Press <Enter> to have PCE present the plot.

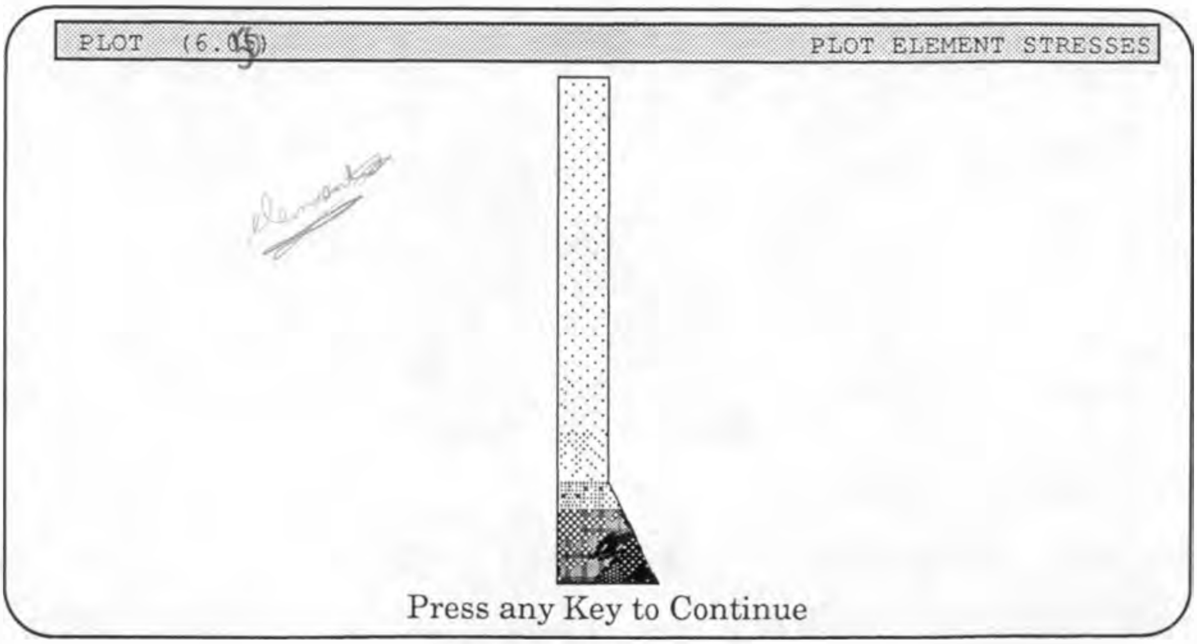


Fig 2.49 Element stress

• Press <Enter> twice (Fig 2.42) to skip the advanced features and to return to the component options (Fig 2.46).

• Type 0 and press <Enter> to return to the Plot menu (Fig 2.39).

7.6 Nodal stresses.

A contour plot of the nodal stresses—coordinate direction and principal—completes the visualization of the potential.

• Type 6 and press <Enter>.

Next you must select a stress you want to plot (Fig 2.50). For illustrative purposes use the default.

• Press <Enter>.

```
Stress Options:

1. COORDINATE-DIRECTION
2. PRINCIPAL

0. None of these

...Select by number <Enter>[1]>
```

Fig 2.50 Stress options

Type 3 and press <Enter>.

```
Type: COORDINATE-DIRECTION

Component Options:
1. R-component
2. THETA-component
3. Z-component
4. R-Z shear

0. None of these

... Select by number <RETURN>[1]>
```

Fig 2.51 Component options

Type 1 and press <Enter> to draw without the mesh.

```
Type: COORDINATE-DIRECTION

Component: Z

Contour Line Options:

1. Plot only contour lines
2. Plot contour lines with dashed mesh

... Select by number <RETURN>[1]>
```

Fig 2.52 Contour line options

PCE retrieves the required data files and presents a summary of values.

• Press <Enter> to accept the default values.

```
Specify plot range and increment...
Data minimum: -1.05366868808264E+0002
Data maximum: -5.10211322194013E+0000

Default is full range with 10 incre-
ments of 1.00264755586324E+0001

Is this OK[Y]?
```

Fig 2.53 Contour specifications

A reminder of the plot options (Fig 2.40) appears.

• Press <Enter> to have PCE present the plot.

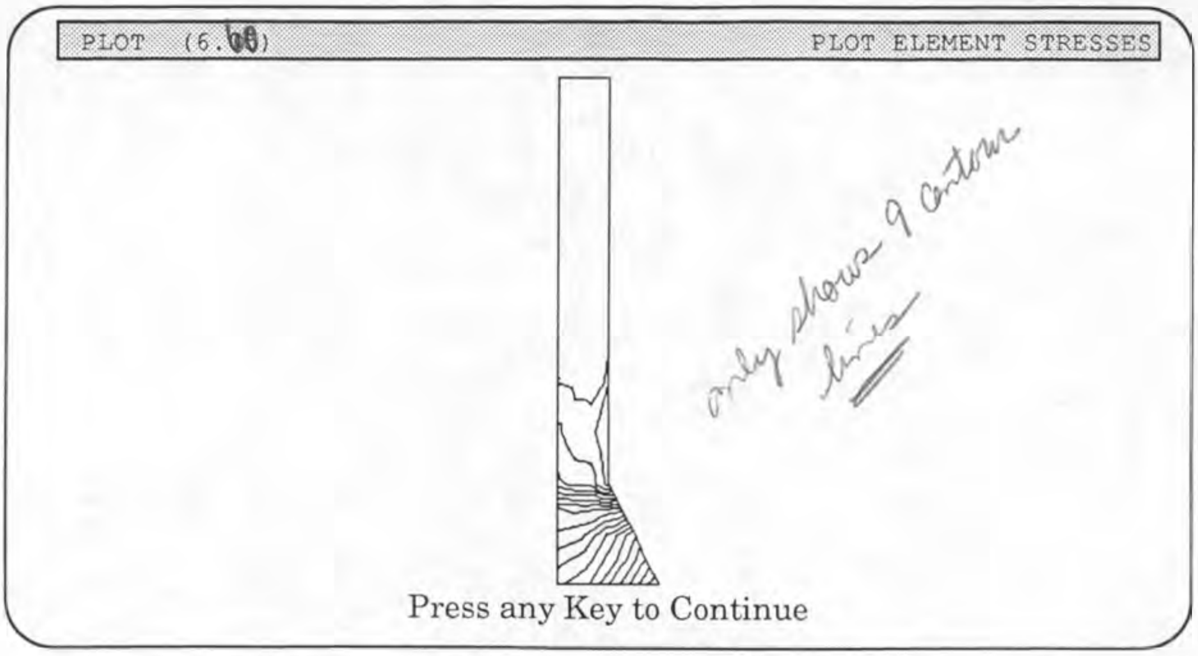


Fig 2.54 Contour plot

• Press <Enter> twice (Fig 2.42) to return to the component option.

• Type 0 and press <Enter> to return to the Plot main menu (Fig 2.39).

To leave this module:

• Type 0 and press <Enter>.

```
PLOT (6.0?)                                                    EXIT


1. Exit to Main Menu
2. Don't Exit; Remain in 'PLOT'

0. Stop

... Select by number <RETURN> [1]>
```

Fig 2.55 Plot exit menu

• Select 1 and press <Enter> to return to the main menu (Fig 2.8).

8. File management.

You can enter the plot module from the main menu (Fig 2.8) or as a continuation from the plot module (Fig 2.41).

• Type 7 and press <Enter>.

```
                              PC-ELASTIC

                            FILE MANAGEMENT

ABSTRACT:

Management of data files is controlled by a master file with the ex-
tension PCE.  You must initialize this file with the problem name and
problem description in order to begin an analysis.

Press <RETURN>[]
```

Fig 2.56 File management abstract

• Press <Enter> to continue.

The FILER MAIN MENU appears.

```
FILER (7.00)                                              MAIN MENU

1. Define problem and prepare master file
2. List Problem Keyword and Description
3. List all records with description and status
4. List contents of data files referenced by records
5. Duplicate an existing application
6. Choose an existing application
7. Change User Interaction Mode

0. None of the Above

... Select by number <RETURN>[1]
```

Fig 2.57 Filer main menu

Examine each of the Filer options. In this first glimpse of the plots, ignore the first option— "Define problem and prepare master file". For completeness the steps are given anyway.

8.1. Define problem and prepare master file. *(Read, but skip to option 2)*

• Select 1 and press <Enter> to begin project definition.

```
Enter path for application [C:\PCE\]?
```

Fig 2.58 Path (including disk)

• Press <Enter> to create the project files in the current subdirectory. Otherwise, supply the desired subdirectory before pressing <Enter>.

```
Define the following:
  Problem Keyword (1 to 8 characters)>
```

Fig 2.59 Project name

• Supply up to eight characters to serve as the file name for the project and press <Enter>.

Blanks are not allowed! Follow the DOS conventions. You will be warned if you attempt to use an existing file name. Each of the data files associated with this project has this common name and a unique extension.

```
Problem description
   (1 to 50 characters)>
```

Fig 2.60 Project description

• Type the description and press <Enter>.

This text serves only as a reminder to you. Normally, you record details of the variation of the problem under study, e.g., mesh refinement, change in properties, etc.

PCE creates a file having the keyword as the name and PCE as the extension.

> NOTE: ALL DATA FILES MUST BE KEPT IN THE SAME SUBDIRECTORY. THE NAMES AND STATUS OF ALL DATA FILES ARE MAINTAINED IN THIS FILE SO DON'T ARBITRARILY REMOVE OR MODIFY ANY OF THE 24 FILES.

```
User Interaction Mode:

1. Maximal (no defaults)
2. Intermediate (some defaults)
3. Minimal (free run)

                            Go OK
...Select by number <Enter>
```

Fig 2.61 User interaction level

The user interaction level controls the level of detail presented to you during execution. Use 3 if you are an experienced user who wishes to be asked for minimal information to solve a well-posed problem. Use 1 if you wish an opportunity to explore the deepest level of the algorithm, e.g., if you are specifically examining the algorithms being used. Level 2 is intermediate to the extremes.

• Select 2 and press <Enter> to return to the Filer main menu (Fig 2.57).

8. 2. List problem keyword and description.

This option echoes the problem keyword, coordinate type, and problem description of the current project.

Note: Use item 6 to change the choice of active application.

• Type 2 and press <Enter>.

```
PROBLEM KEYWORD: COLUMN  (axisymmetric)

PROBLEM DESCRIPTION: WOODEN COLUMN ON CONCRETE BASE
```

Fig 2.62 Problem description

• Press <Enter> to return to the Filer menu (Fig 2.57).

8.3. List all records with descriptions and status.

- Type 3 and press <Enter> twice to review the status of the data files.

```
FILER  (7.22)                                LIST FILE NAMES AND DESCRIPTIONS

Active file (BOLD)        Inactive file (NORMAL)
Letter in first column indicates data type: I = integer; R = real.

RECORD NAME         DESCRIPTION
---------------------------------------------------------------------------
R  1  C:COLUMN.XY   COORDINATES OF INPUT DATA POINTS
I  2  C:COLUMN.REG  NODE AND CONNECTIVITY DATA
I  3  C:COLUMN.EL   NODE AND REGION DATA FOR ELEMENTS
R  4  C:COLUMN.ND   COORDINATES AND ATTRIBUTES OF NODES
I  5  C:COLUMN.REL  RENUMBER ELEMENT NODE NUMBERS
R  6  C:COLUMN.RND  COORDINATES OF RENUMBERED NODES
I  7  C:COLUMN.RLN  RENUMBERED NODES FOR UNIQUE LINES
I  8  C:COLUMN.RBO  RENUMBERED BOUNDARY NODES AND ELEMENTS
R  9  C:COLUMN.PRP  ELASTIC AND THERMAL PROPERTIES
R 10  C:COLUMN.IBC  INPUT BOUNDARY CONDITION SPECIFICATIONS
R 11  C:COLUMN.BC   NODAL D.O.F. BOUNDARY CONDITIONS
R 12  C:COLUMN.IFO  INITIAL (THERMAL) FORCE VECTOR
R 13  C:COLUMN.IST  INITIAL GLOBAL STIFFNESS MATRIX
R 14  C:COLUMN.CFO  COMBINED INITIAL AND BOUNDARY FORCES

Press <Enter> to continue...
```

Fig 2.63a File status (part 1)

```
FILER  (7.22)                                LIST FILE NAMES AND DESCRIPTIONS

Active file (BOLD)        Inactive file (NORMAL)
Letter in first column indicates data type: I = integer; R = real.

RECORD NAME         DESCRIPTION
---------------------------------------------------------------------------
R 15  C:COLUMN.MFO  FORCES MODIFIED BY DISPLACEMENT B.C.
R 16  C:COLUMN.MST  STIFFNESS MODIFIED BY DISPLACEMENT B.C.
I 17  C:COLUMN.NDD  NODAL DISPLACEMENT MAGNITUDES
R 18  C:COLUMN.ESA  ELEMENT STRAINS IN COORDINATE DIRECTIONS
R 19  C:COLUMN.ESE  ELEMENT STRESSES IN COORDINATE DIRECTIONS
R 20  C:COLUMN.EPS  ELEMENT PRINCIPAL STRESSES AND DIRECTIONS
R 21  C:COLUMN.NDS  NODAL STRESSES IN COORDINATE DIRECTIONS
R 22  C:COLUMN.NPS  NODAL PRINCIPAL STRESSES AND DIRECTIONS
R 23  C:COLUMN.NEW  NEW COORDINATES OF NODES
R 24  C:COLUMN.REA  RESULTING REACTIONS AT NODES

Press <Enter> to continue...
```

Fig 2.63b File status (part 2)

- Press <Enter> to see part 2 and again to return to the Filer menu (Fig 2.57).

8.4. List contents of data files referenced by records.

- Type 4 and press <Enter> to select a file for examination.

```
Examine data file for which record (1-22): [1]>
```

Fig 2.64 File selection

- Type 1 and press <Enter>.

```
 FILER  (7.23)                                LIST DATA FILE CONTENTS

Filename:            C:COLUMN.XY
File description:    COORDINATES OF INPUT DATA POINTS
Rows represent       POINTS
Columns represent    DIMENSIONS
-----------------------------------------------------------------------
ROW  1:   0.00000000   0.00000000
ROW  2:   0.00000000   0.25000000
.........
ROW 12:   0.00000000   0.50000000

Press <Enter> to continue...
```

Fig 2.65 File 1 contents (partial)

Press <Enter> twice to resume.

```
Examine other data [Y]?
```

Fig 2.66 Continuation

- Type "Y" to return to Fig 2.64 or "N" to return to the Filer menu.

Note: You can examine each of the 24 files. Refer to the Appendix for a discussion of the file structure, should you wish to access the files using your own program.

8.5. Duplicate an existing file.

Use this option to duplicate the input files and the master file so you can perform a variation on the same project. You need to apply only your changes, not all the data for the entire project.

• Type 5 and press <Enter>.

```
Enter path for application [C:\PCE\]?
```

Fig 2.67 Path (including disk)

• Press <Enter> to create the project files in the current subdirectory. Otherwise, supply the desired path before pressing <Enter>.

PCE supplies a list of all projects contained in the subdirectory you specified.

```
The following applications are available:

1. COLUMN                2. PLATE

Choose an application by number:[1]>
```

Fig 2.68 Project selection

• Type the number and press <Enter>.

```
User Interaction Mode:

1. Maximal (no defaults)
2. Intermediate (some defaults)
3. Minimal (free run)
4. Demo with previous data

...Select by number <Enter>
```

Fig 2.69 User level

The user levels are the same as in Fig 2.61; but Demo Mode has been added. Use Demo Mode if you wish to suppress file saving. This option is useful if you are giving a class demonstration where you wish to explore various alternatives without risk to your data files. Otherwise, PCE designates dependent files as inactive and, therefore, unusable if you change values. This feature provides protection against incompatible data. In other words, if you change the size of an object, the calculated results based on the old values are no longer meaningful.

• Type 4 and press <Enter> to use the Demo Mode.

NOTE: Because file saving is suppressed, you cannot solve new problems in Demo Mode!

```
Enter path for application [C:\PCE\]?
```

Fig 2.70 Path (including disk)

• Press <Enter> to create the project files in the current subdirectory. Otherwise, supply the desired subdirectory before pressing <Enter>.

```
Define the following:
  Problem Keyword (1 to 8 characters)>
```

Fig 2.71 Project name

• Supply up to eight characters to serve as the file name for the project and press <Enter>.

Blanks are not allowed! Follow the DOS conventions. PCE will warn you if you attempt to use an existing file name. Each of the data files associated with this project has this common name and a unique extension.

```
Problem description
   (1 to 50 characters)>
```

Fig 2.72 Project description

• Type the description and press <Enter>.

This text serves only as a reminder to you. Normally, you record details of the variation of the problem under study, e.g., mesh refinement, change in properties, etc.

PCE returns you to the Filer menu.

8.6. Choose an existing application.

• Type 6 and press <Enter>.

```
Enter path for application [C:\PCE\]?
```

Fig 2.73 Path (including disk)

• Press <Enter> to create the project files in the current subdirectory. Otherwise, supply the desired subdirectory before pressing <Enter>.

PCE supplies a list of all projects contained in the subdirectory you specified.

```
The following applications are available:

1. COLUMN                  2. PLATE

Choose an application by number:[1]>
```

Fig 2.74 Project selection

• Type the number and press <Enter>.

```
User Interaction Mode:

1. Maximal (no defaults)
2. Intermediate (some defaults)
3. Minimal (free run)
4. Demo with previous data

...Select by number <Enter>
```

Fig 2.75 User level

The user levels are the same as in Fig 2.69. Demo Mode has been added. Use Demo Mode if you wish to suppress file saving. This option is useful if you are giving a class demonstration where you wish to explore various alternatives without risk to your data files. Otherwise, if you change values, PCE designates dependent files as inactive and, therefore, unusable. This feature provides protection against incompatible data. In other words, if you change the size of an object, the calculated results based on the old values are no longer meaningful.

• Type the desired number and press <Enter> to filer main menu (Fig 2.57).

8.7. Change user interaction mode.

PCE presents the user interaction level menu (Fig 2.75.

• Select the desired user interaction level and press <Enter>.

To leave this module:

• Select 0 and press <enter>.

```
FILER  (7.00)                                                    EXIT


1. Exit to PROGRAM MENU
2. Don't Exit; Remain in 'FILER'

0. Stop

... Select by number <RETURN> [1]>
```

Fig 2.76 Filer exit menu

• Select 1 and press <enter> to return to the main menu (Fig 2.8).

This brings the "Quick Tour" to a close. For additional details, study the reference chapter. Otherwise, try a problem now.

Chapter 3

PC-Elastic Command Reference

3.0 Program Organization

The main menu provides branching to all major modules of PCE. The menu list, with the exception of the file manager which is used both first and last, indicates the typical progression during problem solving. Each module provides a linkage to the next module and to the main menu in order to branch to other modules.

- *Geometry* and *Mesh* provide graphical tools for the automatic generation of the mesh and for you to make detailed refinements in the mesh. Automatic bandwidth reduction, a technique to reduce storage requirements and computation time, is also available here.

- *Preprocess* provides the environment for you to enter the material properties and the constraints. You are responsible for defining a well-posed problem which has a unique solution.

- In *Solve* you compute the desired output. This module forms and solves the required system of equations.

- In *Postprocess* PCE solves for element strains and stresses and nodal stresses.

- In *Plot* you produce graphical output.

- In *File Management* you have access to the numerical results. This module also allows you to create new projects and to duplicate existing projects. You can also use this module to formulate problems without utilizing PCE's graphical support. You might choose to do this in order to gain a deeper understanding of the internal numerical steps or to have greater control over problem formulation.

PC-Elastic commands are described in detail in this chapter; a second, more complicated example is woven into the discussion. The illustrative figures correspond to the classical flat plate with a circular hole which is subjected to tensile loading. However, our primary goal is to present a definitive description of each command. Each command description is free-standing and does not depend intrinsically upon this particular example.

Example: A flat plate with a circular hole subject to tensile loading

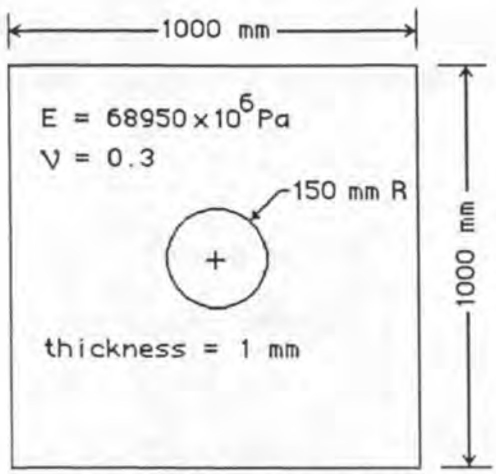


Fig. 3.1 A flat plate with hole

In this chapter we use the classical flat plate problem (Fig 3.1) to illustrate the features of PC-Elastic. A thin flat plate 1 mm thick and 1000 mm on a side has a circular hole of radius 150 mm at the center. We use PCE to find the displaced shape and to compute the stresses and strains. The schematic (Fig 3.2) contains the details you need in order to analyze this problem. The meaning of the notes on the diagram become apparent as you complete the following exercise.

The problem is planar—one of the two general classes of problems which you can solve with PC-Elastic. Using symmetry arguments you need to analyze only one fourth of the plate. By subdividing the five sided object you can use two four-sided mesh generating regions to automatically generate a mesh. One boundary of each of these regions has a curved side which you approximate using straight sided triangles.

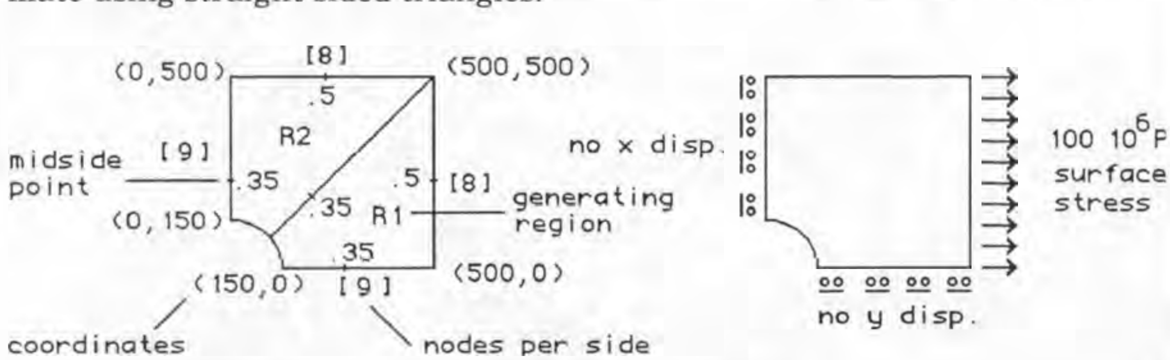


Fig 3.2 Schematic of the flat plate problem.

This chapter provides a detailed reference guide to the various PCE prompts and, unlike Chapter 2, is not a step-by-step tutorial.

3.1. Start your computer and activate PCE.

- Perform the usual startup process using your system disk.
- Select the subdirectory which contains PC-Elastic.
- Type PCE <Enter> to activate the program. The title screen appears.

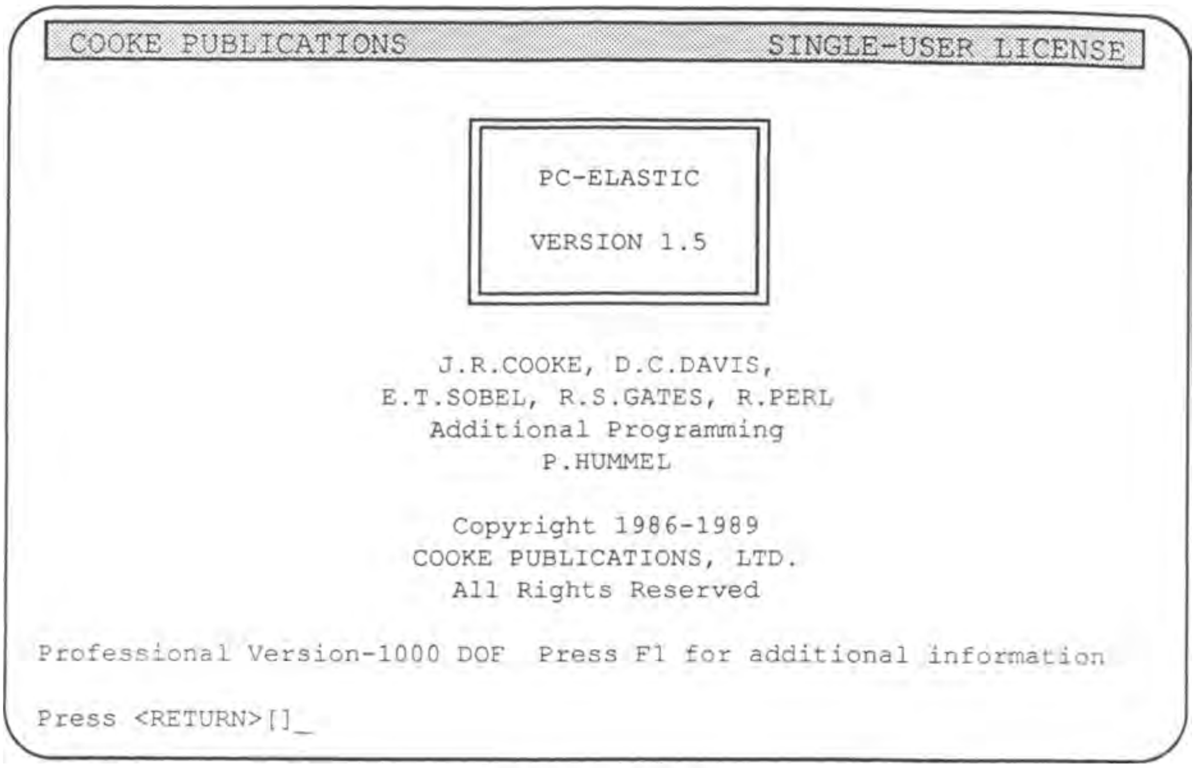


Fig 3.3 Title screen

PC-Elastic™ is a copyrighted program and is licensed, not sold, to the end-user. Duplication for archival purposes by the end-user is governed by US Copyright laws. The standard copy commands provided by DOS are satisfactory for this purpose. The student version of PCE is arbitrarily limited to 300 degrees of freedom; the professional version handles problems up to the 640K limit of addressable memory.

- Press F1 for additional information about the program.

This screen supplies the publisher's address and PCE version information.

Please send comments on PC-Elastic to:

J. Robert Cooke
Cooke Publications, Ltd.
PO Box 4448
Ithaca, NY 14852

If you have solved problems which you are willing to have described to other PCE users, please send them to the above address.

Version: 1.5
Date: January 6,1989

ENTER
Press <~~RETURN~~> to continue...

Fig 3.4 About PCE

The main menu controls movement among the seven submodules of the program. You use File Management first to initiate a problem and last to handle the numerical output. Otherwise, you execute the steps in the order listed on the main menu.

Problem formulation requires the specification of the 1) problem geometry, 2) the material properties, and 3) the boundary conditions. Solve and Postprocess assemble and solve the equations. Plot and File Management provide the output.

PC-ELASTIC PROGRAM MENU

1. GEOMETRY
   Define Problem Geometry
2. MESH
   Generate & Modify Mesh
3. PREPROCESS
   Define Properties and Boundary Conditions
4. Solve
   Assemble Equations and Solve for Nodal Displacements
5. POSTPROCESS
   Solve for Element Strains and Stresses and Nodal Stresses
6. PLOT
   Plot Grid, Boundary Conditions, and Problem Results.
7. FILE MANAGEMENT
   Create or Select Project

0. STOP

ENTER
... Select by number <~~RETURN~~>[1]_

Fig 3.5 The main menu

3.2. Select or create a project in File Management.

• Select 7 from the main menu and press <Enter>.

```
                              PC-ELASTIC

                           FILE MANAGEMENT

ABSTRACT:

Management of data files is controlled by a master file with the ex-
tension PCE.  You must initialize this file with the problem name and
problem description in order to begin an analysis.
         ENTER
Press <RETURN>[]
```

Fig 3.6 File management abstract

The File Management module creates a master file which contains the names and status of the data files. The modular structure of PCE allows you to complete the formulation and solution process during multiple sessions. You can easily suspend and restart work on a project, and you can begin at any step and explore alternative formulations. A problem duplication option allows you to extract the input data from an existing file. This preserves the existing project and provides a framework for the related problem. The master file contains the problem description and a record of the file compatibility. For example, if you modify the mesh, PCE tags all files which are dependent upon this information as inactive to prevent incompatibilities.

• Press <enter> to continue.

The FILER MAIN MENU appears.

```
 FILER  (7.00)                                                 MAIN MENU


1. Define problem and prepare master file
2. List Problem Keyword and Description
3. List all records with description and status
4. List contents of data files referenced by records
5. Duplicate an existing application
6. Choose an existing application
7. Change User Interaction Mode

0. None of the Above
                           ENTER
... Select by number <RETURN>[1]
```

Fig 3.7 Filer main menu

In addition to project creation, this module allows you to select from among existing projects and to change the user interaction mode.

3.2.1. Define problem and prepare master file.

Select 1 from the Filer menu and press <Enter> to begin project definition.

```
Enter path for application [C:\PCE\]?
```

Fig 3.8 Path (including disk)

On which disk and subdirectory should the data files be stored? Press <Enter> to create the project files in the current subdirectory. Otherwise, supply the desired path according to DOS conventions before pressing <Enter>.

```
Define the following:
  Problem Keyword (1 to 8 characters)>
```

Fig 3.9 Project name

Assign a file name to identify the family of data files. Supply up to eight characters to serve as the file name for the project and press <Enter>. Blanks are not allowed! Follow the DOS conventions. You will be warned if you attempt to use an existing file name in the current subdirectory. Each of the data files associated with this project has this common name and a unique three character extension.

```
Problem description
   (1 to 50 characters)>
```

Fig 3.10 Project description

Supply free-form text to characterize and identify the problem. Type the description and press <Enter>. This text serves only as a reminder to you. Normally, you record details of the variation of the problem under study, e.g., mesh refinement, change in properties, etc.

PCE creates a master file having the keyword as the name and PCE as the extension.

NOTE: ALL DATA FILES MUST BE KEPT IN THE SAME SUBDIRECTORY. THE NAMES AND STATUS OF ALL DATA FILES ARE MAINTAINED IN THIS FILE SO DON'T ARBITRARILY REMOVE OR MODIFY ANY OF THE 24 FILES.

```
User Interaction Mode:

1. Maximal (no defaults)
2. Intermediate (some defaults)
3. Minimal (free run)

...Select by number <Enter>
```

Fig 3.11 User interaction level

The user interaction level controls the level of detail presented to you during execution. Option 1 allows you to probe the deepest within the algorithms if you wish to study the method and program. Option 2 suppresses questions about the inner workings which are unnecessary during ordinary usage. Option 3 suppresses all but essential questions and executes steps automatically whenever possible.

• Select an option and press <Enter> to return to the Filer main menu.

Note: You could also have selected an existing project using File Management menu option 6.

```
 FILER  (7.50)                                APPLICATION SELECTION


The following applications are available:

1:  COLUMN                        2: PLATE

Choose an application number:[1]>_
```

Fig 3.12 Project selection

3.2.2 Define the geometric properties.

From the main menu select 1 (Geometry) by pressing <Enter>.

Note: If no applications are present, PCE routes you to the File Management module.

```
                              PC-ELASTIC

                           GEOMETRY (1.00)

ABSTRACT:

Discretization begins with the creation of curvilinear QUADRILATERAL
subregions which together describe the overall geometry.

Subsequently these input regions will be sub-divided into a mesh of 3-
node triangles.
         ENTER
Press <RETURN>[]
```

Fig 3.13 The Geometry abstract

Creation of a mesh to approximate the problem geometry is perhaps the most subjective aspect of using PCE. In PCE you must form a mosaic of triangular elements to completely cover the project geometry. The elements must not overlap and must have no unintended gaps. The elements must match the boundaries because you must subsequently apply boundary conditions.

Subsequently you apply constraint conditions at the vertices (nodes) of the triangles and on the sides of elements on the boundary. In other words, you must position triangle vertices to coincide with the location of applied forces.

The triangles should be well-shaped, i.e., roughly equilateral, to produce the best numerical results. The calculation process produces displacement components at the vertices, and you find the displacement components at all interior points by interpolation. Thin slivers must be avoided.

PCE uses the approximate solution for the individual triangles (elements) to approximate the larger solution. An elaborate bookkeeping scheme tracks the relationship of the individual elements—which triangles are adjacent, which share a common vertex, etc.

```
GEOMETRY (1.05)                                            MAIN MENU


ELEMENT TYPE?

1. Two-Dimensional 3-Node Triangle
2. Axisymmetric 3-Node Triangle

... Select by number <RETURN>[1]
```

Fig 3.14 The element type

Next you must specify the type of element. In this case, you choose from a two-dimensional triangle and a triangular annulus for axisymmetric problems. Commercial finite element software includes a larger collection of elements. The elements not only contain the geometric building blocks but also the governing partial differential equation content. Note: The default reflects the selected application.

Planar uses Cartesian coordinates (x,y) and assumes no change in the z-direction. Axisymmetric type uses cylindrical coordinates (r,z) and assumes no change in geometry or constraints with respect to angular position about the z axis of symmetry.

• Select the element type and press <Enter>.

With these preliminary questions answered we are now ready to formulate a mesh. In general, the mesh should consist of smaller triangles in the vicinity of largest change in the dependent variable and larger triangles where the change is modest. Unlike a finite difference formulation, the finite element method easily accommodates this mesh variation.

For classical elasticity problems the smaller the elements, the better the approximate solution you can obtain. The tradeoff, of course, is the finer the mesh (and the larger the number of calculated values), the larger the storage space required in memory and on disk, and the larger the computation time. In practice, you compute the result using one mesh and again with a refined mesh. This process is repeated until you judge that the process has converged.

The mesh generation process is subjective, must anticipate the subsequent steps, and works best when you can anticipate the solution. Otherwise, you must use additional exploratory meshes.

Although you can generate a mesh manually (option 2 in Fig 3.16), you will quickly come to appreciate the desirability of having this process automated. The automatic mesh generation scheme used in PCE is described by Segerlind (1976).

In brief, mesh generating regions are constructed first and then used to create the mesh—the variable mesh sizes, the coordinates of the vertices, the numbering of the vertices (nodes) and the numbering of the triangles (elements), the connectivity of triangles, etc.

The mesh generating regions (Fig 3.15) in PCE must be curvilinear quadrilaterals, i.e., each of the four sides can be straight or a second degree polynomial. This includes squares and rectangles with straight sides and with curved sides, as well as the degenerate case of a triangle where two consecutive sides are collinear. You need three points to define each side of the quadrilateral. Therefore, a total of eight points define each quadrilateral.

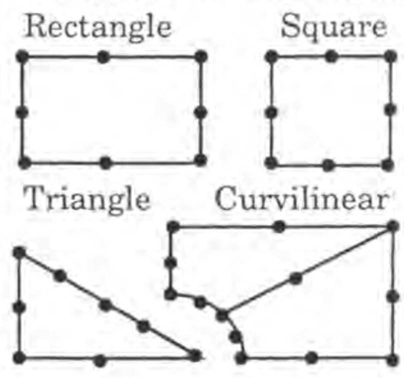
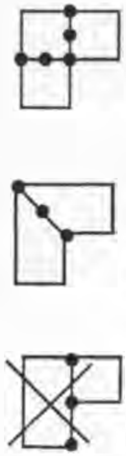


Fig 3.15 Examples of regions



Correct joining (3 points common)

Correct joining (3 points common)

Incorrect joining (3 points not common)

Fig 3.16 Joining regions

Often more than one of these regions is required. In that case the regions must share a common boundary and the three points which define that common side must be common points.

• Select option 1 in the Geometry menu (Fig 3.17).

```
GEOMETRY (1.05)                                              MAIN MENU

--- POINT COORDINATE ENTRY ---
1. Input with Graphics
2. Input without Graphics

--- DATA REVIEW ---
3. List Input Data
4. Plot Input Data

0. None of the Above
                                      Enter
... Select by number <RETURN>[1]
```

Fig 3.17 The Geometry menu

> **You** must assure a consistent set of units throughout the project!

3.3 Define mesh generating regions.

The geometry creation occurs in two phases: phase 1 entails creating an outline of the problem (points representing endpoints and midpoints of line segments comprising the region). Phase 2 involves selecting these points in an appropriate manner to create regions.

```
GEOMETRY (1.05)                                    DATA ENTRY WITH GRAPHICS

Is data stored in a disk file [N]?
```

Fig 3.18 Retrieve existing data?

- Type "N" and press "Enter".

- Supply the axes endpoints.

```
GEOMETRY (1.10)                                    DATA ENTRY WITH GRAPHICS

-- SET AXIS LIMITS --

Minimum X [0.00]>
Maximum X [0.00]>500

Minimum Y [0.00]>
Maximum Y [0.00]>500
```

Fig 3.19 The axis endpoints

- Press <Enter> after entering the Y axis maximum.

GEOMETRY (1.20) COORDINATE ENTRY

INSTRUCTIONS:

Enter the EIGHT points which define each quadrilateral region. There must be FOUR vertices and ONE point between each set of TWO vertices.

Adjacent regions share the THREE points along the common side.

As a degenerate case, the sides may be straight lines or TWO connected straight lines to form a single straight side of a triangular region.

Press <~~RETURN~~ ENTER> to continue...

Fig 3.20 Geometry instructions

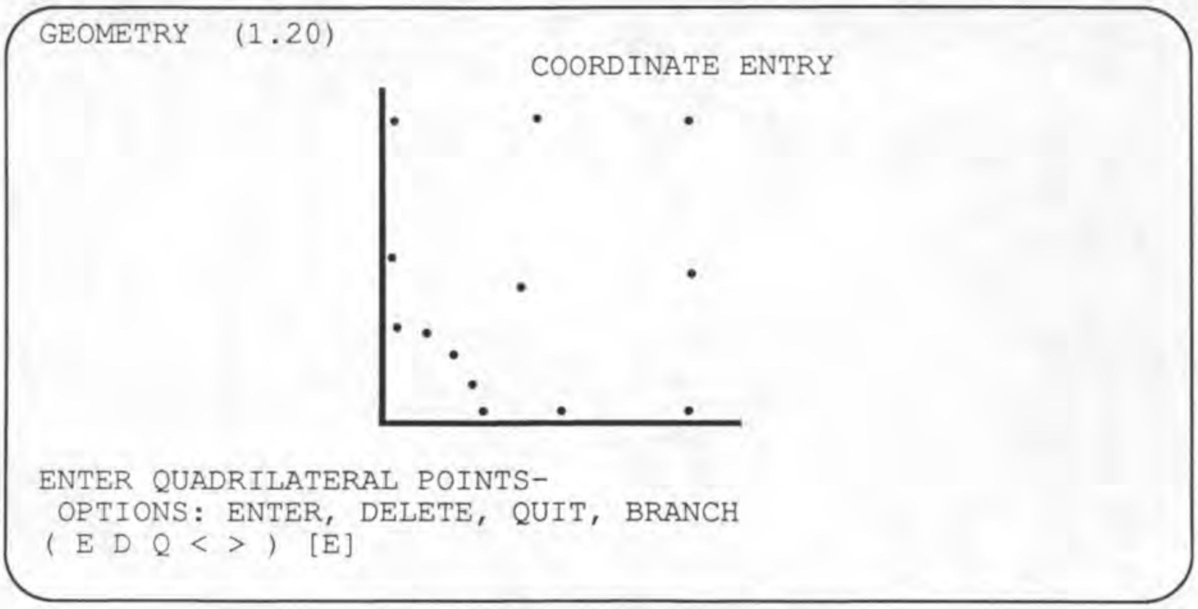


Fig 3.21 The region generation points

For reference purposes, here are the tools you use to create regions:

Note: In graphics mode, do not press <Enter>; the commands require only a keystroke.

Press "E" to gain access to point entry.

Press "D" to delete points you select using the cursor, and press "D" to delete.

Press "Q" to quit entry of points.

"<" and ">" allow you to branch backward and forward, skipping the pending input. [Note: These have not been reactivated yet in this version.]

will let you

If you press "E", you can add single points (P), points on a line (L), points on an arc (A), as well as quit (Q) and branching.

P : Supply the two coordinates of a point.
L : Supply the starting point coordinates, ending point coordinates, the number of intermediate collinear points you want to generate by interpolation, whether you want the points to be evenly spaced, and if not, their fractional distance (a decimal between .3 and .7) from the starting point. The placement of the intermediate point nearer one end of a side causes the elements to be smaller. Use 0.3 for the plate problem to make the elements smaller near the hole.
A : Supply the coordinates of the center of the arc, the coordinates of the starting point of the circular arc, the total angle of arc in degrees (counterclockwise is positive), the number of intermediate points you want generated by interpolation if evenly spaced, and if not, the fractional distance from the starting point.
Q : Select after you have placed all the generating points.

- Add the 13 points shown in Fig 3.21. PCE ignores extra points.
- Select "Q" to leave this section.

```
GEOMETRY (1.25)                                      REGION DEFINITIONS

INSTRUCTIONS:

Each region is defined by a COUNTERCLOCKWISE sequence of 8 points.
Use arrows to move the cursor to the desired point.  Then select the
point by pressing <S>.  The first point must be a VERTEX.

Each region will be numbered at its centroid when completed.

Press <RETURN> to continue...
```

Fig 3.22 Instructions for forming regions

- Press <Enter>.

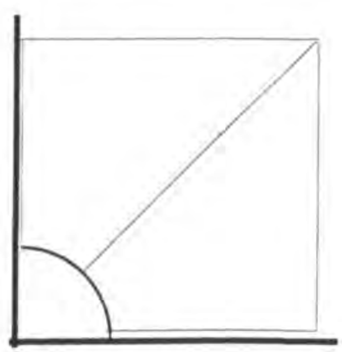
```
DEFINE REGION NUMBER-
REGION = [1]>
```

- Press <Enter> to accept the default of 1.
- Connect the points to define the regions.

Note: The order of points located by pressing the arrow keys are probably not in the sequence to form the region(s) in counterclockwise order. ***Remember to select eight in counterclockwise order starting at a vertex!***

After you have selected exactly eight:

```
GEOMETRY  (1.20)                                      REGION DEFINITION
```



```
DEFINE NODES FOR REGION 2
USE ARROWS TO MOVE CURSOR
Selecting node 8 of 8
( S D Q < > ) [S]>
```

Fig 3.23 Counterclockwise region formation

• Press “Q” to quit selection.

Watch as PCE forms the region to assure that you assigned the region points in a counterclockwise order. PCE asks:

```
Is this OK?
Are the 8 nodes defined in a COUNTERCLOCKWISE sequence? [N]>
```

• Press “Y” and <Enter> if you traced the region in a counterclockwise order.

• `DEFINE ANOTHER REGION [N]?`

Type “N” after you define the second region.

PCE notifies you of the number of regions you have defined.

When you have completed exploring this module:

• Press <Enter> twice to return to the Geometry menu.

> Note: To avoid corrupting your files, never simply turn the power off in the midst of a project.

• Select items 3 and 4 to review your results.

• Select the last item of the menu to leave this module.

The final module menu allows you:

```
1. to proceed to the next logical step ("Mesh")
2. to return to the main menu
3. to change your mind and remain in this module

0. to terminate the session altogether
```

• Select 1 and press <Enter>.

3.4. Create the mesh.

You can enter the Mesh module from the main menu or as a continuation from the Geometry module above.

```
                              PC-ELASTIC

                              MESH (2.00)

ABSTRACT:

A mesh of 3-node triangular elements is generated for the problem do-
main.  Next, nodes are renumbered to reduce the bandwidth, and unique
lines and nodes on boundaries are identified automatically.


Press <RETURN>[]
```

Fig 3.24 Mesh module abstract

The Mesh module has two phases. The automatic mesh generator takes each properly defined region and breaks it into triangular areas referred to as "elements". PCE saves the nodal coordinates and mesh connectivity data. This module then automatically renumbers the element vertices, referred to as nodes, to improve computational efficiency and identifies the boundaries.

This module uses the data file of points (.XY) and connectivity data (.REG) created in the GEOMETRY module and creates node and region data for the elements (.EL), and the coordinates and attributes of the nodes (.ND). After nodal renumbering the data for renumbered element nodes (.REL), coordinates of renumbered nodes (.RND), renumbered nodes for unique lines (.RLN), and renumbered boundary nodes and elements (.RBO) are saved.

• Press <Enter>.

```
 MESH (1.05)                                                MAIN MENU

-- MESH DEFINITION --
1. Do complete Mesh Generation (2-4)

2. Define Mesh Elements & Nodes
3. Renumber Nodes to Reduce Bandwidth
4. Sort Unique Lines & Boundary Nodes

-- MESH EXAMINATION --
5. List Element Nodal Coordinates
6. List Line & Boundary Nodes
7. Plot Entire Mesh

0. None of the Above

... Select by number <RETURN>[1]
```

Fig 3.25 Mesh module menu

3.5 Generate mesh.

Mesh option 1 executes options 2-4 without user interruption; you created all the required data files in the previous module. Consider separately each of the steps to define a mesh, to reduce the bandwidth, and to locate the boundaries, which you will use later when you assign boundary conditions.

- Type 2 and press <Enter>.

PCE retrieves the coordinates of the points you will use to define the mesh generating regions (.XY).

```
MESH      (2.77)                                  DEFINE NODES PER SIDE

INSTRUCTIONS:

The number of triangular element nodes on each quadrilateral region
side (including both end nodes) must be specified.  The minimum number
is TWO, which includes the endpoints.  Use arrows to move the cursor
to the desired side; select by pressing <S>.

Because elements must be connected at their nodes, opposite sides of a
region must have the same number of element nodes.  Therefore,
adjacent regions and their opposite sides are matched automatically.

Enter the number of nodes for each marked side.  Change any entries by
reselecting the side.

Press <RETURN> to continue...
```

Fig 3.26 Mesh instructions

As indicated in Fig 3.26, you specify the number of elements you want to generate within each of the mesh generating regions. PCE generates the triangular elements by creating four-sided "elements" (Fig 3.27) and then drawing the shorter diagonal to form two triangular elements. The shorter, rather than the longer, diagonal is used in order to produce triangles having roughly equal length sides.

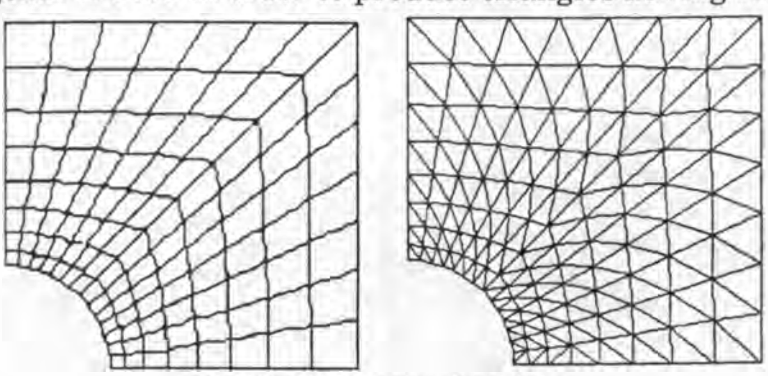


Fig 3.27 Mesh generation steps

The size of the elements is determined by the number of "grid" lines which connect opposite sides of the mesh generating regions (Fig 3.27 left). The intersection of these lines with a region boundary determines the placement of a node on that region boundary;

hence, you specify the number of "nodes per side". Opposite sides of a region have the same number of nodes and regions sharing a common boundary have the same number of nodes on that boundary. For properly defined regions, PC-Elastic enforces these constraints automatically. Fig 3.28 shows the input process.

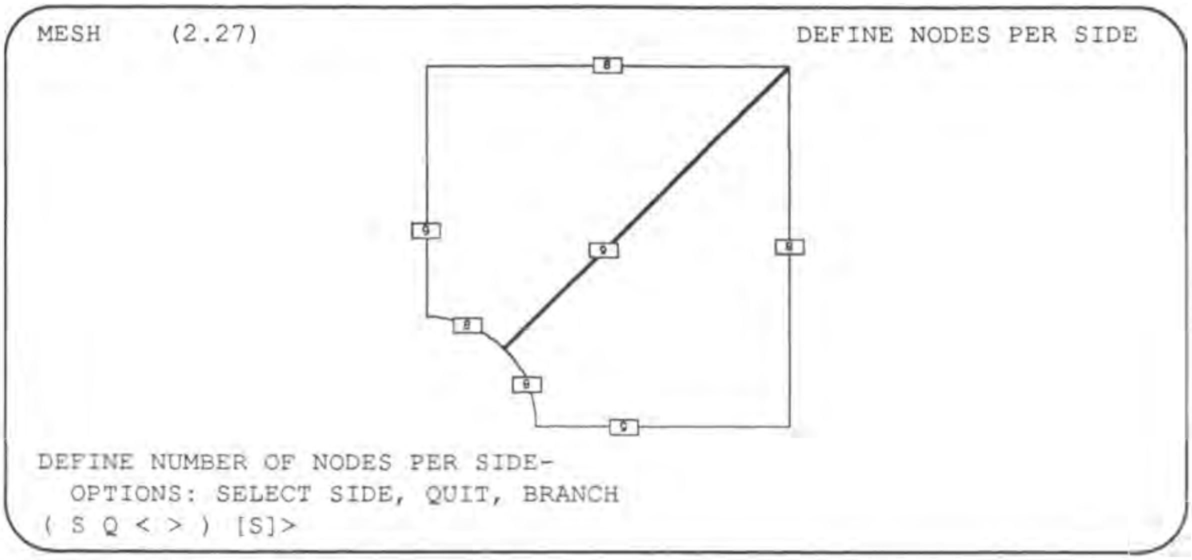


Fig 3.28 Assign number of nodes per mesh generating region side

The rectangle on each side of the mesh generating regions is an input box.

• Use the left and right cursor keys (with the NumLock key NOT set) to position the cursor on a side.

• Press S to select the current side.

```
DEFINE NUMBER OF NODES PER SIDE-
REGION: 2 sides: 2  4
Enter number of nodes (2-20) [3]>
```

PCE automatically assigns the same number to the opposite side and to any other side which is common to either of these two sides.

• Enter the number and press <Enter>. *Note: The <Enter> key is required because the number entry may contain two digits.*

• Move the cursor to each unassigned side and repeat the above process.

You can reassign values as often as desired. When satisfied:

• Press "Q" to quit.

```
Your problem has 270 Degrees of Freedom

Is this acceptable?[Y]>
```

PCE calculates and displays the number of unknowns (or degrees of freedom) required by the mesh you just defined. This number is a measure of the problem size (and, therefore, amount of memory and computation time). The student version has been arbitrarily limited to 300 DOF. The professional version allows you to use all the remaining computer RAM (of the 640 Kilobytes).

If the DOF number is too large, i.e., not acceptable, type "N"; otherwise,

• press <Enter>.

Alternative choices—

```
OUTPUT OPTIONS:

1. Output WITH plotting
2. Output WITHOUT plotting

... Select by number <RETURN>[1]>
```

A) If you choose 1, "Output WITH plotting", you must specify whether you wish to assign no labels, label nodes, or elements.

```
PLOTTING OPTIONS:

1. Plot mesh WITHOUT labels
2. Plot and label NODES
3. Plot and label ELEMENTS

... Select by number <RETURN>[1]>
```

If you select option 1, "Plot mesh WITHOUT labels", PCE creates each region separately and scales each to utilize the available screen area.

• Press <Enter> after each display to advance to the next region or to the composite generated mesh.

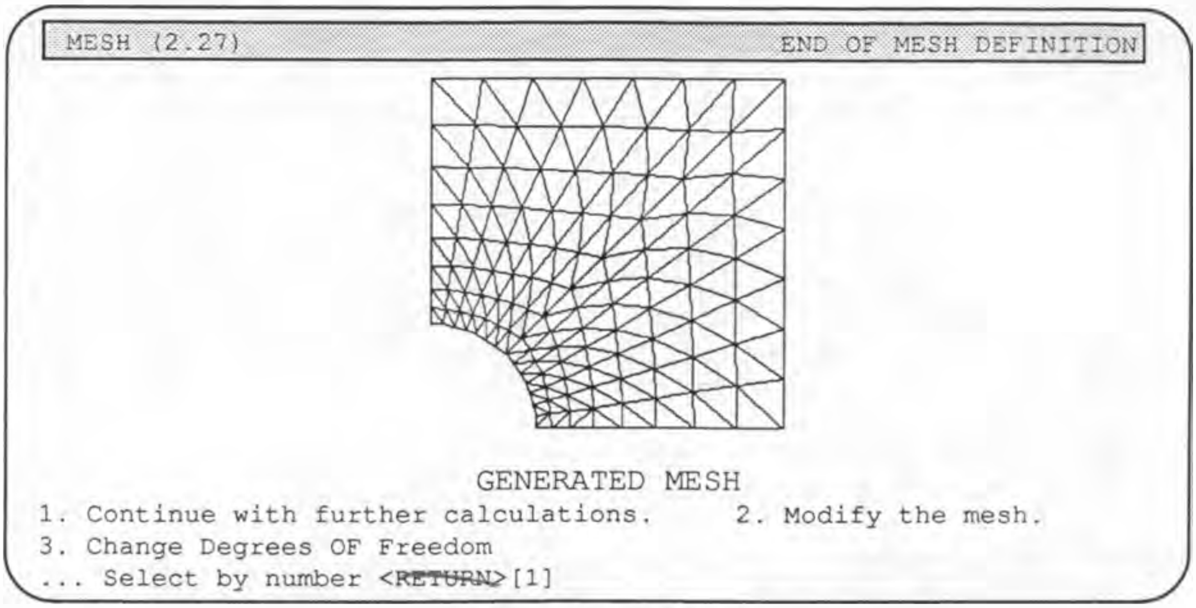


Fig 3.29 Generated mesh

We described two techniques for specifying mesh refinement: 1) move the middle point between two vertices on the side of a mesh generating region nearer one end of the side to make the elements smaller at that end, and 2) specify the number of nodes on each side of the mesh generating regions.

Since you must specify the displacement boundary conditions at nodal points, place nodes such that:

• elements are smaller in regions of greatest change of dependent variable;

• elements are roughly equilateral; and

• element sides coincide with geometric boundaries, material property changes, and points of loading.

NOTE: Nodal placement must anticipate these requirements.

If you are satisfied with the results (Fig 3.29), use option 1 to proceed to bandwidth reduction. If major mesh refinement is needed, choose option 3 to return to Fig 3.26.

Three additional graphical techniques are provided in this module to accommodate these requirements. Facilities to modify the mesh in other ways (i.e., move a node slightly, reorient a diagonal, and subdivide elements) are available. If only minor modifications are needed:

• Type 2 (for Modify the mesh) and press <Enter>.

```
MESH MODIFICATION OPTIONS:
1. Add nodes (and associated elements) 3. Reverse element definitions
2. Adjust node positions ENTER          0. None
... Select by number <RETURN>[1]
```

To refine the mesh by adding nodes (and subdividing elements),

• Type 1 and press <Enter>.

Fig 3.30 Before, during, and after added node

```
ADDITION OF INTERNAL NODE-
Select line on which node is to be created by selecting endpoints.
Now select FIRST ENDPOINT.    NODE 74
Press <S> at desired node. Press <Q> to exit.
```

• Use the cursor keys to identify a node.
• Press <S> to select the node.
• Repeat the process to select another node on the ***same*** element.

PCE displays the coordinates of the new node and draws the new elements (only one if you chose a boundary).

• Press <Q> to exit to mesh modification options.

To adjust node placement:

• Type 2 and press <Enter>.

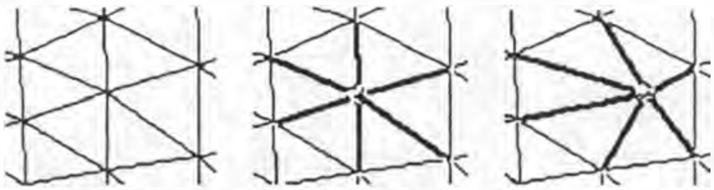


Fig 3.31 Before, during, and after node movement

```
Move box to nodes using arrow keys; select a node by pressing <S>.
Move node either by pressing <V> to input a value OR using arrow keys.
Press <Q> to stop adjustment of this node; press <Q> again to quit.
Node #10 X = 117.51    Y = 93.05
(S Q ) [S]>
```

• Select and position nodes using either the cursor or coordinate values.

• Type <Q> to accept the adjusted node position.

• Adjust other node, if necessary, or press <Q> to return to the mesh modifications options.

To reverse element definitions:

• Type 3 and press <Enter>.

Fig 3.32 Before, during, and after redefine

Reverse elements (Fig 3.32) allows you redefine the boundary between two adjacent elements, i.e., choose the longer rather than shorter diagonal in Fig 3.27.

```
Move cursor to selected pair using arrow keys.
Press <S> to select elements.
Press <Q> to stop element changes.
Elements: 199 and 200
(S Q ) [S]>
```

• Use the cursor keys to identify a boundary common to two adjacent elements.

• Press <S> to select the elements.

• Repeat the process as desired, and press <Q> to quit.

To leave this section:

• Type 0 and press <Enter>.

```
Number of ELEMENTS = 224
Number of NODES = 135
Bandwidth for 2 degrees of freedom per node is 256
Press <RETURN> to continue...
```

The "Plot and label NODES" and "Plot and label ELEMENTS" options mentioned before overlay the element and node numbers on the mesh. Since no zoom capability is provided here, these options are marginally useful. Refer to the PLOT module where zoom support for enlargement is provided. Also use option 7 of the Mesh main menu to identify individual values.

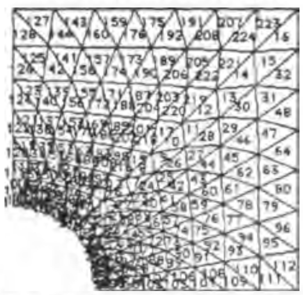


Fig 3.33 Element numbering

```
PLOTTING OPTIONS:

1. Plot mesh WITHOUT labels
2. Plot and label NODES
3. Plot and label ELEMENTS

... Select by number <RETURN>[1]>
```

Return now to the other alternative—tabular output.

B) If you choose "Output WITHOUT plotting",decide whether to display in a table the node numbers and coordinates or just a progress indicator. The Mesh Examination options (5-6) also provide access to this data.

```
OUTPUT WITHOUT PLOTTING:

1. Print NODES and COORDINATES of mesh
2. Print only CALCULATION STATUS

... Select by number <RETURN>[1]>
```

3.6 Bandwidth reduction.

When the mesh is satisfactory, the program goes through calculations (using the Collins algorithm) to reduce the bandwidth of the global "stiffness" matrix. This step reduces memory requirements and reduces computational time.

From the Mesh module main menu (Fig 3.25):

• Type 3 and press <Enter> to execute "Renumber Nodes to Reduce Bandwidth".

PCE retrieves the required data files (.EL and .ND).

• Press <Enter>.

```
PRINTING OPTIONS:

1. Print only CALCULATION STATUS
2. Print STATUS and FINAL RENUMBERING
3. Print DETAIL of renumbering

... Select by number <RETURN>[1]>
```

The option provides increasingly more detail.

The Collins algorithm first compiles a table of 'related nodes', i.e., a table of nodes which share an element side. The algorithm then uses each node as a starting point and heuristically reassigns node numbers using the nearest neighbor information. This does not guarantee an optimal numbering, but usually results in a substantial reduction if you are using multiple mesh generating regions. If no improvement is achieved, PCE uses the original numbering. PCE stores the renumbered nodes for later use. In the present example, PCE reduces the column width to store the equations from 128 to 15!

Use options 2 and 3 only if you wish to examine the renumbering process in detail.

3.7 Identify boundaries.

PCE will use the renumbered mesh throughout the remainder of the analysis. This option identifies unique lines and the connectivity of line segments. If all element nodes are traversed in a counterclockwise direction, the boundaries of elements which share a common side will be traversed in opposite directions. PCE can identify duplicate lines so only one of these lines need be plotted. Element sides which do not share a common side are external boundaries (or interior boundaries of multiply-connected regions), and you must assign boundary values to them in a subsequent module.

• Type 4 and press <Enter> to "Sort unique lines and boundary nodes" (Fig 3.25).

PCE retrieves the renumbered nodes and their nodal coordinates (.REL and .RND).

• Press <Enter>.

```
PRINTING OPTIONS:

1. Print only CALCULATION STATUS
2. Print STATUS and FINAL LISTS

... Select by number <RETURN>[1]>
```

Option 1 shows that of the 358 unique lines associated with the 224 elements, 44 are boundary lines with 44 nodes and one counterclockwise boundary.

Option 2 lists the unique mesh lines with node numbers and identifies the boundaries. Then PCE displays the information from this option. The results are also available below.

• Press <Enter>, as needed, to return to the Mesh main menu.

3.8 Mesh examination.

Mesh menu options 5-7 (Fig 3.25) list the nodal coordinates and the unique lines and boundary nodes.

• Type 5 and press <Enter>.

PCE displays the problem keyword, the coordinate system type, and the description of the problem.

• Press <Enter> as often as needed to review the nodal coordinates and the node numbers (in counterclockwise order) for each element.

Note: Use shift print screen to obtain a copy on an attached printer.

• Type 6 and press <Enter> to list the unique and boundary lines.

PCE retrieves the required files (.RLN and .RBO).

• Press <Enter> as often as needed to review the list.

• Type 7 and press <Enter> to plot the mesh.

PCE displays the problem keyword, the coordinate system type, and the description of the problem.

• Press <Enter> and PCE displays the mesh.

• Accept the default by pressing <Enter>.

```
Identify  nodes or elements [Y]?
```

• Type the starting node number and press <Enter>.

```
IDENTIFY  NODES
 Start at node # (1-135) [1]>
```

```
IDENTIFY NODES
MOVE MARKER USING ARROW KEYS. TYPE <Q> TO QUIT NODE IDENTIFICATION
NODE #1    X = 215.6250    Y = 0.0000
( Q ) [Q]>
```

• Use the cursor keys to move the marker.

PCE displays the node number and associated node coordinates.

• Type <Q> to advance to element identification.

```
IDENTIFY  ELEMENTS
Start at element # (1-224) [1]>
```

```
IDENTIFY ELEMENTS
MOVE MARKER USING ARROW KEYS. TYPE <Q> TO QUIT ELEMENT IDENTIFICATION
ELEMENT #1
( Q ) [Q]>
```

- Use the cursor keys to move the marker.

PCE displays the element number.

- Type <Q> to leave element identification.

- Type <Enter> to return to the Mesh menu.

Use the last menu option (0. None of the above) to leave the mesh module.

- Type 1 and press <Enter>.

Note: The module exit menu allows you to move to the next logical module, to return to the main program menu, to change your mind and remain in the current module, or to terminate the program gracefully.

```
1. Proceed to "PREPROCESS'
2. Exit to Program Menu
3. Don't Exit: Remain in 'MESH'

0. Stop

... Select by number<RETURN>[1]>
```

3.9 Preprocess.

After you have prepared the mesh files using the Mesh module (or without graphics support using the Library module), define the element material properties and body forces and assign the boundary conditions or constraints (on boundaries and at interior nodes). You enter body force conditions here because the input structure is the same as for property definition. These steps are required to

```
                              PC-ELASTIC

                           PREPROCESS (3.00)

ABSTRACT:

Specify MATERIAL PROPERTIES and BOUNDARY CONDITIONS, in that order.
The required properties are the ELASTIC MODULUS, POISSON'S RATIO,
THICKNESS, TEMPERATURE, and THERMAL EXPANSION COEFFICIENT for each el-
ement. Boundary conditions include POINT FORCE, FIXED NODAL
DISPLACEMENT, and SURFACE STRESS.

Press <RETURN>[] <ENTER> to continue...
```

Fig 3.34 Preprocess abstract

uniquely define a problem and to create the properties file (.PRP), the initial (or input) boundary conditions (.IBC), and final (or equivalent nodal values) boundary conditions (.BC). This is the final step in problem formulation.

- Press <Enter>.

PCE displays the problem keyword and description.

- Press <Enter>.

PCE retrieves the data required for this module that you created in the earlier steps.

- Press <Enter> to display the Preprocess menu (Fig 3.35).

```
PREPROCESS (3.00)                                              MAIN MENU

--- DATA INPUT ---
1. Input Material Properties and Thickness
2. Input Potential, Flux, and Source Boundary Conditions

--- DATA EXAMINATION ---
3. List Properties
4. Plot Properties
5. List Input Boundary Conditions
6. List Equivalent Boundary Conditions
7. Plot Input Boundary Conditions
8. Plot Equivalent Boundary Conditions

0. None of the Above

... Select by number <RETURN>[1]
```

Fig 3.35 Preprocess module main menu

You can enter the Properties module from the main menu (Fig 2.6) or as a continuation from the previous Mesh module.

This module handles two major tasks: 1) entering and editing element properties and 2) entering and editing conditions at boundary and interior nodes. Distributed body conditions use the same input structure.

3.10 Properties.

- Select 1 and press <Enter> to enter the properties.

PCE finds the element centroids and the boundary nodes.

• Type Y and press <Enter> to retrieve the existing data files to become the default values or N if no such files exist..

```
Are properties in a disk file [N]?
```

```
PREPROCESS (3.18)                                      EDIT PROPERTIES

PROPERTY EDITING OPTIONS:

1. ELASTIC MODULUS
2. POISSON'S RATIO
3. THICKNESS
4. TEMPERATURE
5. THERMAL EXPANSION COEFFICIENT

0. NONE

... Select by number <RETURN>[1]
```

Fig 3.36a Preprocess module main menu

You must select and assign values for each property. For axisymmetric problems, the thickness does not appear. *NOTE: For two-dimensional problems, you must specify the thickness. Each element can have a different thickness.*

• Type 1 and press <Enter>.

```
PREPROCESS (3.18)                                      EDIT PROPERTIES

                      PROPERTY: ELASTIC MODULUS

ELASTIC MODULUS IS UNIFORM BY:

1. ENTIRE BODY
2. INPUT REGION
3. ELEMENT

... Select by number <RETURN>[1]
```

Fig 3.36b Elastic modulus menu

The menu allows you to enter common properties for the largest possible unit.

• Select 1 and press <Enter>.

```
                      PROPERTY: ELASTIC MODULUS
ELASTIC MODULUS is uniform in the body.

What is the ELASTIC MODULUS value [68950.00]>
```

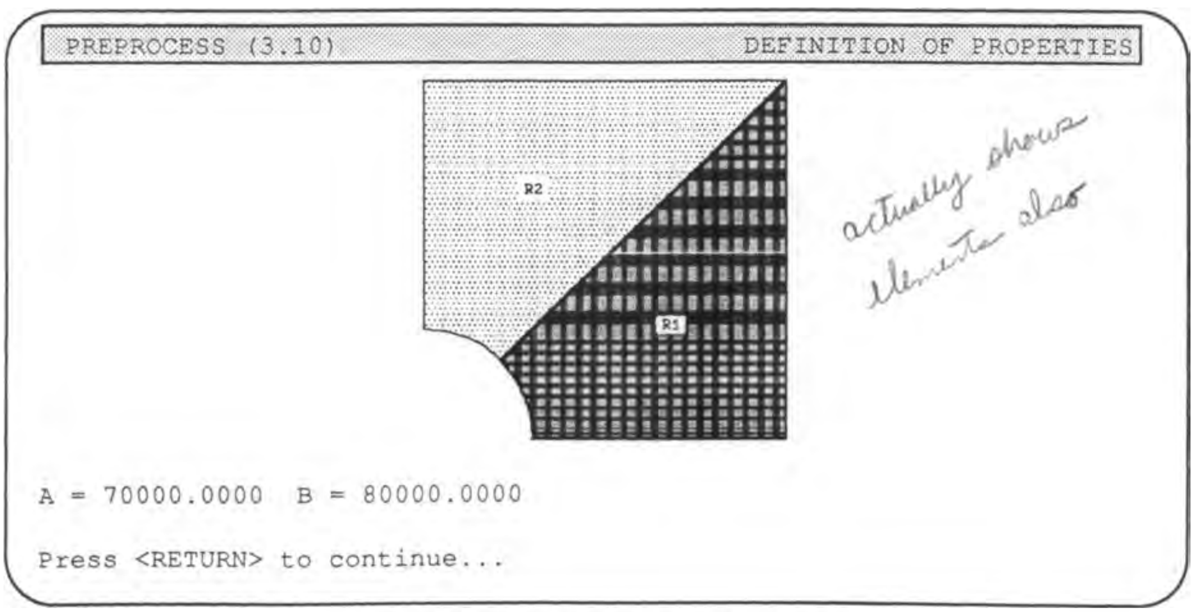
If you had selected variation by input region or by element, then PCE would require you to supply that additional information.

- If you select option 2 (by region):

```
                         PROPERTY: ELASTIC MODULUS
ELASTIC MODULUS is uniform in the body.

How many code ELASTIC MODULUS values are needed [2]>
```

Assign a value for each letter code.

```
                         PROPERTY: ELASTIC MODULUS
ELASTIC MODULUS is uniform in the body.

How many code ELASTIC MODULUS values are needed [2]>

Input a different value for each code:

CODE    ELASTIC MODULUS
  A       [70000] 70000
  B       [80000] 80000
```



actually shows elements also

Fig 3.37 Assign coded values by region

To assign the values,

- Type a letter code and press <Enter> for each region.

Option 3 (by element) is handled similarly.

```
Use ARROW keys to move cursor
Select coded value by letter  N to enter a value  Q to Quit
```

Note: Enter <N> if you wish to assign a specific numerical value.

- Press <Enter> to continue.

Enter values for the remaining properties (Fig 3.36) by the same process.

Note: DO NOT ASSIGN MEANINGLESS VALUES SUCH AS ZERO THICKNESS.

- Type 0 and press <Enter>.

```
Do you wish to proceed to definition of BOUNDARY CONDITIONS [Y]?>
```

3.11 Boundary conditions.

```
Are BOUNDARY CONDITIONS in a disk file [N]?
```

- Type Y and press <Enter> to retrieve the data files, or type N if it doesn't exist.

```
PREPROCESS (3.20)                         ENTRY OF BOUNDARY CONDITIONS

INSTRUCTIONS:

Boundary conditions may be DEFINED or DELETED at a node or for a range
of nodes and for each component direction.

First define the details of the B.C.; then specify the range.

Ranges are COUNTERCLOCKWISE (viewed from the body interior) from
starting point to ending point.

Press <RETURN> to continue...
```

Fig 3.38 Boundary conditions instructions

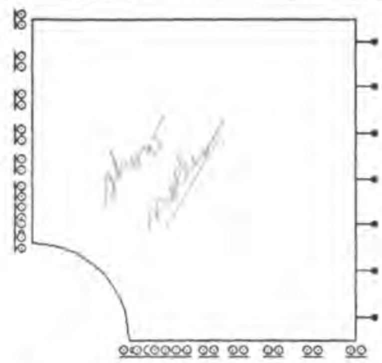
Press <Enter> (Fig 3.38) for the next screen .

Then use the initial letter of the command.

E: enter boundary conditions mode. The boundary condition choices are as follows:

```
1.Point force   2. Displacement  3. Surface stress  4. Body force
```

```
PREPROCESS (3.20)                                   INPUT BOUNDARY CONDITIONS
```



```
DISPLACEMENT     POINT FORCE     SURFACE STRESS
DEFINE BOUNDARY CONDITIONS-
OPTIONS: ENTER, DELETE, QUIT, BRANCH
( E D Q < > [E]>
```

Fig 3.39 Boundary conditions

You can (Fig 3.39) Enter new conditions, Delete existing conditions, or Quit.

If you select E, you can: 1) assign point forces at interior nodes or on the boundaries, 2) specify displacement components at interior or boundary nodes, 3) specify stress components (x-direction, y-direction, normal, or tangent), or assign body forces (x-direction or y-direction). You can assign conveniently the boundary conditions over a range of the boundary. The same options apply to the delete mode.

- Press E (without pressing <Enter>).

```
DISPLACEMENT     POINT FORCE     SURFACE STRESS
ENTER-
1. POINT FORCE         3. SURFACE STRESS
2. DISPLACEMENT        4. BODY FORCE
( 1 2 3 4 Q < > )[1]>
```

You can assign four types of boundary conditions.

- Type 1 (and do not press <Enter>).

Will you assign point forces at interior nodes? If "N", then only boundary nodes will be accessible.

```
DISPLACEMENT      POINT FORCE      SURFACE STRESS
ENTER POINT FORCE-
AT INTERIOR NODES?
( N Y Q < > )[N]>
```

- If "Y", then choose an x-direction or y-direction component.

```
DISPLACEMENT      POINT FORCE      SURFACE STRESS
ENTER POINT FORCE-
1. X-DIRECTION          2. Y-DIRECTION
( 1 2 Q < > )[1]>
```

- Assign a component value.

```
DISPLACEMENT      POINT FORCE      SURFACE STRESS
ENTER X-DIRECTION POINT FORCE-
POINT FORCE = [0.00]>
```

- Use the cursor keys to locate a node and press "S" to assign the value.

```
DISPLACEMENT      POINT FORCE      SURFACE STRESS
ENTER X-DIRECTION POINT FORCE = 1.00000
Use arrows to move box to node.       Node 82
Press <S> at node.  Press <Q> to quit.
```

- Press "Q" to return to the "Define Boundary Condition" state.

Let's illustrate the assignment of a continuous range of boundary values.

- Press "E" to enter conditions (Fig.3.39).
- Press "2" to select displacement.

```
DISPLACEMENT      POINT FORCE      SURFACE STRESS
ENTER-
1. POINT FORCE          3. SURFACE STRESS
2. DISPLACEMENT         4. BODY FORCE
( 1 2 3 4 Q < > )[1]>
```

- Assign values only on the boundary.

```
DISPLACEMENT      POINT FORCE      SURFACE STRESS
ENTER DISPLACEMENT-
AT INTERIOR NODES?
( N Y Q < > )[N]>
```

No boxes around symbols

• Specify the component.

```
DISPLACEMENT      POINT FORCE      SURFACE STRESS
ENTER DISPLACEMENT-
1. X-DIRECTION          2. Y-DIRECTION
( 1 2 Q < > )[1]>
```

• Assign a component value.

```
DISPLACEMENT      POINT FORCE      SURFACE STRESS
ENTER X-DIRECTION DISPLACEMENT-
DISPLACEMENT = [0.00]>
```

• Specify a continuous range.

```
DISPLACEMENT      POINT FORCE      SURFACE STRESS
X-DIRECTION DISPLACEMENT = 0.00000000
Specify where to ENTER boundary condition.
1. Single point,  or 2. Continuous
( 1 2 Q < > )[1]>
```

• *NOTE: The range must be assigned in counterclockwise order!*

```
DISPLACEMENT      POINT FORCE      SURFACE STRESS
X-DIRECTION DISPLACEMENT = 0.00000000
Use arrows to move box to STARTING node.
Press <S> at STARTING node.           node 54
```

• Assign the ending point.

```
DISPLACEMENT      POINT FORCE      SURFACE STRESS
X-DIRECTION DISPLACEMENT = 0.00000000
Use arrows to move box to ENDING node.
Press <S> at ENDING node.           node 126
```

Surface stress assignments are handled in the same manner.

• Choose a component.

```
DISPLACEMENT      POINT FORCE      SURFACE STRESS
ENTER SURFACE STRESS-
1. X-DIRECTION         3. NORMAL (OUT)
2. Y-DIRECTION         4. TANGENT (CCW)
```

• Assign the (signed) magnitude.

```
DISPLACEMENT      POINT FORCE      SURFACE STRESS
ENTER X-DIRECTION SURFACE STRESS-
 SURFACE STRESS = [0.00]>50
```

- Specify the initial point of a continuous counterclockwise range.

```
DISPLACEMENT        POINT FORCE        SURFACE STRESS
X-DIRECTION SURFACE STRESS = 50.0
Use arrows to move box to STARTING node.
Press <S> at STARTING node.          node 72
```

- Designate the ending point.

```
DISPLACEMENT        POINT FORCE        SURFACE STRESS
X-DIRECTION SURFACE STRESS = 50.0
Use arrows to move box to ENDING node.
Press <S> at ENDING node.          node 126
```

Body force assignments
Assign body forces in the same manner. The cursor moves among the element centroids rather than among nodes.

- Select "Q" as often as necessary to quit.

- Indicate whether you want additional changes before leaving this module.

- Press <Enter> to see a summary for this section.

```
Assembling boundary conditions for each degree of freedom...

Stresses: 7
Forces: 0
Body Forces: 0
Displacements: 18

Press <RETURN> to continue...
```

- Press <Enter> again to return to the Preprocess main menu (Fig 2.24).

In this module you assigned constraints —boundary conditions and internal nodal conditions—which the solution must satisfy. To be "well-posed" the problem must lead to a unique solution. ***The responsibility for creating a well-posed problem rests with the user.*** An improperly formulated problem can lead to an ill-conditioned system of linear algebraic equations.

3.12 Data examination.

Options 3 - 8 of the Preprocess menu allow you to review (list and plot) the data entered or created so far.

3.13 List properties.

Type 3 and press <Enter>.

```
PREPROCESS (3.30)                                        LIST PROPERTIES
                     PROPERTY: ELASTIC MODULUS
ELASTIC MODULUS is uniform.
For all elements (1 to 224) the value is 68950.00000

Press <RETURN> to continue...
```

Fig 3.40 List properties

- Press <Enter>.
- Repeat the process for each of the other properties.

3.14 Plot properties.

- Type 4 and press <Enter>.

```
Plot all properties  [Y]?
```

- Type N and press <Enter>.

```
PLOT A SINGLE PROPERTY

1. ELASTIC MODULUS
2. POISSON'S RATIO
3. THICKNESS
4. TEMPERATURE
5. THERMAL EXPANSION COEFFICIENT

0. NONE

...Select by number <Return>[0]>
```

- Type 1 and press <Enter>.

```
                     PROPERTY: ELASTIC MODULUS
ELASTIC MODULUS is uniform.
For all elements (1 to 224) the value is 68950.00000

Press <RETURN> to continue...
```

- Press <Enter> to display the plot.
- Select other properties (2-5) or 0 and press <Enter> to leave this section.

3.15 List input boundary conditions.

- Type 5 and press <Enter> to review the assigned boundary conditions.

```
NODE      DIRECTION & TYPE            VALUE
135       X-DIRECTION DISPLACEMENT    0.0000
133       X-DIRECTION DISPLACEMENT    0.0000
...       .....                       ....
Press <RETURN> to continue...
```

- Press <Enter> as needed to return to the Preprocess menu.

3.16 List equivalent boundary conditions.

- Type 6 and press <Enter> to see the nodal equivalent boundary conditions.

```
NODE      B.C. TYPE                   VALUE
1X        FREE BOUNDARY               0.0000
1Y        DISPLACEMENT                0.0000
2X        FREE BOUNDARY               0.0000
...       .....                       ....
Press <RETURN> to continue...
```

- Press <Enter> as needed to return to the Preprocess menu.

3.17 Plot input boundary conditions.

- Type 7 and press <Enter>.

PCE displays the same plot (Fig 4.41) created when you assigned the boundary conditions.

- Press <Enter> to return to the Preprocess menu.

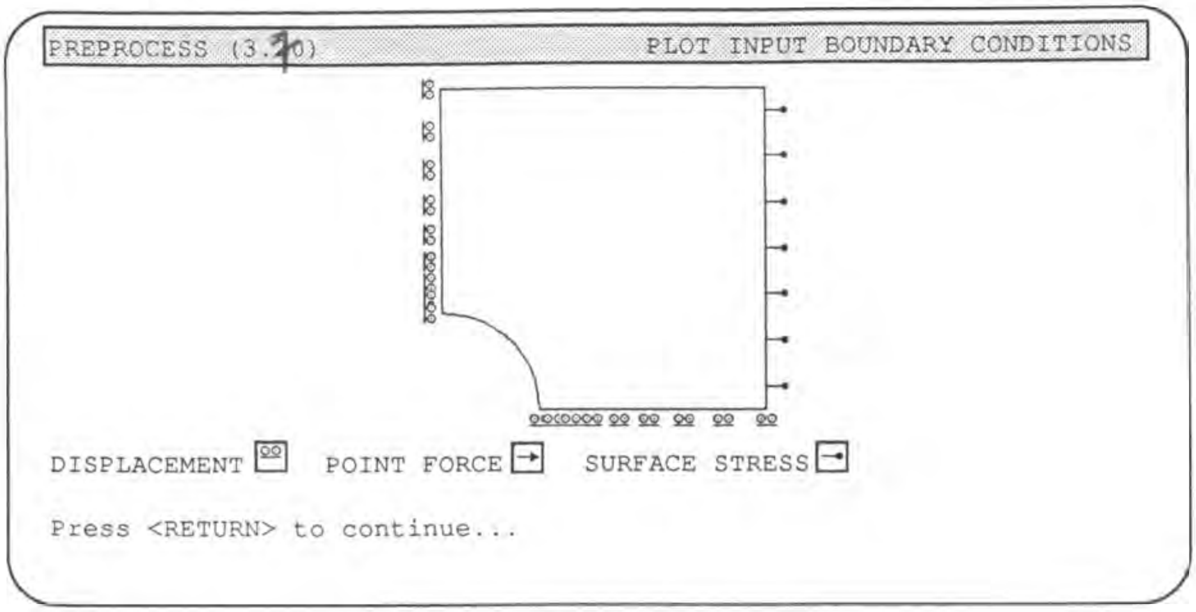


Fig 3.41 Plot input boundary conditions

3.17 Plot equivalent boundary conditions.

- Type 8 and press <Enter>.

Recall that the strategy used in PCE is to formulate a system of equations to find the dependent value at the nodal points. Consequently, PCE converts the input

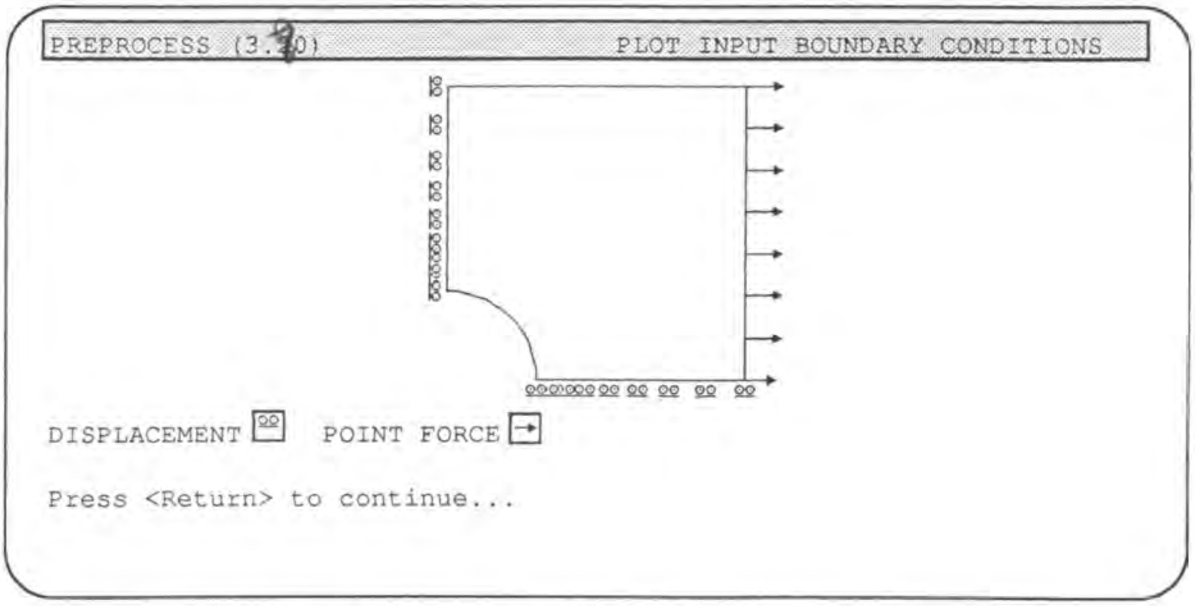


Fig 3.42 Plot equivalent nodal boundary conditions

constraints into "nodal equivalents" which it will use in subsequent calculations. It applies nodal forces and displacements at nodes, and surface stress are applied to the face of an element. PCE assigns surface stresses at the midpoint between two nodes, but internally converts them to equivalent nodal force values. In all instances you must use a consistent set of units! (Refer to Chapter 4.) In this example the surface load on the right edge is shown with equivalent point loads located at the nodes.

- Press <Enter> to return to the Preprocess menu.
- Type 0 and press <Enter> to leave Preprocess.

```
PREPROCESS (3.09)                                                   EXIT

1. Proceed to 'SOLVE"
2. Exit to Program Menu
3. Don't Exit; Remain in "PREPROCESS"

0. Stop
... Select by number <RETURN>[1]>
```

Fig 3.43 Preprocess exit menu

- Select 1 (Fig 2.30) to advance to the SOLVE module.

3.18 Form and solve the equations.

You can enter the Solve module from the main menu or as a continuation from the Properties module (Fig 3.43).

```
                              PC-ELASTIC

                              SOLVE (4.00)

ABSTRACT:

Finite element equations are defined for each element, assembled into
a global matrix equation, modified by boundary conditions, and solved
for nodal displacements.

Press <RETURN>[] to continue...
```

Fig 3.44 Solve module abstract

- Press <Enter> to continue.

After PCE reminds you of the project selected:

- Press <Enter> to continue, and PCE retrieves the required data files.

For two-dimensional problems specify whether you wish to assume plane stress or plane strain deformation. For a thin body with no loads perpendicular to the plane of the body, assume plane stress. On the other hand, if the dimension of the body normal to the applied load may not change, assume plane strain. See Chapter 23 of Segerlind (1976) for further details.

```
Does the analysis assume:

1. Plane Stress or
2. Plane Strain

... Select by number <RETURN>[1]>
```

• Type 1 and Press <Enter> to reach the Solve main menu (Fig 3.45).

```
SOLVE  (4.05)                                          MAIN MENU

-- FORM AND SOLVE ALL EQUATIONS --
1. Do Complete Problem Solution (Menu Options 2-5)

-- FORM SYSTEM EQUATIONS --
2. Assemble Global Force and Stiffness Matrices
3. Combine External Force Boundary Conditions
4. Apply Displacement Boundary Conditions

-- SOLVE SYSTEM OF EQUATIONS --
5. Solve for Nodal Displacements

-- LIST RESULTS --
6. List Nodal Displacements

0. None of the Above

... Select by number <RETURN>[1]
```

Fig 3.45 Solve main menu

3.19 Run with minimal intervention (Option 1)

An experienced user would select 1 to complete the formulation of the global system of equations, apply the boundary conditions, and find the nodal values without user intervention.

• To run without intervention, press <Enter> (Fig 2.33) .

Option 2 (Fig 2.33) asks if you wish to save intermediate values to disk and then asks whether to display the original global force vector, new global force vector, or both. When PCE applies the displacement boundary conditions, you can decide to display the original stiffness matrix, modified stiffness matrix, original force vector, modified force vector, all four of these, or the modified stiffness and force matrices. When PCE solves the system of equations, you may list the node displacements as PCE calculates them.

```
SOLVE (4.05)                              COMPLETE PROBLEM FORMULATION

OPTIONS:
1. Run WITHOUT user interaction (using all defaults)
2. Run WITH user interaction

... Select by number <RETURN> [1]>
```

Fig 3.46 Solve main menu

For instructional purposes you can handle options 2-4 separately. *If so, they must be executed in ascending order.*

3.20 Assemble global force and stiffness matrices (Option 2).

• Type 2 and press <Enter>.

```
Do you wish to save intermediate values to the disk?(slower)[Y]>
```

Type "N" and press <Enter>. Note: Save these intermediate results only if you wish to examine these intermediate calculations. *Otherwise, you will choose to avoid using this disk space and losing the time required to produce them.* See the File Management section for a discussion of the files PCE would create.

```
DISPLAY OPTIONS:

1. Print only Final Global Matrices
2. Print Element and Global Matrices
3. Print Details During Calculation

0. No Printing

... Select by number <Return>[1]>
```

Options 1-3 produce increasing amounts of detail; you can disable the output if PCE provides too much detail. Option 1 displays the final global matrices; option 2 displays the element matrices in addition. Option 3 supplements option 2 with the incremental changes as PCE constructs the global matrices by superposition of the element matrices. Use option 0 if you need no intermediate values.

3.21 Combine external force boundary conditions (Option 3).

• Type 3 and press <Enter>.

```
   DISPLAY OPTIONS
1. Original Global Force Vector
2. New Global Force Vector
3. Both Force Vectors

0. No Display

... Select by number <RETURN>[0]>
```

The options provide increasing detail.

• Type 0 and press <Enter>.

• Press <Enter> as needed to return to the Solve main menu.

3.22 Apply displacement boundary conditions (Option 4).

• Type 4 and press <Enter>.

```
   DISPLAY OPTIONS
1. Original Stiffness Matrix
2. Modified Stiffness Matrix
3. Original Force Vector
4. Modified Force Vector
5. All of the above (Opt 1 to 4)
6  Modified Stiffness & Force Matrices

0. No Display

... Select by number <RETURN>[0]>
```

You can disable the display as soon as you have the information you need.

3.23 Solve for nodal displacements (Option 5).

Type 5 and press <Enter>.

```
List node displacements as obtained [N]
```

PCE uses the Gauss elimination technique to decompose the system of equations. If you request the displacement output as obtained, the X and Y displacement components will be displayed in reverse node number order.

Note: You can display the values in ascending node order using option 6.

```
SOLVING FOR DISPLACEMENTS...
NODE           X- DISPLACEMENT                 Y-DISPLACEMENT
135         0.00000000000000E+0000       -5.83838763632162E+0000
134         0.00000000000000E+0000       -7.29654538879903E-0001
...          .....                        ....
Press <RETURN> to continue...
```

- Type "N" and press <Enter>.

3.24 List nodal displacements (Option 6).

```
NODE               X                              Y
1          -6.7496919E-0004               0.0000000E+0000
2          -1.4104233E-0002               0.0000000E+0000
...          .....                         ....
Press <RETURN> to continue...
```

- Press <Enter> as needed to complete the list and return to the Solve main menu (Fig 3.45).

To leave this module (Fig 3.45):

- Type 0 and press <Enter>.

```
SOLVE (4.00)                                                 EXIT

1. Proceed to 'POSTPROCESS'
2. Exit to Program Menu
3. Don't Exit; Remain in 'SOLVE'

0. Stop

... Select by number <RETURN> [1]>
```

Fig 3.47 Exit Solve menu

- Select 1 and press <Enter>.

3.25 Stresses and strains.

Using the nodal displacements obtained in the last section along with the mesh data from previous sections, you can now find the stress and strain within each element. Using the consistent element smoothing technique (Segerlind, 1976), you can also find smoothed nodal values for the stress components in the coordinate and principal directions. PCE also supplies the equivalent nodal reactions which would have produced the same results and the displaced nodal positions.

```
                         PC-ELASTIC

                      POSTPROCESS (5.00)

ABSTRACT:

Node displacements determined in 'SOLVE' are used to define element
strains, element stresses, and node stresses.  Stresses are defined in
coordinate and principal directions.

In addition node reactions and new node positions are determined.

Press <RETURN>[] to continue...
```

Fig 3.48 Postprocess abstract

• Press <Enter> to continue. PCE again reminds you of the project selected.

• Press <Enter> to continue. PCE retrieves the required data files prepared by previous modules and presents the Postprocess main menu (Fig 3.49).

```
POSTPROCESS  (5.00)                                        MAIN MENU

-- SOLVE ALL EQUATIONS --
1. Do all calculations (Menu Options 2-6)

-- SOLVE FOR STRAINS AND STRESSES --
2. Element Strains and Stresses
3. Element Principal Stresses
4. Nodal Principal Stresses

-- OTHER CALCULATIONS --
5. New Node Positions
6. Node Reactions

-- LIST RESULTS --
7. List Results

0. None of the Above

... Select by number <RETURN>[1]
```

Fig 3.49 Postprocess main menu

3.26 Run with minimal intervention (Option 1).

An experienced user would select 1 (Fig 3.49) to find the element stresses and strains, element principal stresses, and nodal principal stresses without user intervention and select option 1 below.

```
SOLVE (4.05)                              COMPLETE PROBLEM FORMULATION

OPTIONS:
1. Run WITHOUT user interaction (using all defaults)
2. Run WITH user interaction

... Select by number <RETURN> [1]>
```

Fig 3.50 Display options

- To run without intervention, press <Enter> (Fig 3.50).

Option 2 (Fig 2.33) asks if you wish to display element strains and stresses as obtained, principal stresses as obtained, conjugate matrices, nodal coordinate direction stresses as obtained, and principal stresses as obtained. As with the Solve menu, the first menu item allows you to complete all steps with one selection. Alternatively, you can use options 2-7 to examine various levels of intermediate calculations.

For instructional purposes you can handle options 2-5 separately. *If so, you must execute them in ascending order.*

3.27 Element strains and stresses (Option 2).

- Type 2 and press <Enter>.

```
Display STRAINS & STRESSES as obtained [N]
```

If you type Y and press <Enter> the following display appears:

```
CALCULATING ELEMENT STRAINS AND STRESSES

  FOR ELEMENT 1  OF 224

STRAINS
      X                Y                X-Y
9.2958E-0004    -2.4365E-0005    -1.4915E-0003

STRESSES
      X                Y                X-Y
6.9880E+0001   1.9284E+0001     -3.9553E+0001

Press <RETURN> to continue...
```

- Press <Enter> as required to return to the Postprocess main menu.

3.28 Element principal stresses (Option 3).

- Type 3 and press <Enter>.
- Press <Enter> after reading the files.

```
Display Principal Stresses?  [N]
```

If you type Y and press <Enter> the following element principal stress display appears:

```
SOLVING FOR ELEMENT 1  OF 224

         STRESSES
         MAX                     MIN                     SHEAR
9.15331239191961E+0001 -2.36905094967970E+0000 4.69510874344379E+0001

ANGLE CCW FROM X (DEG)
-2.86984716087537E+0001 6.13015283912463E+0001 1.63015283912463E+0001

Press <RETURN> to continue...
```

- Press <Enter> as needed to return to the Postprocess main menu.

3.29 Nodal coordinate stresses (Option 4).

- Type 4 and press <Enter>.

• Press <Enter> after reading the files.

```
Display Conjugate matrices?[N]
```

If you type Y and press <Enter>, the following display appears:

```
ASSEMBLING CONJUGATE MATRICES

   FOR ELEMENT 1  OF 224 ELEMENTS

ELEMENT CONJUGATE STIFFNESS
COLUMNS        45                46                54
ROW 45   2.688238E+0001    1.344119E+0001   1.344119E+0001
ROW 46   1.344119E+0001    2.688238E+0001   1.344119E+0001
ROW 54   1.344119E+0001    1.344119E+0001   2.688238E+0001

Press <RETURN> to continue...
```

• Press <Enter> to continue.

```
ASSEMBLING CONJUGATE MATRICES

   FOR ELEMENT 1  OF 224 ELEMENTS

UPDATED GLOBAL K[45,45] IS 2.68823831732143E+0001
UPDATED GLOBAL K[45,46] IS 1.34411915866071E+0001
UPDATED GLOBAL K[45,54] IS 1.34411915866071E+0001
UPDATED GLOBAL K[46,46] IS 2.68823831732143E+0001
UPDATED GLOBAL K[46,54] IS 1.34411915866071E+0001
UPDATED GLOBAL K[54,54] IS 2.68823831732143E+0001

Press <RETURN> to continue...
```

• Press <Enter> as needed to complete the total number of elements.

```
Display Nodal Stresses?[N]
```

• Type “Y” and press <Enter>.

```
NODAL STRESSES

NODE       X               Y                X-Y
1     1.813196E+0001  -1.964383E+0001  -3.997800E+0000
2     4.959096E-0001  -4.423728E+0001  -5.536701E+0000
...    ...             ...              ...

Press <RETURN> to continue...
```

- Press <Enter> as needed to return to the Postprocess main menu.

3.30 Nodal principal stresses (Option 5).

- Type 5 and press <Enter>.
- Press <Enter> after reading the files.

```
Display Principal Stresses?[N]
```

If you type Y and press <Enter>, the following display appears:

```
SOLVING FOR NODE 1  OF 135

                 STRESSES
          MAX                        MIN                        SHEAR
1.85504114391249E+0001 -2.00622851301295E+0001 1.93063482846272E+0001

ANGLE CCW FROM X (DEG)
-5.97540267042618E+0000 8.40245973295738E+0001 3.90245973295738E+0001

Press <RETURN> to continue...
```

- Press <Enter> as needed to return to the Postprocess main menu.

3.31 Node new positions (Option 6).

- Type 6 and press <Enter>.
- Press <Enter> as needed to return to the Postprocess main menu.

3.32 Node Reactions (Option 7).

- Type 7 and press <Enter>.

• Press <Enter> as needed to return to the Postprocess main menu.

To leave Postprocess,

• Type 0 and press <Enter>.

```
POSTPROCESS (5.00)                                              EXIT


1. Proceed to 'PLOT'
2. Exit to Program Menu
3. Don't Exit; Remain in 'POSTPROCESS'

0. Stop

... Select by number <RETURN> [1]>
```

Fig 3.51 Exit Postprocess

• Press <Enter> to advance to PLOT, the default.

3.33 Plot.

In the Plot module PCE creates plots of the input and output—mesh, boundary conditions, nodal displacements, element strains, element stresses, and nodal stresses. Line drawings, shaded plots and contour plots are supported, along with zooming for enlargements. Labeling of elements, nodes, contour lines and simple text are supported. You can send copies of the plots to a dot matrix printer ***if you executed the GRAPHICS.COM file before starting PCE***.

You can enter the Plot module from the main menu or as a continuation from the Postprocess module (Fig 3.51).

NAME PURPOSE Serial #

```
                            PC-ELASTIC

                            PLOT (6.00)

ABSTRACT:

Prepare finished plots of MESH, BOUNDARY CONDITIONS, or RESULTS.  Any
plot may be labeled and printed.  Zones may be selected to enlarge
parts of interest.

Press <RETURN>[]
```

Fig 3.52 Plot module abstract

- Press <Enter>, and PCE displays the keyword and problem description.
- Press <Enter> to continue, and PCE retrieves the required data files.

```
PLOT  (6.00)                                              MAIN MENU

-- PLOT FORMULATION --
1. Plot Generated Mesh
2. Plot Boundary Conditions


-- PLOT RESULTS --
3. Plot Nodal Displacements
4. Plot Element Strains
5. Plot Element Stresses
6. Plot Nodal Stresses

0. None of the Above

... Select by number <RETURN>[1]
```

Fig 3.53 Plot main menu

Examine each of the six plot options in detail.

3.34 Generated mesh.

- Select 1 and press <Enter> to plot the mesh.

```
PLOT  (6.01)                                                   EXIT

Whole body will be plotted; then ZONE plotting is offered.  After each
plot you will choose from the following labeling options:

 NO LABELS
 LABEL ELEMENTS
 LABEL NODES
 LABEL CONTOUR LINES (ONLY FOR MENU OPTION - NODE STRESSES)
 ADD ANY LABEL      LABEL ELEMENT VALUES

Note: Labels falling on lines or shaded regions will appear fuzzy.  To
clear this up, choose the redrawing option.

Press <RETURN> to continue...
```

Fig 3.54 Plot options

This screen reminds you of the features available.

• Press <Enter>.

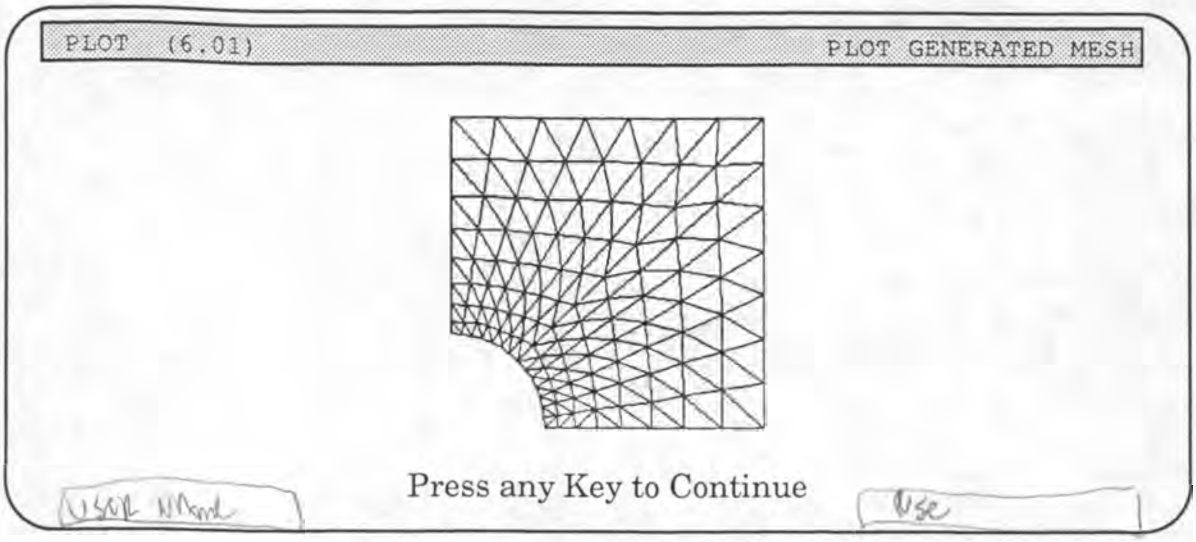


Fig 3.55 Mesh

• Press any key to continue.

• Type "Y" and press <Enter> to examine labeling capabilities. These same capabilities are available in each of the six plot menu options. If you type "N", you advance immediately to the zone plotting option below.

```
Label this plot  [N]?
```

Type "Y" and press <Enter> to label the elements.

```
Label ELEMENTS  [N]?
```

Type 1 and press <Enter> to label all elements. Too many labels produce a cluttered screen.

```
LABEL OPTIONS:
1. ALL ELEMENTS (Default positions)
2. Selected ELEMENTS (User positions)

0. None

... Select by number <RETURN>[0]>
```

Delay using this option until you

examine selective labeling.

• Press <Enter>. Later you will find that repositioning of labels improves clarity.

```
Reposition ELEMENT labels  [N]?
```

Press <Enter>.

This option redraws the labels by overwriting the mesh. The current plot is too cluttered for this to matter.

```
Redraw ELEMENT labels  [N]?
```

You can add text labels, but type "N" and press <Enter>.

```
Add another label to plot  [N]?
```

Let's examine the zoom capability to overcome the cluttering problem.

• Type 1 and press <Enter>

```
PLOT DESTINATION OPTIONS:
1. Proceed with zone plotting

0. End this plot

...Select by number <Enter>[0]
```

Two approaches to enlargement are possible. Either select a single rectangular portion (zone) of the screen for enlargement, or produce multiple zones of equal size in order to reassemble the plots into a mosaic.

```
Whole body plot is completed.
Plots of selected parts may now
be made.  You have the option
of a single or multiple equal
sized zones (from 1 to 36). You
can select any of these smaller
zones and have it enlarged into
a single plot.

Press any key to continue.
```

Let's examine both forms.

Press <Enter> to test single zone.

```
ZONE PLOTTING OPTIONS:
1. Single zone
2. Multiple equal-sized zones
0. None

...Select by number <RETURN>[1]
```

• Press any key to select a rectangular region for enlargement.

• Position the cursor to the lower left corner of the desired rectangular zone and press "S". Press shift key with arrow to re-

```
DEFINE LOWER LEFT CORNER
USE (SHIFT-) ARROW KEYS to move
cursor. <S> to select corner.

Press any key to continue.
```

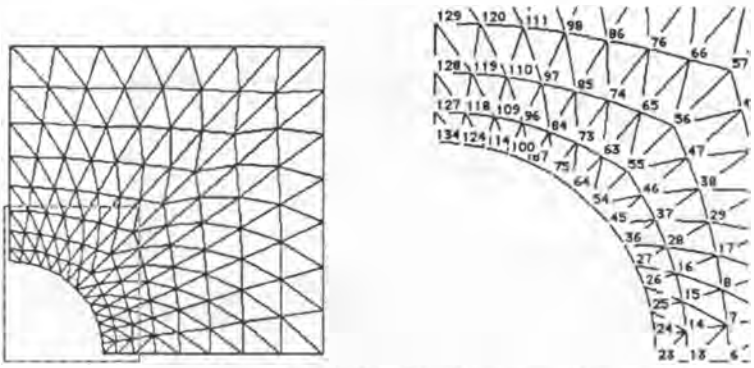
duce movement of cursor.

• Position the cursor to the upper right corner of the desired rectangular zone and press "S".

• Confirm choice by typing "Y" and pressing <Enter>.

```
DEFINE upper right corner
USE (SHIFT-) ARROW KEYS to move
cursor. <S> to select corner.

Press any key to continue.
```

Note: Press special function key 1 (F1) to remove an input window to view the full screen.



(redo)

Fig 3.56 Single zone enlargement

Now label the elements selectively.

• Press any key to continue.

You can reposition element labels, redraw element labels,redraw label nodes, and add a text label of up to 50 characters.
40

```
INSTRUCTIONS:
Use ARROWS to move cursor.
<S> SELECTS label to be reposi-
tioned. <X> ADDS/DELETES label.
ARROW KEYS move label.
Shift-ARROW KEY - small increments.
<Q> quits label repositioning.

Press any key to continue.
```

Alternative to a single zone, the multiple zone option produces the following possibilities.

• Type 2 and press <Enter>.

```
What magnification is
desired (integer (2 - 6) [2]>
```

The magnification is the number of rows (and columns) of equal size which will split the plot. These equal-sized zone plots can be grouped to form an enlarged mosaic of the original plot.

• Press <Enter> and the plot will be divided into four equal-sized zones.

```
OK?  (Y/N)
```

• To accept, type "Y" and press <Enter>. Otherwise, you can make another attempt.

```
ZONE PLOTTING OPTIONS:
1.Plot ALL zones   3.SELECT ZONES
2. OMIT zones      0. Plot NONE

... Select by number<RETURN>[1]>
```

Option 1 produces plots of all zones. Option 2 allows you to select the zones you want to neglect. Option 3 allows you to select the regions you want to plot. Option 0 produces no plots.

Illustrate with option 3.

```
<S> Selects zones INCLUDED IN
plotting. Use arrows to move
circle around the zones. Shaded
zones will be plotted. Press F1
to access plot. Use <Q> to quit.

Press Any Key to Continue
```

• Type 3 and press <Enter>.

• Press F1 to access the full plot.

• Select the zones and press "Q".

Fig 3.57 Zone selection

3.35 Boundary conditions.

The *current* default at the plot menu is now 2.

• Type 2 and press <Enter>.

PCE retrieves the required data files and presents the introductory list of options (Fig 3.54).

• Press <Enter> to resume.

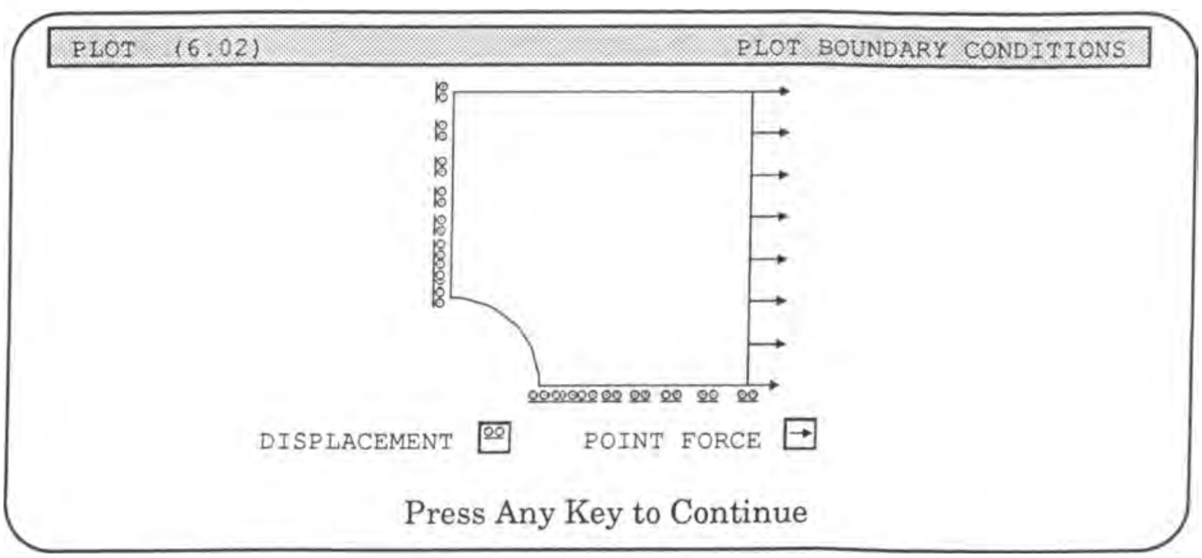


Fig 3.58 Boundary conditions

The various plotting and labeling options presented in Mesh plot are available.

3.36 Nodal displacements.

Option 3 produces a plot of the principal results—the displaced mesh.

• Type 3 and press <Enter>.

Again, use the default.

• Accept the default by pressing <Enter>.

Mesh Options:

1. Entire Mesh
2. Only boundary lines

~~0. None of these~~

...Select by number <Enter>[1]

no 0 - all one screen

In this window you can accept the default values again.

• Press <Enter> to accept the default values.

Line Type Options:

1. Original: dashed
   Displaced: solid
2. Original: solid
   Displaced: dashed

~~0. None of these~~

... Select by number <RETURN>[1]>

A reminder of the plot options (Fig 2.42) appears.

• Press <Enter> to resume.

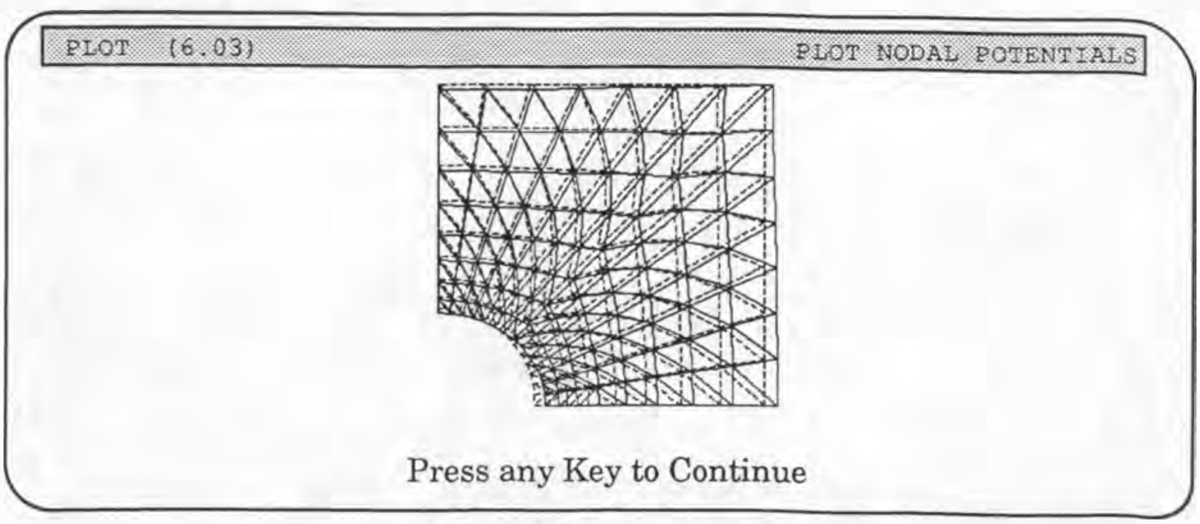


Fig 3.59 Nodal displacements

• Press <Enter> to continue.

The labeling and zoom capabilities were described with the mesh plot. Fig 3.60 shows the displacement of the boundary.

• Press <Enter> twice (Fig 2.42) to skip the advanced features and to return to the Plot main menu (Fig 2.39).

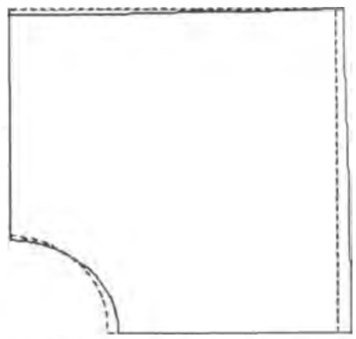


Fig 3.60 Displacement without mesh

3.37 Element strains.

A plot of the strain for each element provides an easily visualized characterization of the conditions. Shading density conveys a feel for the variation throughout the body.

• Type 4 and press <Enter>.

• Type 3 and press <Enter> to plot the x-component of element strain.

PCE displays the data minimum and maximum and the default increment size.

• Press <Enter> to accept the default.

```
Type: COORDINATE-DIRECTION

Component Options:
1. X-component
2. Y-component
3. X-Y shear

0. None of these

... Select by number <RETURN>[1]>
```

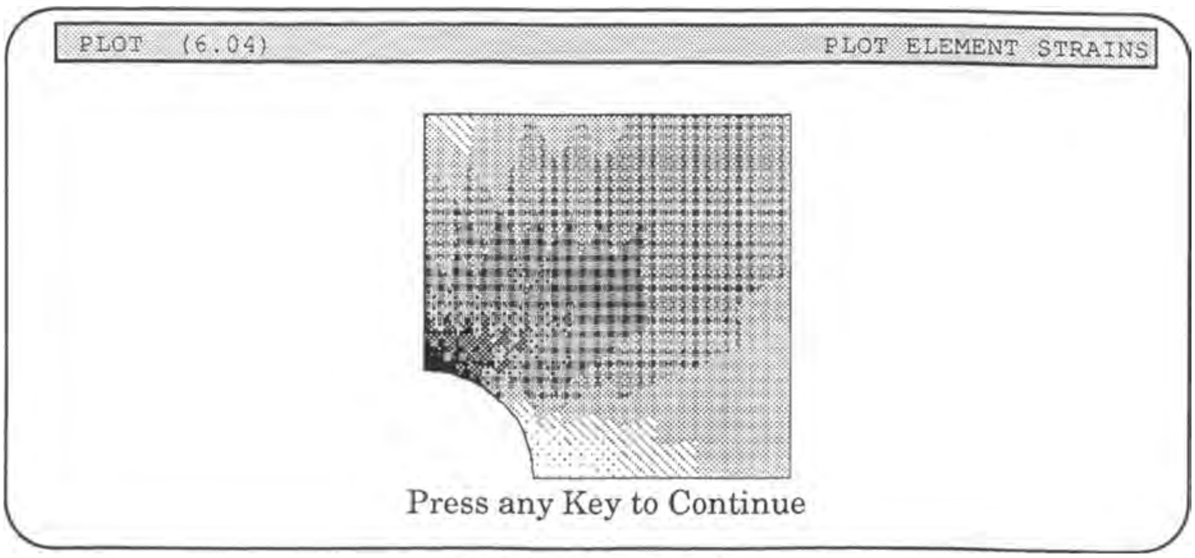


Fig 3.61 Element strain

• Press <Enter> to continue.

Again, examine the labeling and zoom capabilities in mesh.

• Press <Enter> twice (Fig 3.61) to skip the advanced features and to return to the component options .

• Type 0 and press <Enter> to return to the plot menu (Fig 3.53).

3.38 Element stresses.

PCE earlier computed the element stresses, and you can display the results as a shaded plot of constant stress components—x, y, and x-y components.

• Type 5 and press <Enter>.

The component options are the same as for element strains. To select the x-component:

• Type 1 and press <Enter>.

In this window you can accept the default values or request a window in which you can specify the values.

• Press <Enter> to accept the default values.

• Press <Enter> to have PCE present the plot.

```
Specify plot range and increment...
Data minimum: -6.75715937682517E+0000
Data maximum:  3.67313339623018E+0002

Default is full range with 8 increments
of 4.678812374980E+0001

Is this OK[Y}?
```

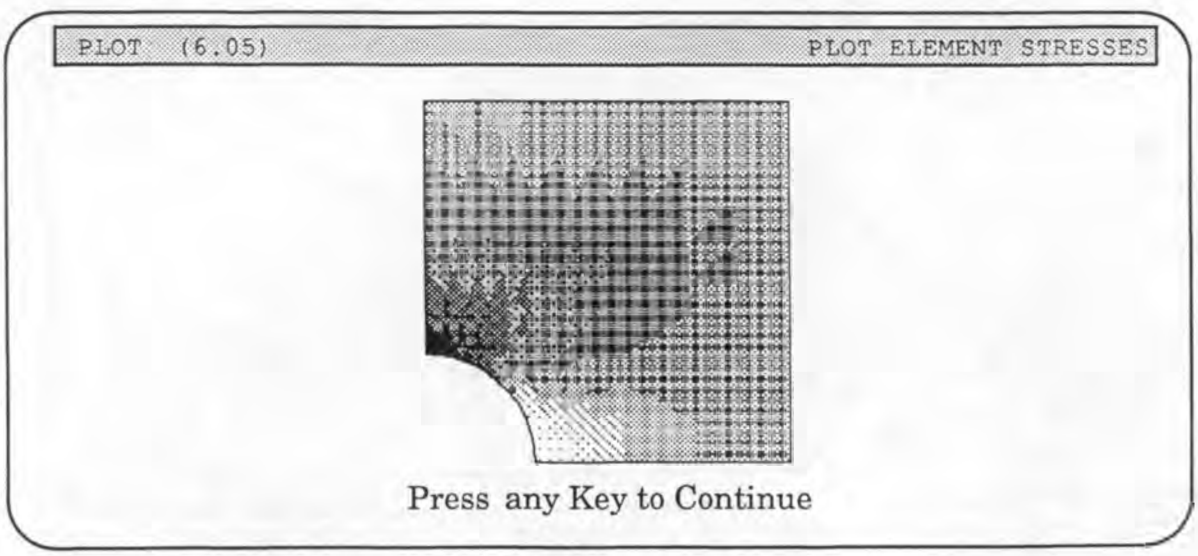


Fig 3.62 Element stress

• Press <Enter> twice to skip the advanced features and to return to the component options.

• Type 0 and press <Enter> to return to the plot menu (Fig 3.53).

3.39 Nodal stresses.

A contour plot of the nodal stresses—coordinate direction and principal—completes the visualization of the potential.

• Type 6 and press <Enter>.

Next you must select a stress option. For illustrative purposes use the default.

• Press <Enter>.

```
Stress Options:

1. COORDINATE-DIRECTION
2. PRINCIPAL

0. None of these

...Select by number <Enter>[1]>
```

Type 1 and press <Enter> to plot the X-component.

```
Type: COORDINATE-DIRECTION

Component Options:
1. X-component
2. Y-component
3. X-Y shear

0. None of these

... Select by number <RETURN>[1]>
```

Type 1 and press <Enter> to draw the plot without the mesh.

```
Type: COORDINATE-DIRECTION

Component: X

Contour Line Options:

1. Plot only contour lines
2. Plot contour lines with dashed mesh

... Select by number <RETURN>[1]>
```

PCE retrieves the required data files and presents a data range summary.

• Type "N" and press <Enter>.

```
Specify plot range and increment...
Data minimum: -8.29111911793080E+0000
Data maximum:  3.95791764733239E+0002

Default is full range with 10 incre-
ments of 4.04082883851170E+0001

Is this OK[Y]?
```

Supply rounded values for the minimum, maximum, and increment suitable for up to 10 contours. After you have entered the increment value, the list of plot options appears.

```
Minimum: [-8.29] > -50 0
Maximum: [395.79]> 400
Increment: [40.41]> 50 40
```

40.00

Calculates based on Minimum and Maximum input

goes back up here to give you a change to review

• Press <Enter> and PCE produces the plot.

NODAL

PLOT (6.06) PLOT ELEMENT STRESSES

Press any Key to Continue

Fig 3.63 Contour plot

• Press <Enter>. Label the contour lines.

• Type "y" and press <Enter> to examine labeling capabilities. These same capabilities are available in each of the six plot menu options.

```
Label this plot  [N]? y
```

Note: Upper or lower case ("y" or "Y") is acceptable.

• Press <Enter> to bypass element labels.

```
Label ELEMENTS  [N]?
```

• Press <Enter> to bypass node labels.

```
Label NODES  [N]?
```

• Type "y" and press <Enter> to label contour lines.

```
Label NODES  [N]?
```

• Type 2 and press <Enter>.

```
OPTIONS:
1. Use LETTER      0. None
2. Use VALUE

...Select by number<RETURN>[1]>
```

• Press any key. Note: F1 also removes an input window for an unobstructed view.

```
Use ARROWS to move cursor
<S> Select Contour Line
<F> Change Format Setting
<Q> Quit
 Press ANY KEY to Continue
```

• Press the first letter to set floating or exponential format.

• Press "D" or "N" to set the decimal places and numeral width.

• Press "Q" to return to the previous window.

```
Format Specification for Value Labels
Floating          Exponential
Decimal places: 1
Numeral width: 5
                 0.0
Press first letter of item to be changed
Q to quit
```

• Use the arrow keys to move among the contours.

• Press "S" to choose a contour line. The contour value is displayed beneath the figure.

Use the arrow keys to position the label.

```
Locate center of label using ARROW keys
Type <X> to delete label.
Type <Q> to quit.

Press Any Key to Continue
```

• If the label collides with the contour, use redraw to assure that the label overwrites the contour.

```
Redraw ELEMENT labels  [N]?
```

You can add text labels; but
• Type “y” and press <Enter>.

```
Add another label to plot  [N]?
```

• Type a text string and press <Enter>.

```
Label (up to 50 characters) =
```

• Use the cursor keys to position the screen cursor at the center of the desired text location. Finer cursor control is available if you also depress the shift key while using the cursor keys.

Note: Press F1 if you need to retrieve or remove the instruction window.

• Type “y” and press <Enter>.

```
Reposition label  [N]?
```

Use the arrow keys (and shift-arrows) to position the center of the label.

Note:
Use F1 to toggle the window.

```
Move label position using arrow keys
Type <Q> when label is in the desired
position.

Press Any Key to Continue
```

If desired,

• Type “N” to cause the label to overwrite the background.

```
Redraw labels [N]?
```

Returning to the Plot main menu requires several steps:

```
Add another label to plot [N]?
```

• Press <Enter> to return to the component options.

```
PLOT DESTINATION OPTIONS:
1. Proceed with zone plotting

0. End this plot

...Select by number <Enter>[0]
```

• Type “0” and press <Enter> to return to the stress options.

```
Type: COORDINATE-DIRECTION

Component Options:
1. X-component
2. Y-component
3. X-Y shear

0. None of these

... Select by number <RETURN>[1]>
```

• Type “0” and press <Enter> to return to the Plot main menu (Fig 3.53).

```
Stress Options:

1. COORDINATE-DIRECTION
2. PRINCIPAL

0. None of these

...Select by number <Enter>[1]>
```

This completes the review of the Plot module.

• Press <Enter> to return to the main menu.

```
PLOT  (6.08)                                          EXIT


1. Exit to Main Menu
2. Don't Exit; Remain in 'PLOT'

0. Stop

... Select by number <RETURN> [1]>
```

Fig 3.64 Plot exit menu

The remaining module provides file management, creates a master file for a new problem, duplicates an existing problem for modification, chooses an existing problem, displays the contents of the files of existing problems, or changes the level of user interaction.

3.40 File management.

The file management module provides tools to enable you to initiate new projects, to modify existing problems, and to examine the numerical results of existing projects. It provides the primary access to tabular results—screen and printed. *(Note: You prepare graphical results in the plot module, not the file management module.)* This module also creates a master data file to coordinate the creation of the project files. This master file maintains the status flags for each of the data files. If you wish to solve a problem which is a variation of an existing project, you can duplicate the problem formulation files in this module.

If you have not opened a project when you enter this module, the Library module allows you to open a project, create a new project, or duplicate an existing project and enable/disable the demo mode. In demo mode all file saving is disabled in order to allow you to explore a problem without concern about corrupting your data. Also, PCE maintains a status indicator for each file; any data change in a file which affects the validity of another file causes PCE to disable the dependent file in order to prevent incompatibilities.

From the main program menu:

• Type 7 and press <Enter>.

```
                              PC-ELASTIC

                           FILE MANAGEMENT

ABSTRACT:

Management of data files is controlled by a master file with the ex-
tension PCE.  You must initialize this file with the problem name and
problem description in order to begin an analysis.

Press <RETURN>[]
```

Fig 3.65 File management abstract

• Press <Enter> to continue.

The FILER MAIN MENU appears.

```
FILER (7.00)                                                MAIN MENU

1. Define problem and prepare master file
2. List Problem Keyword and Description
3. List all records with description and status
4. List contents of data files referenced by records
5. Duplicate an existing application
6. Choose an existing application
7. Change User Interaction Mode

0. None of the Above

... Select by number <RETURN>[1]
```

Fig 3.66 Filer main menu

Examine each of the Filer options.

3.41 Define problem and prepare master file.

• Select 1 and press <enter> to begin project definition.

```
Enter path for application [C:\PCEX]?
```

Fig 3.67 Path (including disk)

• Press <Enter> to create the project files in the current subdirectory. Otherwise, supply the desired subdirectory before pressing <Enter>.

```
Define the following:
  Problem Keyword (1 to 8 characters)>
```

Fig 3.68 Project name

• Supply up to eight characters to serve as the file name for the project and press <Enter>.

Blanks are not allowed! Follow the DOS conventions. You will be warned if you attempt to use an existing file name. Each of the data files associated with this project has this common name and a unique extension.

```
Problem description
   (1 to 50 characters)>
```

Fig 3.69 Project description

• Type the description and press <Enter>.

This text serves only as a reminder to you. Normally, you record details of the variation of the problem under study, e.g., mesh refinement, change in properties, etc.

PCE creates a file having the keyword as the name and PCE as the extension.

> NOTE: ALL DATA FILES MUST BE KEPT IN THE SAME SUBDIRECTORY. THE NAMES AND STATUS OF ALL DATA FILES ARE MAINTAINED IN THIS FILE SO DON'T ARBITRARILY REMOVE OR MODIFY ANY OF THE 24 FILES.

```
User Interaction Mode:

1. Maximal (no defaults)
2. Intermediate (some defaults)
3. Minimal (free run)

...Select by number <Enter>
```

Fig 3.70 User interaction level

The user interaction level controls the level of detail presented to you during execution. Use 3 if you are an experienced user who wishes to be asked for minimal information to solve a well-posed problem. Use 1 if you wish an opportunity to explore the deepest level of the algorithm, e.g., if you are specifically examining the algorithms PCE uses. Level 2 is intermediate to the extremes.

• Select 2 and press <Enter> to return to the Filer main menu (Fig 3.66).

3.42 List problem keyword and description.

This option echoes the problem keyword, coordinate type, and problem description of the current project.

Note: Use item 6 to change the choice of active application.

• Type 2 and press <Enter>.

```
PROBLEM KEYWORD: PLATE  (Cartesian)

PROBLEM DESCRIPTION: FLAT PLATE WITH HOLE SUBJECT TO
TENSILE LOADING
```

Fig 3.71 Problem description

• Press <Enter> to return to the Filer menu (Fig 3.66).

3.43 List all records with descriptions and status.

- Type 3 and press <Enter> twice to review the status of the data files.

```
FILER  (7.22)                              LIST FILE NAMES AND DESCRIPTIONS

Active file (BOLD)        Inactive file (NORMAL)
Letter in first PLATE indicates data type: I = integer; R = real.

RECORD NAME          DESCRIPTION
----------------------------------------------------------------------
R  1  C:PLATE.XY     COORDINATES OF INPUT DATA POINTS
I  2  C:PLATE.REG    NODE AND CONNECTIVITY DATA
I  3  C:PLATE.EL     NODE AND REGION DATA FOR ELEMENTS
R  4  C:PLATE.ND     COORDINATES AND ATTRIBUTES OF NODES
I  5  C:PLATE.REL    RENUMBER ELEMENT NODE NUMBERS
R  6  C:PLATE.RND    COORDINATES OF RENUMBERED NODES
I  7  C:PLATE.RLN    RENUMBERED NODES FOR UNIQUE LINES
I  8  C:PLATE.RBO    RENUMBERED BOUNDARY NODES AND ELEMENTS
R  9  C:PLATE.PRP    ELASTIC AND THERMAL PROPERTIES
R 10  C:PLATE.IBC    INPUT BOUNDARY CONDITION SPECIFICATIONS
R 11  C:PLATE.BC     NODAL D.O.F. BOUNDARY CONDITIONS
R 12  C:PLATE.IFO    INITIAL (THERMAL) FORCE VECTOR
R 13  C:PLATE.IST    INITIAL GLOBAL STIFFNESS MATRIX
R 14  C:PLATE.CFO    COMBINED INITIAL AND BOUNDARY FORCES

Press <Enter> to continue...
```

Fig 3.72a File status (part 1)

```
FILER  (7.22)                              LIST FILE NAMES AND DESCRIPTIONS

Active file (BOLD)        Inactive file (NORMAL)
Letter in first PLATE indicates data type: I = integer; R = real.

RECORD NAME          DESCRIPTION
----------------------------------------------------------------------
R 15  C:PLATE.MFO    FORCES MODIFIED BY DISPLACEMENT B.C.
R 16  C:PLATE.MST    STIFFNESS MODIFIED BY DISPLACEMENT B.C.
I 17  C:PLATE.NDD    NODAL DISPLACEMENT MAGNITUDES
R 18  C:PLATE.ESA    ELEMENT STRAINS IN COORDINATE DIRECTIONS
R 19  C:PLATE.ESE    ELEMENT STRESSES IN COORDINATE DIRECTIONS
R 20  C:PLATE.EPS    ELEMENT PRINCIPAL STRESSES AND DIRECTIONS
R 21  C:PLATE.NDS    NODAL STRESSES IN COORDINATE DIRECTIONS
R 22  C:PLATE.NPS    NODAL PRINCIPAL STRESSES AND DIRECTIONS
R 23  C:PLATE.NEW    NEW COORDINATES OF NODES
R 24  C:PLATE.REA    RESULTING REACTIONS AT NODES

Press <Enter> to continue...
```

Fig 3.72b File status (part 2)

• Press <Enter> to see part 2 and again to return to the Filer menu (Fig 2.57).

Refer to the Appendix for a description of the structure of each file.

The project status screen lists the 24 data files listed in the master file. File names consist of the project name and a period plus a descriptor which indicates the type of file.

The first column indicates whether the file contains real or integer data.

The second column provides the file number which corresponds to the order in which you created the files.

The third column indicates the file name.

The fourth column gives a descriptive file name.

The typeface indicates the file status and prevents the use of inconsistent data files. If you change the contents of an input file, PCE makes all files inactive which are now inconsistent and changes the file name from **bold** to normal print.

See Appendix 2 for a description of the contents of these files.

3.44 List contents of data files referenced by records.

• Type 4 and press <Enter> to select a file for examination.

```
Examine data file for which record (1-22): [1]>
```

Fig 3.73 File selection

• Type 1 and press <Enter>.

```
FILER (7.23)                                 LIST DATA FILE CONTENTS

Filename:              C:PLATE.XY
File description:      COORDINATES OF INPUT DATA POINTS
Rows represent         POINTS
Columns represent      DIMENSIONS
-----------------------------------------------------------------------
ROW  1: 150.00000000   0.00000000
ROW  2: 255.00000000   0.25000000
.........
ROW 12:   0.00000000   0.00000000

Press <Enter> to continue...
```

Fig 3.74 File 1 contents (partial)

Press <Enter> twice to resume.

```
Examine other data [Y]?
```

Fig 3.75 Continuation

- Type "Y" to return to Fig 3.73 or "N" to return to the Filer menu.

You can examine each of the 24 files. Refer to the Appendix for a discussion of the file structure should you wish to access the files using your own program.

3.45 Duplicate an existing file.

Use this option to duplicate the input files and the master file so you can perform a variation on the same project. You need to apply only your changes, not all the data for the entire project.

- Type 5 and press <Enter>.

```
Enter path for application [C:\PCEX]?
```

Fig 3.76 Path (including disk)

- Press <Enter> to create the project files in the current subdirectory. Otherwise, supply the desired path before pressing <Enter>.

PCE supplies a list of all projects contained in the subdirectory you specified.

```
The following applications are available:

1. COLUMN              2. PLATE

Choose an application by number:[1]>
```

Fig 3.77 Project selection

• Type the number and press <Enter>.

```
User Interaction Mode:

1. Maximal (no defaults)
2. Intermediate (some defaults)
3. Minimal (free run)
4. Demo with previous data

...Select by number <Enter>
```

Fig 3.78 User level

The user levels are the same as in Fig 3.70; but demo mode has been added. Use demo mode if you wish to suppress file saving. This option is useful if you are giving a class demonstration where you wish to explore various alternatives without risk to your data files. Otherwise, PCE designates dependent files as inactive and, therefore, unusable if you change values. This feature provides protection against incompatible data. In other words, if you change the size of an object, the calculated results based on the old values are no longer meaningful.

• Type 4 and press <Enter> to use the demo mode.

NOTE: Because file saving is suppressed, you cannot solve new problems in demo mode!

```
Enter path for application [C:\PCE\]?
```

Fig 3.79 Path (including disk)

• Press <Enter> to create the project files in the current subdirectory. Otherwise, supply the desired subdirectory before pressing <Enter>.

```
Define the following:
  Problem Keyword (1 to 8 characters)>
```

Fig 3.80 Project name

• Supply up to eight characters to serve as the file name for the project and press <Enter>.

Blanks are not allowed! Follow the DOS conventions. PCE will warn you if you attempt to use an existing file name. Each of the data files associated with this project has this common name and a unique extension.

```
Problem description
    (1 to 50 characters)>
```

Fig 3.81 Project description

• Type the description and press <Enter>.

This text serves only as a reminder to you. Normally, you record details of the variation of the problem under study, e.g., mesh refinement, change in properties, etc.

PCE returns you to the Filer menu.

3.46 Choose an existing application.

• Type 6 and press <Enter>.

```
Enter path for application [C:\PCE\]?
```

Fig 3.82 Path (including disk)

• Press <Enter> to create the project files in the current subdirectory. Otherwise, supply the desired subdirectory before pressing <Enter>.

PCE supplies a list of all projects contained in the subdirectory you specified.

```
The following applications are available:

1. COLUMN                    2. PLATE

Choose an application by number:[1]>
```

Fig 3.83 Project selection

• Type the number and press <Enter>.

```
User Interaction Mode:

1. Maximal (no defaults)
2. Intermediate (some defaults)
3. Minimal (free run)
4. Demo with previous data

...Select by number <Enter>
```

Fig 3.84 User level

The user levels are the same as in Fig 3.70. Demo mode has been added. Use demo mode if you wish to suppress file saving. This option is useful if you are giving a class demonstration where you wish to explore various alternatives without risk to your data files. Otherwise, if you change values, PCE designates dependent files as inactive and, therefore, unusable. This feature provides protection against incompatible data. In other words, if you change the size of an object, the calculated results based on the old values are no longer meaningful.

• Type the desired number and press <Enter> to return to the Filer main menu (Fig 3.66).

3.47 Change user interaction mode.

The user interaction level menu (Fig 3.84) is presented.

• Select the desired user interaction level and press <Enter>.

To leave this module:

• Select 0 and press <Enter>.

```
FILER (7.00)                                                  EXIT


1. Exit to PROGRAM MENU
2. Don't Exit; Remain in 'FILER'

0. Stop

... Select by number <RETURN> [1]>
```

Fig 3.85 Exit Filer menu

• Select 1 and press <enter> to return to the main menu (Fig 3.5).

Chapter 4

Computational Details

This chapter presents a brief overview of the computational details handled by MacElastic. Refer to *Applied Finite Element Analysis* (Segerlind, 1976,1984) and *The Finite Element Method* (Zienkiewicz, 1977) for additional details.

4.1 Function variation within an element.

In our use of the finite element method in elasticity, displacements are expressed as a continuous function within each element. We select both the shape of the element and the form of the displacement function, but they are related. Element shapes can be triangles or quadrilaterals with straight or curved sides. Straight-sided elements require a node at each vertex. Curved-sided elements require additional nodes to define the shape of the element sides.

4.1.1 Shape functions.

The form of the element displacement function is determined by the interpolation given by a set of shape functions. Commonly, shape functions are polynomials defined using the nodal coordinates, so the order of the polynomial depends on the number of nodes in the element. Here, we use the simplest element, a 3-node, straight-sided triangle.

The shape functions used for a 3-node triangular element are linear functions of the two problem coordinates. Thus, displacements are linear functions of the coordinates. Consider the two-dimensional element shown in Fig 4.1.

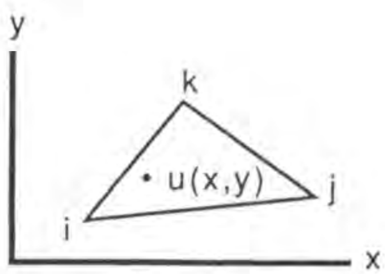


Fig 4.1. Representation of a 3-node triangular element

Let a displacement in a given direction be represented by the symbol u, which is a continuous function within this triangular element, i.e., u(x,y). Displacement values at the nodes i, j, and k are U_i, U_j, and U_k, respectively. Displacements throughout the element are defined as linear combinations of the node values:

$$u(x, y) = N_i(x, y)\,U_i + N_j(x, y)\,U_j + N_k(x, y)\,U_k \qquad 4.1$$

or, in matrix form,

$$u = [\, N_i \;\; N_j \;\; N_k \,] \begin{Bmatrix} U_i \\ U_j \\ U_k \end{Bmatrix} = [\, N \,]^T \{\, U \,\} \tag{4.2}$$

where N_i, N_j, N_k are shape functions[1] (functions of the coordinates) defined by

– coordinates of the nodes for the element and
– coordinates of the point of interest.

Shape functions used here are linear functions of x and y (or r and z for axisymmetric bodies) within an element, and are defined as:

$$\begin{aligned} N_i &= [\, a_i + b_i\, x + c_i\, y \,] / 2A \\ N_j &= [\, a_j + b_j\, x + c_j\, y \,] / 2A \\ N_k &= [\, a_k + b_k x + c_k\, y \,] / 2A \end{aligned} \tag{4.3}$$

where A is the area of the element
(x,y) are the coordinates of the point within the element.

In matrix form, the shape functions are a column vector:

$$[\, N \,] = (1 / 2A) \begin{Bmatrix} a_i + b_i x + c_i y \\ a_j + b_j x + c_j y \\ a_k + b_k x + c_k y \end{Bmatrix} \tag{4.4}$$

where constants are defined from the node coordinates as follows:

$$2A = X_j Y_k + X_i Y_j + X_k Y_i - X_j Y_i - X_k Y_j - X_i Y_k \tag{4.5}$$

$$\begin{aligned} a_i &= X_j Y_k - X_k Y_j \\ a_j &= X_k Y_i - X_i Y_k \\ a_k &= X_i Y_j - X_j Y_i \\[6pt] b_i &= Y_j - Y_k \\ b_j &= Y_k - Y_i \\ b_k &= Y_i - Y_j \\[6pt] c_i &= X_k - X_j \\ c_j &= X_i - X_k \\ c_k &= X_j - X_i \end{aligned} \tag{4.6}$$

[1] Segerlind, 1976, p26; Zienkiewicz, 1979, p23

Example 4.1 - Shape functions for a 3-node triangle.

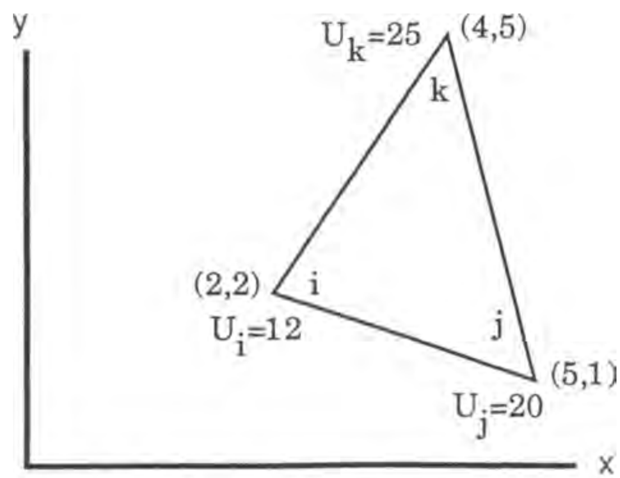


Fig 4.2. A 3-node triangular element example

Evaluate the shape functions for the element shown in Figure 4.2.

Coefficients in the shape functions are determined by substituting the nodal coordinates of this element into equations 4.4, 4.5 and 4.6:

$$2\,A = X_jY_k + X_iY_j + X_kY_i - X_jY_i - X_kY_j - X_iY_k$$
$$2\,A = (5)(5) + (2)(1) + (4)(2) - (5)(2) - (4)(1) - (2)(5) = 11$$

$$a_i = X_jY_k - X_kY_j = (5)(5) - (4)(1) = 21$$
$$a_j = X_kY_i - X_iY_k = (4)(2) - (2)(5) = -2$$
$$a_k = X_iY_j - X_jY_i = (2)(1) - (5)(2) = -8$$

$$b_i = Y_j - Y_k = 1 - 5 = -4$$
$$b_j = Y_k - Y_i = 5 - 2 = 3$$
$$b_k = Y_i - Y_j = 2 - 1 = 1$$

$$c_i = X_k - X_j = 4 - 5 = -1$$
$$c_j = X_i - X_k = 2 - 4 = -2$$
$$c_k = X_j - X_i = 5 - 2 = 3$$

For this element the shape functions evaluated using equation 4.3 are:

$$[\,N\,] = (\,1/2A\,)\begin{Bmatrix} a_i + b_ix + c_iy \\ a_j + b_jx + c_jy \\ a_k + b_kx + c_ky \end{Bmatrix} = (\,1/11\,)\begin{Bmatrix} 21 - 4x - y \\ -2 + 3x - 2y \\ -8 + x + 3y \end{Bmatrix}$$

Compute the shape function values at the nodes of this element:

◊ at node i (2,2)

$$
\begin{aligned}
N_i &= [\,21 - (4)(2) - (1)(2)]\,/\,11 &= 1\\
N_j &= [\,-2 + (3)(2) - (2)(2)]\,/\,11 &= 0\\
N_k &= [-8 + (1)(2) + (3)(2)]\,/\,11 &= 0
\end{aligned}
$$

◊ at node j (5,1)

$$
\begin{aligned}
N_i &= [\,21 - (4)(5) - (1)(1)]\,/\,11 &= 0\\
N_j &= [\,-2 + (3)(5) - (2)(1)]\,/\,11 &= 1\\
N_k &= [-8 + (1)(5) + (3)(1)]\,/\,11 &= 0
\end{aligned}
$$

◊ at node k (4,5)

$$
\begin{aligned}
N_i &= [\,21 - (4)(4) - (1)(5)]\,/\,11 &= 0\\
N_j &= [\,-2 + (3)(4) - (2)(5)]\,/\,11 &= 0\\
N_k &= [-8 + (1)(4) + (3)(5)]\,/\,11 &= 1
\end{aligned}
$$

This illustrates an important property of element shape functions: they are unity at their node and zero at the other two nodes. Elsewhere in the element they have values between zero and one.

4.1.2 Computing displacements.

A displacement within an element is defined by the product of the shape functions and the displacements at the nodes:

$$
u = [\,N\,]^T \{\,U\,\} = [\,N_i \;\; N_j \;\; N_k\,] \begin{Bmatrix} U_i \\ U_j \\ U_k \end{Bmatrix} \tag{4.7}
$$

Thus, a check of the shape function coefficients is this:

◊ at node i, u must equal U_i, which requires that

$$
\begin{aligned}
N_i &= 1\\
N_j &= 0\\
N_k &= 0
\end{aligned}
$$

◊ at node j, u must equal U_j, which requires that

$$
\begin{aligned}
N_i &= 0\\
N_j &= 1\\
N_k &= 0
\end{aligned}
$$

◊ at node k, u must equal U_k, which requires that

$N_i = 0$
$N_j = 0$
$N_k = 1$.

The shape functions for this element, being unity at their node and zero at the other nodes, produces the proper interpolated displacement values at the nodes. You can use these shape functions to determine the displacement u(x,y) at any point in this element by substituting the coordinates of the selected point into the shape function equations.

Example 4.2 - Displacement at a point.

For the element used in the previous example, determine the displacement at the interior point with coordinates (3,3). Now compute the shape functions at this point using the element shape functions obtained in the previous example and coordinates (3,3):

$$[\,N\,] = (\,1/11\,)\begin{Bmatrix} 21-(4)(3)-(1)(3) \\ -2+(3)(3)-(2)(3) \\ -8+(1)(3)+(3)(3) \end{Bmatrix} = (1/11\,)\begin{Bmatrix} 6 \\ 1 \\ 4 \end{Bmatrix}$$

and the displacement is defined by equation 4.7:

$$u = [\,N\,]^T\,\{\,U\,\} = (\,1/11\,)\,[\,6\;\;1\;\;4\,]\begin{Bmatrix} 12 \\ 20 \\ 25 \end{Bmatrix} = 17.45$$

Thus, using the shape function and the known displacement values at the nodes (U_i, U_j U_k), you determine the displacement value u(3,3) to be 17.45. Likewise, you can determine displacement values at all points within any element using the shape functions for the selected element and the displacement values at the nodes of that element.

4.1.3 Derivatives of functions.

The finite element equations for elasticity presented below are based on derivatives of the displacements[2]. Recall that the displacement functions within a 3-node triangular element are linear with respect to each coordinate direction and are used to obtain displacements from:

$$u = [N]\,\{U\} = (1/2A)\,[(a_i + b_i x + c_i y)\;(a_j + b_j x + c_j y)\;(a_k + b_k x + c_k y)]\,\{U\} \qquad 4.8$$

The partial derivative of u(x,y) with respect to x is:

[2] Segerlind, 1984,p138; Zienkiewicz,1979, p423

$$\begin{Bmatrix} \partial u / \partial x \\ \partial u / \partial y \end{Bmatrix} = \begin{Bmatrix} \partial N / \partial x \\ \partial N / \partial y \end{Bmatrix} \{U\} = [B] \begin{Bmatrix} U_i \\ U_j \\ U_k \end{Bmatrix} \qquad 4.9$$

where [B] is the shape function derivative matrix given by

$$[B] = \begin{Bmatrix} \partial N/\partial x \\ \partial N/\partial y \end{Bmatrix} = \frac{1}{2\,A} \begin{bmatrix} b_i & b_j & b_k \\ c_i & c_j & c_k \end{bmatrix} \qquad 4.10$$

This illustrates that the derivatives of the displacements in an element are constants. We show strains and stresses within an element to be functions of the derivatives of displacements[3]. Thus, because the derivatives are constant, the element strains and stresses are too. This also requires that you divide your problem into many elements when 3-node triangles are used if strain or stress are of interest to you. We discuss the calculation of strains and stresses in a later section.

Example 4.3 - Derivatives of displacement for a 3-node triangle.

Evaluate the derivatives of the displacement within the element shown in Figure 4.2. To do this use equations 4.9 and 4.10 together with coefficients calculated in example 4.1:

$$\begin{Bmatrix} \partial u / \partial x \\ \partial u / \partial y \end{Bmatrix} = (1/11) \begin{bmatrix} -4 & 3 & 1 \\ -1 & -2 & 3 \end{bmatrix} \begin{Bmatrix} 12 \\ 20 \\ 25 \end{Bmatrix} = \begin{Bmatrix} 3.36 \\ 2.09 \end{Bmatrix}$$

Within this element, the displacement gradient is constant as indicated by the vector shown in Figure 4.3.

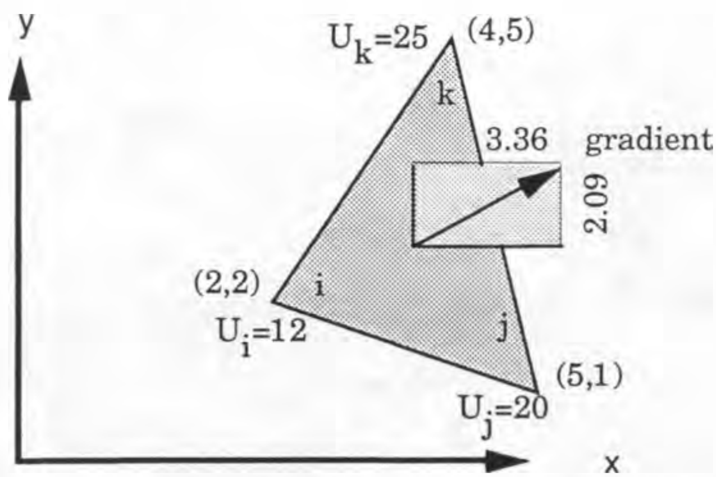


Fig 4.3. Derivatives of the displacement in an element

[3] Segerlind, 1984, p288ff or Zienkiewicz, 1979, p22ff

4.2 Finite element equation formulation.

The general form of finite element equations is a matrix product of a square stiffness (coefficient) matrix and a column displacement vector set equal to a column force vector:

$$\underset{n\times n}{[K]}\ \underset{n\times 1}{\{U\}} = \underset{n\times 1}{\{F\}} \quad \text{(rows x columns)} \qquad 4.11$$

where

[K] is a square, symmetric "stiffness" matrix,
{U} is the "unknown"displacement vector (not same as in 4.8),
{F} is the "force" vector, and
n is the number of "unknown" displacements.

A matrix equation of this type is formed for each element and combined into a global equation for the entire problem domain. After you form the global matrix equation for the entire problem domain, you apply boundary conditions to modify the equations. Solution of the modified equations yields the displacements for the given problem and boundary conditions. Then, if desired, you can evaluate element strains and stresses and nodal strains and stresses.

In 2-dimensional (and axisymmetric) elasticity problems, each node has two "unknown" displacement components. For 2-dimensional problems, these correspond to the x and y component displacements as shown in Fig 4.4. Symbols i, j, and k denote node numbers, and subscripts of the displacement components are functions of the node numbers. Odd subscripts refer to x components and even subscripts refer to y components of the displacements U.

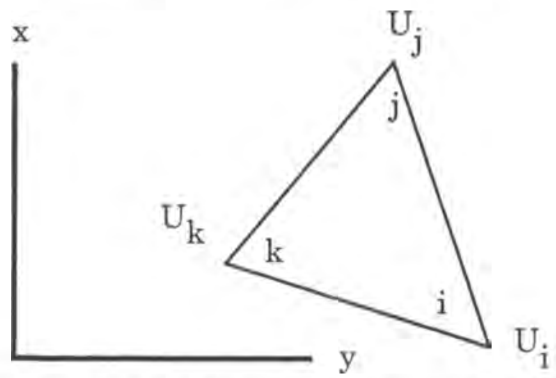


Fig 4.4. Notation for nodal displacement values

Because the unknown displacement at each node is a vector quantity and has two components, the matrix equation for each element has six "unknown" nodal displacements. You evaluate the 2-dimensional displacement at a point in the element from the shape functions and nodal displacements as follows:

$$\underset{1\times 1}{u} = \underset{1\times 3}{\{N\}^T}\ \underset{3\times 1}{\{U\}} \quad \text{(rows x columns)} \qquad 4.12$$

where

$$\{U\} = \begin{Bmatrix} U_{2i-1} \\ U_{2i} \\ U_{2j-1} \\ U_{2j} \\ U_{2k-1} \\ U_{2k} \end{Bmatrix}, \ [N] = \begin{bmatrix} N_i & 0 \\ 0 & N_i \\ N_j & 0 \\ 0 & N_j \\ N_k & 0 \\ 0 & N_k \end{bmatrix}, \text{ and } \{u\} = \begin{Bmatrix} u_x \\ u_y \end{Bmatrix} \qquad 4.13$$

and N_i, N_j, N_k are the linear shape functions defined in equation 4.3.

4.2.1 Two-dimensional Cartesian coordinate systems.

For elasticity in 2-dimensional Cartesian (x,y) coordinate systems, the strain in an element has three components:

$$\{\varepsilon\} = \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{xy} \end{Bmatrix} \qquad 4.14$$

and can be defined as derivatives of nodal displacements:

$$\overset{3\times1}{\{\ \varepsilon\ \}} = \overset{3\times6}{[\ B\ ]}\overset{6\times1}{\{\ U\ \}} \quad \text{(rows} \times \text{columns)} \qquad 4.15$$

where the shape function derivative matrix is:

$$[B] = (1/2A)\begin{pmatrix} b_i & 0 & b_j & 0 & b_k & 0 \\ 0 & c_i & 0 & c_j & 0 & c_k \\ c_i & b_i & c_j & b_j & c_k & b_k \end{pmatrix} \qquad 4.16$$

Stresses in the element are defined from strains and material properties[4]

$$\overset{3\times1}{\{\ \sigma\ \}} = \overset{3\times3}{[\ D\ ]}\overset{3\times1}{\{\ \varepsilon\ \}} \quad \text{(rows} \times \text{columns)} \qquad 4.17$$

where [D] is the material property matrix.

The material property matrix depends on the choice of assumptions used to achieve a 2-dimensional problem.

• For plane stress (loads applied only in the plane of the body), the material property matrix is:

[4] Segerlind, 1984, p287; Zienkiewicz, 1979, p28

$$[D] = \left(\frac{E}{1-\mu^2}\right)\begin{bmatrix} 1 & \mu & 0 \\ \mu & 1 & 0 \\ 0 & 0 & \frac{1-\mu}{2} \end{bmatrix} \qquad 4.18$$

• For plane strain (body constrained so no displacement can occur perpendicular to the plane of the body), the material property matrix is:

$$[D] = \left(\frac{E}{1+\mu}\right)\begin{bmatrix} \frac{1-\mu}{1-2\mu} & \frac{\mu}{1-2\mu} & 0 \\ \frac{\mu}{1-2\mu} & \frac{1-\mu}{1-2\mu} & 0 \\ 0 & 0 & \frac{1}{2} \end{bmatrix} \qquad 4.19$$

where μ is Poisson's ratio and
E is Young's modulus.

The finite element matrix equation describing 2-dimensional elasticity in a 3-node linear triangular element is:

$$\overset{6\times 6}{[\, t\ A\ [B]^T\, [\, D\,]\, [B]\,]}\ \overset{6\times 1}{\{\, U\, \}} = \overset{6\times 1}{\{\, F\, \}} \quad \text{(rows} \times \text{columns)} \qquad 4.20$$

which matches the form of equation 4.11. The force matrix {F} is zero prior to application of boundary conditions. The element stiffness matrix for 2-dimensional elasticity is the 6-by-6 matrix evaluated by the matrix multiplications shown in the first term of equation 4.20. The unknown displacements are two component nodal values; thus, 3-node triangular elements have 6 degrees of freedom (6 unknowns).

The finite element equations presented are valid for any consistent set of units. For SI (metric) systems, forces, displacements, and stiffnesses can be expressed in Newtons, meters, and Newtons/meter, respectively. Stresses and elastic moduli are defined in Pascals.

If we use local node numbers for nodes i, j, and k equal to 1, 2, and 3, respectively, then subscripts for the unknowns become 1 through 6; and equation 4.20 can be expanded to the following matrix equation:

$$\begin{bmatrix} k_{1,1} & k_{1,2} & k_{1,3} & k_{1,4} & k_{1,5} & k_{1,6} \\ k_{2,1} & k_{2,2} & k_{2,3} & k_{2,4} & k_{2,5} & k_{2,6} \\ k_{3,1} & k_{3,2} & k_{3,3} & k_{3,4} & k_{3,5} & k_{3,6} \\ k_{4,1} & k_{4,2} & k_{4,3} & k_{4,4} & k_{4,5} & k_{4,6} \\ k_{5,1} & k_{5,2} & k_{5,3} & k_{5,4} & k_{5,5} & k_{5,6} \\ k_{6,1} & k_{6,2} & k_{6,3} & k_{6,4} & k_{6,5} & k_{6,6} \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \\ U_5 \\ U_6 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \\ F_5 \\ F_6 \end{Bmatrix} \qquad 4.21$$

Each of the six displacements represents a degree of freedom (dof) for this ment. U_1 is the x-displacement of node 1 (dof=1), U_2 is the y-displacement of node 1 (dof=2), U_3 is the x-displacement of node 2 (dof=3), etc. Each stiffness coefficient, $k_{i,j}$, represents the force that must be applied at one node to produce a unit displacement at another node. The first subscript denotes the degree of freedom of the force, and the second subscript denotes the displacement degree of freedom. Thus, $k_{1,2}$ is the x-direction force applied at node 1 (dof=1) that produces a unit y-direction displacement of node 1 (dof=2); $k_{3,5}$ is the x-direction force applied at node 2 (dof=3) to produce a unit x-direction displacement of node 3 (dof=5).

Example 4.4 - Finite element equation formulation.

Determine the element stiffness matrix for the element shown below. Assume plane stress conditions. Material properties are:

E = 50 x 103 Pa
μ = 0.3
t = 0.1 m

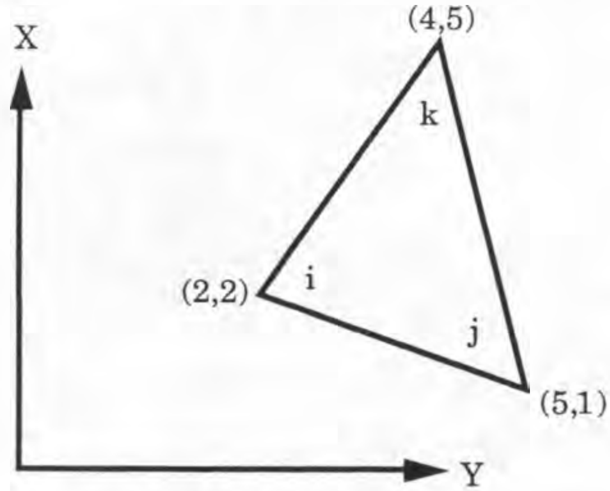


Fig 4.5 Material properties

For this 2-dimensional element, the shape function derivative matrix is defined by equation 4.16 together with coefficients determined in Example 1:

$$[B] = (1/2A)\begin{bmatrix} b_i & 0 & b_j & 0 & b_k & 0 \\ 0 & c_i & 0 & c_j & 0 & c_k \\ c_i & b_i & c_j & b_j & c_k & b_k \end{bmatrix} = (1/11)\begin{bmatrix} -4 & 0 & 3 & 0 & 1 & 0 \\ 0 & -1 & 0 & -2 & 0 & 3 \\ -1 & -4 & -2 & 3 & 3 & 1 \end{bmatrix} \qquad 4.22$$

The material property matrix is defined by equation 4.18:

$$[D] = \left(\frac{E}{1-\mu^2}\right)\begin{bmatrix} 1 & \mu & 0 \\ \mu & 1 & 0 \\ 0 & 0 & \frac{1-\mu}{2} \end{bmatrix} = 54{,}945\begin{bmatrix} 1 & .3 & 0 \\ .3 & 1 & 0 \\ 0 & 0 & .35 \end{bmatrix} \qquad 4.23$$

The element stiffness matrix is determined using the matrix products in the first term of equation 4.20, $[k^{(e)}] = t\ A\,[B]^T\,[D]\,[B]$, which, by grouping constants, is

$$[k^{(e)}] = 499.5 \begin{bmatrix} -4 & 0 & -1 \\ 0 & -1 & -4 \\ 3 & 0 & -2 \\ 0 & -2 & 3 \\ 1 & 0 & 3 \\ 0 & 3 & 1 \end{bmatrix} \begin{bmatrix} 1 & .3 & 0 \\ .3 & 1 & 0 \\ 0 & 0 & .35 \end{bmatrix} \begin{bmatrix} -4 & 0 & 3 & 0 & 1 & 0 \\ 0 & -1 & 0 & -2 & 0 & 3 \\ -1 & -4 & -2 & 3 & 3 & 1 \end{bmatrix} \tag{4.24}$$

Substituting values from above yields the following **6 x 6** element stiffness matrix:

$$[k^{(e)}] = (499.5) \begin{bmatrix} \mathbf{16.35} & 2.6 & -11.3 & 1.35 & -5.05 & -3.95 \\ 2.6 & \mathbf{6.6} & 1.9 & -2.2 & -4.5 & -4.4 \\ -11.3 & 1.9 & \mathbf{10.4} & -3.9 & 0.9 & 2.0 \\ 1.35 & -2.2 & -3.9 & \mathbf{7.15} & 2.55 & -4.95 \\ -5.05 & -4.5 & 0.9 & 2.55 & \mathbf{4.15} & 1.95 \\ -3.95 & -4.4 & 2.0 & -4.95 & 1.95 & 9.35 \end{bmatrix} \tag{4.25}$$

Note that the diagonal coefficients (in **bold** type) are positive and have the largest magnitude in each row and column. Also note that the coefficients which are symmetric about the diagonal are equal, e.g., $k_{1,3} = k_{3,1} = -11.3$. The sum of coefficients in any row or in any column is equal to zero. These attributes exist in all of the element stiffness matrices and in the global stiffness matrix produced by assembling the element matrices.

If you number nodes i, j, and k for the element of interest locally as 1, 2, and 3, respectively, the finite element equation for this element becomes:

$$(499.5) \begin{bmatrix} \mathbf{16.35} & 2.6 & -11.3 & 1.35 & -5.05 & -3.95 \\ 2.6 & \mathbf{6.6} & 1.9 & -2.2 & -4.5 & -4.4 \\ -11.3 & 1.9 & \mathbf{10.4} & -3.9 & 0.9 & 2.0 \\ 1.35 & -2.2 & -3.9 & \mathbf{7.15} & 2.55 & -4.95 \\ -5.05 & -4.5 & 0.9 & 2.55 & \mathbf{4.15} & 1.95 \\ -3.95 & -4.4 & 2.0 & -4.95 & 1.95 & \mathbf{9.35} \end{bmatrix} \begin{Bmatrix} U_5 \\ U_6 \\ U_9 \\ U_{10} \\ U_{13} \\ U_{14} \end{Bmatrix} = \begin{Bmatrix} F_5 \\ F_6 \\ F_9 \\ F_{10} \\ F_{13} \\ F_{14} \end{Bmatrix} \tag{4.26}$$

The force matrix is zero because no forces are applied to this element. The boundary conditions are added only after all of the element equations have been assembled into a global matrix equation.

4.2.2 Axisymmetric systems.

For axisymmetric (r-z coordinate) systems, the 3-node triangular element is a triangular toroid (a triangle of rotation about the z-axis). In axisymmetric coordinate systems, the strain in an element has four components[5]

[5] Segerlind, 1984, p314ff; Zienkiewicz, 1979, p119ff

$$\{\varepsilon\} = \begin{Bmatrix} \varepsilon_{rr} \\ \varepsilon_{\theta\theta} \\ \varepsilon_{zz} \\ \varepsilon_{rz} \end{Bmatrix} \qquad 4.27$$

and can be defined as derivatives of node displacements:

$$\underset{4\times1}{\{\ \varepsilon\ \}} = \underset{4\times6}{[\ \bar{B}\ ]}\underset{6\times1}{\{\ U\ \}} \qquad 4.28$$

where the shape function derivative matrix (evaluated at the element centroid) is:

$$[\bar{B}] = (1/2A)\begin{bmatrix} b_i & 0 & b_j & 0 & b_k & 0 \\ 0 & c_i & 0 & c_j & 0 & c_k \\ \frac{N_i}{\bar{r}} & 0 & \frac{N_j}{\bar{r}} & 0 & \frac{N_k}{\bar{r}} & 0 \\ c_i & b_i & c_j & b_j & c_k & b_k \end{bmatrix} \qquad 4.29$$

where $\bar{r}$ is the r-coordinate of the element centroid.

Stresses in the element are defined from strains and material properties.

$$\underset{4\times1}{\{\ \sigma\ \}} = \underset{4\times4}{[\ D\ ]}\underset{4\times1}{\{\ \varepsilon\ \}} \qquad 4.30$$

where [D] is the material property matrix.

For axisymmetric geometries, the material property matrix is:

$$[D] = \left(\frac{E\,(1-\mu)}{(1+\mu)(1-2\mu)}\right)\begin{bmatrix} 1 & \frac{\mu}{1-\mu} & \frac{\mu}{1-\mu} & 0 \\ \frac{\mu}{1-\mu} & 1 & \frac{\mu}{1-\mu} & 0 \\ \frac{\mu}{1-\mu} & \frac{\mu}{1-\mu} & 1 & 0 \\ 0 & 0 & 0 & \frac{1-2\mu}{2\,(1-2\mu)} \end{bmatrix} \qquad 4.31$$

You can evaluate the finite element equations at the element centroids with minimal error if the elements are small[6] .

The finite element matrix equation defining axisymmetric elasticity in a 3-node triangular element is:

$$\underset{6\times6}{[\ 2\,\pi\,\bar{r}\,A\,[\ \bar{B}\ ]^T\,[D]\,[\bar{B}]\ ]}\underset{6\times1}{\{\ \bar{U}\ \}} = \underset{6\times1}{\{\ F\ \}} \qquad 4.32$$

Note that A is the element area, $2\pi\,\bar{r}$ is an equivalent thickness for the element, $\bar{r} = (R_i+R_j+R_k)/3$, and $[\bar{B}]$ is the shape function derivative matrix evaluated at the element centroid.

[6] Segerlind, 1976, p201

The form of equation 4.32 matches the form of equation 4.11. Note that this element equation has the same dimensions as does equation 4.20 for 2-dimensional Cartesian coordinate systems even though the dimensions of [D], [B], {ε}, and {σ} differ in these two coordinate systems.

4.2.3 Review of element equations for elasticity.

◊ Element shapes and sizes can differ because nodes need not be on a regular grid. Nodes i, j, and k are at coordinates: (X_i, Y_i), (X_j, Y_j), and (X_k, Y_k), respectively.

◊ Unknowns are defined by continuous polynomial shape functions in an element. For 3-node triangular elements, the shape functions are linear with respect to x and y (or r and z):

$$u(x,y) = [N(x,y)]\,\{U\}$$

◊ Finite element equations are defined in a common matrix form:

$$[K]\,\{U\} = \{F\}$$

◊ The element stiffness matrix is a function of node coordinates (in A & [B]) and material properties (in [D]):

$$[K] = t\,A\,[B]^T\,[D]\,[B]$$

◊ The strain is defined within an element as:

$$\{\varepsilon\} = [B\}\,\{U\}$$

and is constant throughout an element.

◊ The stress is defined within an element as:

$$\{\sigma\} = [D]\,\{\varepsilon\}$$

◊ Yet to be discussed are:
- ◊ Mapping from element coordinates to global coordinates
- ◊ Combining element equations into a global matrix equation
- ◊ Applying boundary conditions to the global equations
- ◊ Solving for "unknown" displacement components at the nodes

4.3 Global matrices.

A finite element equation of the form of equation 4.33 exists for each element in the problem domain.

$$[k^{(e)}]\,\{u^{(e)}\} = \{f^{(e)}\} \qquad 4.33$$

where the superscript (e) indicates the element number.

The solution must satisfy all of these equations simultaneously to develop a solution to the problem posed. Thus, it assembles a larger matrix equation (a global matrix equation) from the individual element equations to establish the total set of conditions that govern the problem solution. The global matrix equation must have a number of rows equal to the number of degrees of freedom (unknowns) for the problem; for elasticity problems, this is equal to twice the number of nodes.

The program systematically combines equations from individual elements to develop a global set of equations. First, it establishes the global matrix equation to provide the number of rows (and columns in the global stiffness matrix) needed for all degrees of freedom. Then it inserts element equations into the global matrices by adding stiffness coefficients and sources to values in their respective global locations. We illustrate this later.

The number of simultaneous equations existing for each element is equal to the number of degrees of freedom (unknowns) for the element. A 3-node triangular element has six degrees of freedom if two unknowns occur at each node as is the case with two-dimensional elasticity (and three degrees of freedom if one unknown exists at each node as with Poisson's equation).

Figure 4.6 shows a body with elements 1, 2, and 3 and nodes 1, 2, 3, 4, and 5. Because two unknown displacements exist at each node, you need six equations for each element. Degree of freedom numbers are based on the node number and the displacement component as defined in Table 4.1.

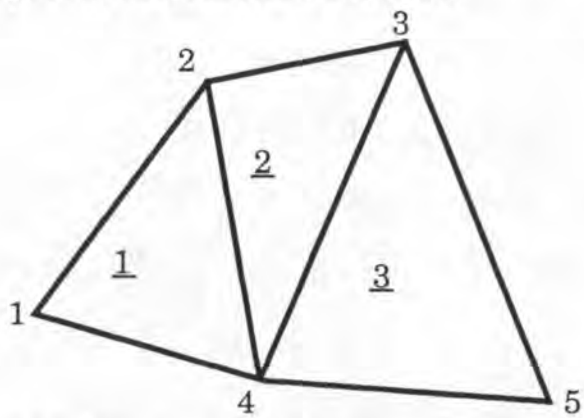


Fig 4.6. A body with three 3-node triangular elements

Table 4.1 Degree of Freedom Definitions for Problem in Fig 4.6.

DOF	Node	Direction
1	1	x
2	1	y
3	2	x
4	2	y
5	3	x
6	3	y
7	4	x
8	4	y
9	5	x
10	5	y

For element 1, containing nodes 1, 4, and 2, the element equation is:

$$\begin{bmatrix} k_{1,1}^{(1)} & k_{1,2}^{(1)} & k_{1,3}^{(1)} & k_{1,4}^{(1)} & k_{1,5}^{(1)} & k_{1,6}^{(1)} \\ k_{2,1}^{(1)} & k_{2,2}^{(1)} & k_{2,3}^{(1)} & k_{2,4}^{(1)} & k_{2,5}^{(1)} & k_{2,6}^{(1)} \\ k_{3,1}^{(1)} & k_{3,2}^{(1)} & k_{3,3}^{(1)} & k_{3,4}^{(1)} & k_{3,5}^{(1)} & k_{3,6}^{(1)} \\ k_{4,1}^{(1)} & k_{4,2}^{(1)} & k_{4,3}^{(1)} & k_{4,4}^{(1)} & k_{4,5}^{(1)} & k_{4,6}^{(1)} \\ k_{5,1}^{(1)} & k_{5,2}^{(1)} & k_{5,3}^{(1)} & k_{5,4}^{(1)} & k_{5,5}^{(1)} & k_{5,6}^{(1)} \\ k_{6,1}^{(1)} & k_{6,2}^{(1)} & k_{6,3}^{(1)} & k_{6,4}^{(1)} & k_{5,6}^{(1)} & k_{6,6}^{(1)} \end{bmatrix} \begin{Bmatrix} u_1^{(1)} \\ u_2^{(1)} \\ u_3^{(1)} \\ u_4^{(1)} \\ u_7^{(1)} \\ u_8^{(1)} \end{Bmatrix} = \begin{Bmatrix} f_1^{(1)} \\ f_2^{(1)} \\ f_3^{(1)} \\ f_4^{(1)} \\ f_7^{(1)} \\ f_8^{(1)} \end{Bmatrix} \qquad 4.34$$

Values of the stiffness coefficients depend on the geometry and material properties of the element as discussed earlier but are designated only by symbols for this illustration. Note that a complete matrix equation would include rows and columns for degrees of freedom 5 and 6; because these coefficients are zero for element 1, they are not shown here. The total number of columns required to display this element stiffness matrix (including intermediate zero values) is called the bandwidth (BW) of the element stiffness matrix, given by:

$$BW = (\text{highest dof}) - (\text{lowest dof}) + 1$$

which gives for element 1, $\quad BW^{(1)} = 8 - 1 + 1 = 8$

Thus, in a global matrix format, element 1 equations would require 8 rows, and the stiffness matrix would have 8 rows and 8 columns if the intermediate row and column were included.

For element 2, containing nodes 2, 4, and 3, the element equation is:

$$\begin{bmatrix} k_{3,3}^{(2)} & k_{3,4}^{(2)} & k_{3,5}^{(2)} & k_{3,6}^{(2)} & k_{3,7}^{(2)} & k_{3,8}^{(2)} \\ k_{4,3}^{(2)} & k_{4,4}^{(2)} & k_{4,5}^{(2)} & k_{4,6}^{(2)} & k_{4,7}^{(2)} & k_{4,8}^{(2)} \\ k_{5,3}^{(2)} & k_{5,4}^{(2)} & k_{5,5}^{(2)} & k_{5,6}^{(2)} & k_{5,7}^{(2)} & k_{5,8}^{(2)} \\ k_{6,3}^{(2)} & k_{6,4}^{(2)} & k_{6,5}^{(2)} & k_{6,6}^{(2)} & k_{6,7}^{(2)} & k_{6,8}^{(2)} \\ k_{7,3}^{(2)} & k_{7,4}^{(2)} & k_{7,5}^{(2)} & k_{7,6}^{(2)} & k_{7,7}^{(2)} & k_{7,8}^{(2)} \\ k_{8,3}^{(2)} & k_{8,4}^{(2)} & k_{8,5}^{(2)} & k_{8,6}^{(2)} & k_{8,7}^{(2)} & k_{8,8}^{(2)} \end{bmatrix} \begin{Bmatrix} u_3^{(2)} \\ u_4^{(2)} \\ u_5^{(2)} \\ u_6^{(2)} \\ u_7^{(2)} \\ u_8^{(2)} \end{Bmatrix} = \begin{Bmatrix} f_3^{(2)} \\ f_4^{(2)} \\ f_5^{(2)} \\ f_6^{(2)} \\ f_7^{(2)} \\ f_8^{(2)} \end{Bmatrix} \qquad 4.35$$

The bandwidth for element 2 is

$$BW^{(2)} = 8 - 3 + 1 = 6.$$

Because the node numbers for element 2 are consecutive, this bandwidth is the minimum that can occur for a 3-node triangular element with two degrees of freedom per node.

For element 3, containing nodes 3, 4, and 5, the element equation is:

$$\begin{bmatrix} k_{5,5}^{(3)} & k_{5,6}^{(3)} & k_{5,7}^{(3)} & k_{5,8}^{(3)} & k_{5,9}^{(3)} & k_{5,10}^{(3)} \\ k_{6,5}^{(3)} & k_{6,6}^{(3)} & k_{6,7}^{(3)} & k_{6,8}^{(3)} & k_{6,9}^{(3)} & k_{6,10}^{(3)} \\ k_{7,5}^{(3)} & k_{7,6}^{(3)} & k_{7,7}^{(3)} & k_{7,8}^{(3)} & k_{7,9}^{(3)} & k_{7,10}^{(3)} \\ k_{8,5}^{(3)} & k_{8,6}^{(3)} & k_{8,7}^{(3)} & k_{8,8}^{(3)} & k_{8,9}^{(3)} & k_{8,10}^{(3)} \\ k_{9,5}^{(3)} & k_{9,6}^{(3)} & k_{9,7}^{(3)} & k_{9,8}^{(3)} & k_{9,9}^{(3)} & k_{9,10}^{(3)} \\ k_{10,5}^{(3)} & k_{10,6}^{(3)} & k_{10,7}^{(3)} & k_{10,8}^{(3)} & k_{10,9}^{(3)} & k_{10,10}^{(3)} \end{bmatrix} \begin{Bmatrix} u_5^{(3)} \\ u_6^{(3)} \\ u_7^{(3)} \\ u_8^{(3)} \\ u_9^{(3)} \\ u_{10}^{(3)} \end{Bmatrix} = \begin{Bmatrix} f_5^{(3)} \\ f_6^{(3)} \\ f_7^{(3)} \\ f_8^{(3)} \\ f_9^{(3)} \\ f_{10}^{(3)} \end{Bmatrix} \qquad 4.36$$

The bandwidth for element 3 is

$$BW^{(3)} = 10 - 5 + 1 = 6.$$

The global matrix equation for the body being considered has ten degrees of freedom (five nodes with two dof each). The values in the global matrices are the sums of the corresponding values in all of the element matrices. For example, for a 3-element body, the force component in the ith row of the global force matrix is

$$f_i = f_i^{(1)} + f_i^{(2)} + f_i^{(3)} \qquad 4.37$$

and the global stiffness coefficient in the ith row and jth column is

$$k_{i,j} = k_{i,j}^{(1)} + k_{i,j}^{(2)} + k_{i,j}^{(3)} \qquad 4.38$$

Because the unknown displacements for each degree of freedom are the same regardless of the element equation from which it arises, they appear in the global equation without superscripts. Thus, the global matrix equation for the 3-element body being considered becomes

$$\begin{bmatrix} \mathbf{k_{1,1}} & \mathbf{k_{1,2}} & \mathbf{k_{1,3}} & \mathbf{k_{1,4}} & \mathbf{0} & \mathbf{0} & \mathbf{k_{1,7}} & \mathbf{k_{1,8}} & 0 & 0 \\ k_{2,1} & \mathbf{k_{2,2}} & \mathbf{k_{2,3}} & \mathbf{k_{2,4}} & \mathbf{0} & \mathbf{0} & \mathbf{k_{2,7}} & \mathbf{k_{2,8}} & \mathbf{0} & 0 \\ k_{3,1} & k_{3,2} & \mathbf{k_{3,3}} & \mathbf{k_{3,4}} & \mathbf{k_{3,5}} & \mathbf{k_{3,6}} & \mathbf{k_{3,7}} & \mathbf{k_{3,8}} & \mathbf{0} & \mathbf{0} \\ k_{4,1} & k_{4,2} & k_{4,3} & \mathbf{k_{4,4}} & \mathbf{k_{4,5}} & \mathbf{k_{4,6}} & \mathbf{k_{4,7}} & \mathbf{k_{4,8}} & \mathbf{0} & \mathbf{0} \\ 0 & 0 & k_{5,3} & k_{5,4} & \mathbf{k_{5,5}} & \mathbf{k_{5,6}} & \mathbf{k_{5,7}} & \mathbf{k_{5,8}} & \mathbf{k_{5,9}} & \mathbf{k_{5,10}} \\ 0 & 0 & k_{6,3} & k_{6,4} & k_{6,5} & \mathbf{k_{6,6}} & \mathbf{k_{6,7}} & \mathbf{k_{6,8}} & \mathbf{k_{6,9}} & \mathbf{k_{6,10}} \\ k_{7,1} & k_{7,2} & k_{7,3} & k_{7,4} & k_{7,5} & k_{7,6} & \mathbf{k_{7,7}} & \mathbf{k_{7,8}} & \mathbf{k_{7,9}} & \mathbf{k_{7,10}} \\ k_{8,1} & k_{8,2} & k_{8,3} & k_{8,4} & k_{8,5} & k_{8,6} & k_{8,7} & \mathbf{k_{8,8}} & \mathbf{k_{8,9}} & \mathbf{k_{8,10}} \\ 0 & 0 & 0 & 0 & k_{9,5} & k_{9,6} & k_{9,7} & k_{9,8} & \mathbf{k_{9,9}} & \mathbf{k_{9,10}} \\ 0 & 0 & 0 & 0 & k_{10,5} & k_{10,6} & k_{10,7} & k_{10,8} & k_{10,9} & \mathbf{k_{10,10}} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ u_5 \\ u_6 \\ u_7 \\ u_8 \\ u_9 \\ u_{10} \end{Bmatrix} = \begin{Bmatrix} f_1 \\ f_2 \\ f_3 \\ f_4 \\ f_5 \\ f_6 \\ f_7 \\ f_8 \\ f_9 \\ f_{10} \end{Bmatrix} \qquad 4.39$$

The primary diagonal of the global stiffness matrix includes coefficients $k_{1,1}, k_{2,2}, k_{3,3}, \cdots, k_{10,10}$. Coefficients with reversed subscripts are symmetric about the primary diagonal and are numerically equal:
$k_{2,1} = k_{1,2}$, $k_{2,6} = k_{6,2}$, $k_{4,8} = k_{8,4}$, etc. Note that the global stiffness matrix contains zeros (also symmetric about the primary diagonal) where no nonzero element stiffness terms occurred (e.g., no element contained nodes with degrees of freedom 1 and 10, so $k_{1,10} = k_{10,1} = 0$). Note also that all nonzero coefficients lie within a band about the primary diagonal. Matrices of this type are called banded matrices.

Because the global stiffness matrix is a banded symmetric matrix, only the diagonal and half of the nonzero numbers (shown in bold type in equation 4.39) are unique. The bandwidth of the global stiffness matrix is the minimum number of columns required to retain all unique numbers in the stiffness matrix. By condensing the global stiffness matrix as shown in equation 4.40, the bandwidth becomes less than the number of degrees of freedom; thus, 8 rather than 10 columns are kept to retain all unique stiffness coefficients.

$$[\tilde{k}] = \begin{bmatrix}
\tilde{\mathbf{k}}_{1,1} & \tilde{\mathbf{k}}_{1,2} & \tilde{\mathbf{k}}_{1,3} & \tilde{\mathbf{k}}_{1,4} & \mathbf{0} & \mathbf{0} & \tilde{\mathbf{k}}_{1,7} & \tilde{\mathbf{k}}_{1,8} \\
\tilde{\mathbf{k}}_{2,2} & \tilde{\mathbf{k}}_{2,3} & \tilde{\mathbf{k}}_{2,4} & \mathbf{0} & \mathbf{0} & \tilde{\mathbf{k}}_{2,7} & \tilde{\mathbf{k}}_{2,8} & \mathbf{0} \\
\tilde{\mathbf{k}}_{3,3} & \tilde{\mathbf{k}}_{3,4} & \tilde{\mathbf{k}}_{3,5} & \tilde{\mathbf{k}}_{3,6} & \tilde{\mathbf{k}}_{3,7} & \tilde{\mathbf{k}}_{3,8} & \mathbf{0} & \mathbf{0} \\
\tilde{\mathbf{k}}_{4,4} & \tilde{\mathbf{k}}_{4,5} & \tilde{\mathbf{k}}_{4,6} & \tilde{\mathbf{k}}_{4,7} & \tilde{\mathbf{k}}_{4,8} & \mathbf{0} & \mathbf{0} & 0 \\
\tilde{\mathbf{k}}_{5,5} & \tilde{\mathbf{k}}_{5,6} & \tilde{\mathbf{k}}_{5,8} & \tilde{\mathbf{k}}_{5,8} & \tilde{\mathbf{k}}_{5,9} & \tilde{\mathbf{k}}_{5,10} & 0 & 0 \\
\tilde{\mathbf{k}}_{6,6} & \tilde{\mathbf{k}}_{6,7} & \tilde{\mathbf{k}}_{6,8} & \tilde{\mathbf{k}}_{6,9} & \tilde{\mathbf{k}}_{6,10} & 0 & 0 & 0 \\
\tilde{\mathbf{k}}_{7,7} & \tilde{\mathbf{k}}_{7,8} & \tilde{\mathbf{k}}_{7,9} & \tilde{\mathbf{k}}_{7,10} & 0 & 0 & 0 & 0 \\
\tilde{\mathbf{k}}_{8,8} & \tilde{\mathbf{k}}_{8,9} & \tilde{\mathbf{k}}_{8,10} & 0 & 0 & 0 & 0 & 0 \\
\tilde{\mathbf{k}}_{9,9} & \tilde{\mathbf{k}}_{9,10} & 0 & 0 & 0 & 0 & 0 & 0 \\
\tilde{\mathbf{k}}_{10,10} & 0 & 0 & 0 & 0 & 0 & 0 & 0
\end{bmatrix} \qquad 4.40$$

The bandwidth for the global stiffness matrix as shown in equations 4.39 and 4.40 for the 3-element body is 4.

$$BW = 8 - 1 + 1 = 8$$

The numbers in the condensed stiffness matrix (equation 4.40) are defined so that the primary diagonal of the original matrix becomes the first column of the condensed matrix; subsequent diagonals are subsequent columns in the condensed matrix, e.g.,

$\tilde{k}_{1,1} = k_{1,1}$, $\tilde{k}_{2,1} = k_{2,2}$, $\tilde{k}_{3,1} = k_{3,3}$, $\tilde{k}_{1,2} = k_{1,2}$, $\tilde{k}_{2,2} = k_{2,3}$, $\tilde{k}_{3,2} = k_{3,4}$, etc.

Here, the condensed matrix contains only 80 (10 x 8) numbers as compared to the 100 (10 x 10) numbers in the original stiffness matrix. The amount of space saved by condensing the matrix in this manner depends on the number of unknowns and the manner in which nodes were numbered.

The bandwidth of the global stiffness matrix (and the number of columns in the condensed matrix) depends on where the nonzero coefficients occur in the global stiffness matrix, which depends on the bandwidths of the included element stiffness matrices. The largest element bandwidth becomes the bandwidth of the global stiffness matrix. Larger bandwidths require more computer memory and longer computation times; therefore, it is important that you number nodes in a manner that minimizes the bandwidth.

When you manually assign node numbers, you can achieve smaller bandwidths by numbering nodes sequentially in the direction with the fewest nodes. For example, consider the mesh in Figure 4.7 with two different node numbering schemes. Numbering first along the longer side (side with the most nodes) as shown on the left results in a larger bandwidth than numbering parallel to the shorter side. In more complex geometries or in automatic node numbering approaches, node renumbering algorithms can be used to reduce the bandwidth after the entire mesh has been defined. ME uses the Collins renumbering algorithm to reduce the bandwidth of the global stiffness matrix before the element equations are assembled to produce the condensed stiffness matrix. Global stiffness matrices are stored in the condensed format.

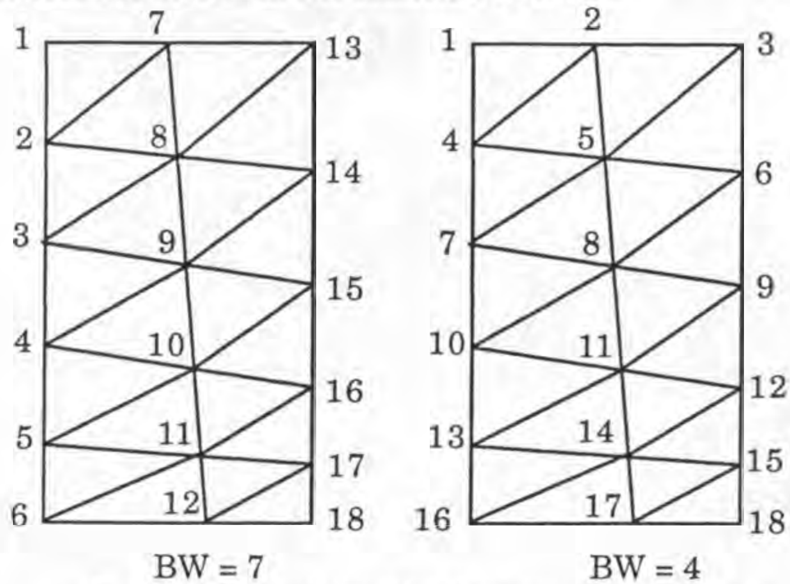


Fig 4.7. Bandwidths resulting from two node numberings

Example 4.5 - Global stiffness matrix assembly.

A mesh has numerous elements with a total of 50 degrees of freedom. The element stiffness matrices for elements 1 and 2 are given below. Combine these two element stiffness matrices as a first step in assembling a global stiffness matrix.

$$
\begin{array}{cc}
 & \text{dof} \\
[k^{(1)}] = \begin{bmatrix} 12 & 3 & -7 & 1 & -6 & -3 \\ 3 & 10 & 4 & -6 & -4 & -7 \\ -7 & 4 & 10 & -5 & -3 & 1 \\ 1 & -6 & -5 & 6 & 3 & 1 \\ -6 & -4 & -3 & 3 & 11 & -1 \\ -3 & -7 & 1 & 1 & -1 & 9 \end{bmatrix} & \begin{matrix} 1 \\ 2 \\ 5 \\ 6 \\ 7 \\ 8 \end{matrix}
\end{array}
$$

$$
\begin{array}{cc}
[k^{(2)}] = \begin{bmatrix} 6 & -1 & -3 & 1 & 1 & -4 \\ -1 & 9 & -2 & 1 & -3 & -4 \\ -3 & -2 & 7 & 1 & -4 & 1 \\ 1 & 1 & 1 & 5 & -2 & -6 \\ 1 & -3 & -4 & -2 & 9 & -1 \\ -4 & -4 & 1 & -6 & -1 & 14 \end{bmatrix} & \begin{array}{c} \text{dof} \\ 3 \\ 4 \\ 5 \\ 6 \\ 9 \\ 10 \end{array}
\end{array}
$$

The global stiffness matrix has 50 degrees of freedom, the first 10 of which are shown below. All remaining coefficients are zero at this time. You obtain the first coefficient in the global stiffness matrix, $k_{1,1}$, by adding the $k_{1,1}^{(1)}$ coefficient from the element 1 matrix ($k_{1,1}^{(1)} = 12$) to that from element 2 ($k_{1,1}^{(2)} = 0$), yielding $k_{1,1} = 12 + 0 = 12$. (The element 2 matrix has a zero value for $k_{1,1}^{(2)}$ and is not shown because element 1 does not have dof #1.) Both element matrices have degree of freedom 6; therefore, $k_{6,5} = k_{6,5}^{(1)} + k_{6,5}^{(2)} = -5 + 1 = -4$. You can obtain all other coefficients in the global stiffness matrix similarly by adding corresponding coefficients from the element matrices.

$$
\begin{array}{cc}
\begin{bmatrix} 12 & 3 & 0 & 0 & -7 & 1 & -6 & -3 & 0 & 0 & 0 & \cdots \\ 3 & 10 & 0 & 0 & 4 & -6 & -4 & -7 & 0 & 0 & 0 & \cdots \\ 0 & 0 & 6 & -1 & -3 & 1 & 0 & 0 & 1 & -4 & 0 & \cdots \\ 0 & 0 & -1 & 9 & -2 & 1 & 0 & 0 & -3 & -4 & 0 & \cdots \\ -7 & 4 & -3 & -2 & 17 & -4 & -3 & 1 & -4 & 1 & 0 & \cdots \\ 1 & -6 & 1 & 1 & -4 & 11 & 3 & 1 & -2 & -6 & 0 & \cdots \\ -6 & -4 & 0 & 0 & -3 & 3 & 11 & -1 & 0 & 0 & 0 & \cdots \\ -3 & -7 & 0 & 0 & 1 & 1 & -1 & 9 & 0 & 0 & 0 & \cdots \\ 0 & 0 & 1 & -3 & -4 & -2 & 0 & 0 & 9 & -1 & 0 & \cdots \\ 0 & 0 & -4 & -4 & 1 & -6 & 0 & 0 & -1 & 14 & 0 & \cdots \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & \cdots \\ \cdots & \cdots & \cdots & \cdots & \cdots & \cdots & \cdots & \cdots & \cdots & \cdots & \cdots & \cdots \end{bmatrix} & \begin{array}{c} \text{dof} \\ 1 \\ 2 \\ 3 \\ 4 \\ 5 \\ 6 \\ 7 \\ 8 \\ 9 \\ 10 \\ 11 \\ \end{array}
\end{array}
$$

4.4 Material properties.

Material properties used in elasticity problems include the following:

- E elastic (Young's) modulus of the material
- μ Poisson's ratio
- t thickness (when not axisymmetric)
- α coefficient of thermal expansion
- ΔT temperature difference, relative to a reference.

You can use any consistent set of units for these properties. We present example units for material properties in Table 4.2.

Table 4.2 Units for material properties

Property	*SI Units*	*English Units*
Elastic modulus	Pascals	Pounds/inch2
Poisson's ratio	--	--
Thickness	meters	inches
Coeff. thermal expansion	1/°C	1/°F
Temperature difference	°C	°F

You may specify each property for each element, thus allowing widely varying property variations throughout the body. Assume Young's modulus, Poisson's ratio, and the coefficient of thermal expansion to be isotropic (equal in all directions). The coefficient of thermal expansion and temperature difference specify initial thermal strains in the material, a part of the boundary conditions. The temperature difference is the temperature of the material above some reference temperature at which no thermal stresses exist in the material.

4.4.1 Two-dimensional Cartesian system thermal strains.

Incorporate thermal strains into the finite element equations by defining fictitious point forces at the nodes of the elements under thermal strain. The element thermal force vector for Cartesian coordinate systems is [7]

$$\{F_t^{(e)}\} = t\,A\,[B]^T\,[D]\,\{\varepsilon_0\} \qquad 4.41$$

where, the thermal strain, $\{\varepsilon_0\}$, is

$$\{\varepsilon_0\} = \alpha\,\Delta T \begin{Bmatrix} 1 \\ 1 \\ 0 \end{Bmatrix} \qquad 4.42$$

for plane stress, and

$$\{\varepsilon_0\} = (1+\mu)\,\alpha\,\Delta T \begin{Bmatrix} 1 \\ 1 \\ 0 \end{Bmatrix} \qquad 4.43$$

for plane strain.

The element thermal force vector has six components (force in two directions acting at each of the three nodes).

4.4.2 Axisymmetric system thermal strain.

In axisymmetric systems with thermally induced stresses, the equations are modified in a manner similar to that in the two-dimensional case. The element force equation resulting from thermal strains is:

[7] Segerlind, 1984, p294

$$\{F_t^{(e)}\} = 2\,\pi\,\bar{r}\,A\,[\bar{B}]\,[D]\,\{\varepsilon_0\} \tag{4.44}$$

where A is the element cross-sectional area, $\bar{r}$ is the r-coordinate of the element centroid, and $[\bar{B}]$ is the shape function derivative matrix evaluated at the element centroid. The thermal strain, $\{\varepsilon_0\}$, is

$$\{\varepsilon_0\} = \alpha\,\Delta T \begin{Bmatrix} 1 \\ 1 \\ 1 \\ 0 \end{Bmatrix} \tag{4.45}$$

Even though the $[\bar{B}]$ and [D] matrices for axisymmetric systems differ in dimension from those for Cartesian coordinate systems, the element thermal force vector has six components, two directions for each of the three nodes.

Example 4.6 - Element thermal strains.

Consider the element used in Example 4.4 with material properties:

$E = 50 \times 10^3$ Pa
$\mu = 0.3$
$t = 0.1$ m
$\alpha = 5 \times 10^{-6}$ 1/°C
$T = 5$ °C

Assuming plane stress conditions (Equation 4.42), the thermal strain is:

$$\{\varepsilon_0\} = \alpha\,\Delta T \begin{Bmatrix} 1 \\ 1 \\ 0 \end{Bmatrix} = (5\times10^{-6})\,(5) \begin{Bmatrix} 1 \\ 1 \\ 0 \end{Bmatrix} = 2.5 \times 10^{-5} \begin{Bmatrix} 1 \\ 1 \\ 0 \end{Bmatrix}$$

and from equation 4.41 , the thermal force vector for this element is:

$$\{F_t^{(e)}\} = t\ A\ [B]^T\,[D]\,\{\varepsilon_0\}$$

Using previously determined matrix values (from Example 4.4) and grouping constants,

$$= (.1)\,(11)\,(1/11)\,(54{,}945)\,(2.5\times10^{-5}) \begin{bmatrix} -4 & 0 & -1 \\ 0 & -1 & -4 \\ 3 & 0 & -2 \\ 0 & -2 & 3 \\ 1 & 0 & 0 \\ 0 & 3 & 3 \end{bmatrix} \begin{bmatrix} 1 & .3 & 0 \\ .3 & 1 & 0 \\ 0 & 0 & .35 \end{bmatrix} \begin{Bmatrix} 1 \\ 1 \\ 0 \end{Bmatrix}$$

which yields the element force vector:

$$\{F_t^{(e)}\} = (0.1374)\begin{Bmatrix} -5.2 \\ -1.3 \\ 3.9 \\ -2.6 \\ 1.3 \\ 1.2 \end{Bmatrix} = \begin{Bmatrix} -0.7143 \\ -0.1786 \\ 0.5357 \\ -0.3571 \\ 0.1786 \\ 0.1648 \end{Bmatrix} = \begin{Bmatrix} f_{ix} \\ f_{iy} \\ f_{jx} \\ f_{jy} \\ f_{kx} \\ f_{ky} \end{Bmatrix}$$

which consists of six components (forces in two directions at three nodes).

4.5 Boundary conditions.

Boundary conditions impose physical forces and displacements on the elasticity problem. Without them, the problem is not well-posed, i.e., [K] is singular. In solving the finite element equations, boundary conditions are used to define specific constraints placed on the matrix equations to yield nodal displacements appropriate for the physical problem being analyzed. Boundary conditions allowed in this program include forces applied at nodes, stresses applied on boundaries, and displacements fixed at nodes. Nodal forces and boundary stresses are used to define the force vector on the right-hand side of the matrix equations; fixed displacements signify fewer actual unknowns and are used to modify the stiffness coefficients to yield these nodal displacements. Because nodal displacements are vector quantities, the boundary conditions have both magnitude and direction. You can specify surface stresses in directions normal or tangent to boundaries, but the program converts them to global coordinate direction components before applying them to the matrix equations.

We give two sets of units commonly used for boundary conditions in Table 4.3. You can use any consistent set of units.

Table 4.3. Units for Boundary Conditions.

Variable	*SI Units*	*English Units*
Force	Newtons	Pounds
Stress	Pascals	Pounds/inch^2
Displacement	Meters	Inches

4.5.1 Point forces.

A force acting at a node has two force components, f_x and f_y. Add these force components to the force matrix, {F}, in the rows corresponding to the proper node and directions (degree of freedom). Change the element force matrix for a force applied at node i as follows:

$$\{F\} = \{F\} + \begin{Bmatrix} f_x \\ f_y \\ 0 \\ 0 \\ 0 \\ 0 \end{Bmatrix} \qquad 4.46$$

where f_x and f_y are x and y components of the force at node i.

In axisymmetric problems, the force matrix modification is:

$$\{F\} = \{F\} + \begin{Bmatrix} f_r \\ f_z \\ 0 \\ 0 \\ 0 \\ 0 \end{Bmatrix} \qquad 4.47$$

where f_r and f_z are r and z components of the force at node i.

4.5.2 Body forces.

Forces acting uniformly over the volume of an element (such as gravitational, inertial, or magnetic forces) are called body forces. You can treat body forces as point forces by distributing the total force among the nodes of an element. The body force modifies the element force matrix in a Cartesian coordinate system as follows:

$$\{F\} = \{F\} + (t\,A/3) \begin{Bmatrix} X \\ Y \\ X \\ Y \\ X \\ Y \end{Bmatrix} \qquad 4.48$$

where X and Y are x and y components of body force per unit volume.

Axisymmetric body forces are not distributed equally to all nodes of an element. Because portions of an element further from the z-axis have greater mass, they experience more of the body forces; thus, nodes with larger z-coordinates receive more of the total body force. The program distributes body forces to the element force matrix as follows[8] :

[8] Segerlind, 1984 p298,318

$$\{F\} = \{F\} + (2\pi\ A/12) \begin{Bmatrix} (2\ R_i+R_j+R_k)\ R \\ (2R_i+R_j+R_k)\ Z \\ (R_i+2R_j+R_k)\ R \\ (R_i+2R_j+R_k)\ Z \\ (R_i+R_j+2R_k)\ Z \end{Bmatrix} \qquad 4.49$$

where R and Z are r and z components of body force per unit volume and R_i, R_j, and R_k are the r-coordinates of nodes i, j, and k, respectively.

4.5.3 Surface stresses.

Treat stresses acting on the boundaries of elements as point forces distributed to the nodes on the element sides affected. Modify in the following manner the element force matrix for a stress applied to the element side bounded by nodes i and j in a Cartesian coordinate system:

$$\{F\} = \{F\} + [\ t\ (L_{i,j})/2] \begin{Bmatrix} s_x \\ s_y \\ s_x \\ s_y \\ 0 \\ 0 \end{Bmatrix} \qquad 4.50$$

where s_x and s_y are x and y components of the surface stress (force per unit area) on side i-j, Li,j is the side length, and t is the element thickness.

For axisymmetric problems, the distribution of the applied stress to boundary nodes depends on the radial coordinate of each node where the stress acts. Because the element thickness is proportional to the r-coordinate and surface area depends on thickness, the allocation of force to the nodes on side i-j of an element is as follows:

$$\{F\} = \{F\} + [\ (2\pi\ L_{i,j}\ )/6\ ] \begin{Bmatrix} (2R_i+R_j)\ s_r \\ (2R_i+R_j)\ s_z \\ (R_i+2R_j)\ s_r \\ (R_i+2R_j)\ s_z \\ 0 \\ 0 \end{Bmatrix} \qquad 4.51$$

where s_r and s_z are r and z components of the surface stress (force per unit area) on side i-j, $L_{i,j}$ is the side length, and R_i and R_j are the r-coordinates of nodes i and j, respectively.

4.5.4 Fixed displacements.

You can assign prescribed values to nodal displacements by altering the matrix equation to yield the desired "unknowns". Below we give a procedure for constraining the solution[9]. This method retains the original size of the matrix equation, i.e., no reduction in problem size is achieved. Perform these changes to the global matrix equation after you apply all other boundary conditions. This procedure fixes one displacement component at a time; you can use it repeatedly for multiple displacement constraints. The procedure modifies the equations to constrain the displacement in row p to the value Up.

1. First, set the force component to the value that will yield the desired displacement, Up, when only the diagonal term in this row is nonzero.
$$F_p = K_{p,p}\,U_p \qquad 4.52$$

2. Next, adjust all other stiffness coefficients in the pth row and the pth column so you can set to zero all non-diagonal coefficients in the pth row and column without invalidating the equations.
$$F_i = F_i - K_{i,p}\,U_p \quad \text{for all } i \neq p \qquad 4.53$$

3. Finally, set the non-diagonal stiffness coefficients in the pth row and pth column equal to zero.
$$K_{p,i} = K_{i,p} = 0 \quad \text{for all } i \neq p \qquad 4.54$$

4.6 Equation solution.

The global set of equations modified by the boundary conditions has a number of rows equal to twice the number of nodes. Solution for the unknowns (nodal displacements) is accomplished by Gaussian elimination with back substitution[10]. However, because the stiffness matrix is stored in condensed form, the solution algorithm includes extra manipulation of subscripts from that normally required in the Gaussian elimination and back substitution methods.

Table 4.4 Degrees of freedom for nodal displacements

DOF	*Node*	*Component*
1	1	x
2	1	y
3	2	x
4	2	y
5	3	x
5	3	y
...	...	...
2i-1	i	x
2i	i	y

Because the number of columns in the condensed stiffness matrix is equal to the bandwidth of the original global stiffness matrix (and is less than the number of unknowns), the condensed stiffness matrix is not square. The global solution vector has displacement components arranged according to their degrees of

[9] Segerlind, 1984, p417ff

[10] Conte, 1965, p156

freedom as shown in Table 4.4. Odd numbers correspond to x-displacements; even numbered degrees of freedom correspond to y-components.

4.7 Postprocessing.

Once you determine the nodal displacements, you can obtain auxiliary values. Element strains in each direction are constant within an element and are defined by equation 4.15 (for Cartesian coordinate systems) or by equation 4.28 (for axisymmetric systems). Element stresses also are constant throughout an element and are defined by equation 4.17 (for Cartesian coordinate systems) or by equation 4.30 (for axisymmetric systems). Using Mohr's circle relationships you can obtain the principal stresses in an element. Generally, you assume element strains and stresses act at the element centroid. Use a weighted interpolation procedure to estimate stress values at the nodes[11].

The program uses the following to determine resultant forces acting at the nodes by the product of the global stiffness matrix (before application of boundary conditions) and the global nodal displacement vector:

$$\{F\} = [K]\{U\} \tag{4.55}$$

Example 4.7 - Strains and stresses in an element.

Consider the element shown in Example 4.4 with node displacements of:

Node	*x-component*	*y-component*
i	.0025	.0020
j	.0010	.0015
k	.0020	.0010

From equation 4.15, the element strains are:

$$\{\varepsilon^{(e)}\} = [B^{(e)}]\{U^{(e)}\} = (1/11)\begin{bmatrix} -4 & 0 & 3 & 0 & 1 & 0 \\ 0 & -1 & 0 & -2 & 0 & 3 \\ -1 & -4 & -2 & -2 & 3 & 1 \end{bmatrix}\begin{Bmatrix} .0025 \\ .0020 \\ .0010 \\ .0015 \\ .0020 \\ .0010 \end{Bmatrix}$$

which yields

$$\{\varepsilon^{(e)}\} = (1/110)\begin{Bmatrix} -0.0050 \\ -0.0020 \\ -0.0010 \end{Bmatrix} = \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{xy} \end{Bmatrix}$$

and the element stresses (assuming plane stress) are given by equation 4.17 with equation 4.18:

[11] Segerlind, 1976, p100ff

$$\{\sigma^{(e)}\} = [D]\{\varepsilon^{(e)}\} = (54{,}945)\,(1/11)\begin{bmatrix} 1 & .3 & 0 \\ .3 & 1 & 0 \\ 0 & 0 & .35 \end{bmatrix}\begin{Bmatrix} -.005 \\ -.002 \\ -.001 \end{Bmatrix}$$

yielding

$$\{\sigma^{(e)}\} = 4995\begin{Bmatrix} -0.00560 \\ -0.00350 \\ -0.00035 \end{Bmatrix} = \begin{Bmatrix} -27.972 \\ -17.483 \\ -1.7483 \end{Bmatrix} = \begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \rho_{xy} \end{Bmatrix}$$

You can use any consistent set of units for these properties. Sample sets of units for material properties are presented in Table 4.1.

COOKE PUBLICATIONS TITLES (May 1989)

1. MathWriter™ 1.4 & MW2TeX™

MathWriter 1.4 (the mathematics editor) is an interactive, WYSIWYG mathematical editor for the Macintosh (which is 30% larger than MacWrite™and can be used with all standard Macintosh word processors). Due to its elegant design it is easy to use by the novice and casual user yet meets the power requirements of the professional. MW2TeX™ automatically translates the MathWriter data structures into the TeX page description language created by Donald Knuth.

2. MathWriter™ 2.0 —The Elegant Scientific Word Processor

A complete word processor for the professional mathematician and scientist. Full "WYSIWYG" support for text AND fully integrated mathematics. No longer must you paste mathematical expressions into a word processor as PICTs. Features include dual control by mouse and keyboard, a visual orientation, automatic equation numbering, automatic revision tracking, hypertext memo notes, and a powerful macro capability. (Available in 1989.)

3. StomateTutor™ 2.0

StomateTutor is a HyperCard presentation of the mechanism used by plants to regulate their gaseous environment. This Macintosh program presents recent original biological engineering research in a manner comprehensible by both upper and lower division biology students and describes implications for crop production. (Requires HyperCard)

4. MacRegistrar™1.03 and PC-Registrar™ 2.0

MacRegistrar handles course management details such as grading and attendance records for a class of any size up to 2,600. With MacRegistrar, class lists can be issued to the faculty on disks or by network; grades and new students can be added manually in random search or alphabetical order or directly from computer created files, e.g., machine scored tests; then final grades or other summaries can be computed and returned to the Registrar by disk or by network. MacRegistrar is the definitive treatment for this specialized database. An IBM-PC version, **PC-Registrar™ 2.0,** is in preparation.

5. MacElastic™ 1.0, MacPoisson™1.0, PC-Elastic™1.5, and PC-Poisson™ 1.5

These instructional finite element analysis programs fully exploit the graphics environment for both input and output. With these programs this powerful numerical technique for solving the equations of elasticity, electrostatics, ideal fluid flow, steady state heat conduction, etc. can be introduced in the undergraduate curriculum.

6. DiskManagerPC™ 1.1

This hard disk security and access control software provides password access to the IBM-PC hard disk, read only and execute only access to files on the hard disk, and optional purging of temporary student files in an instructional facility.

7. QuikBase™ 1.0

QuikBase is a general purpose database for the IBM PC which emphasizes speed and simplicity using point and shoot style. Variable length fields make this ideal for bibliographic retrieval. In preparation.