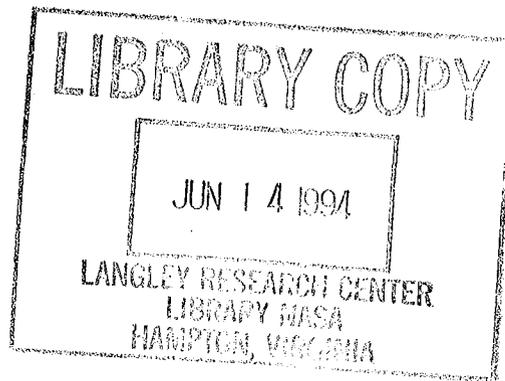
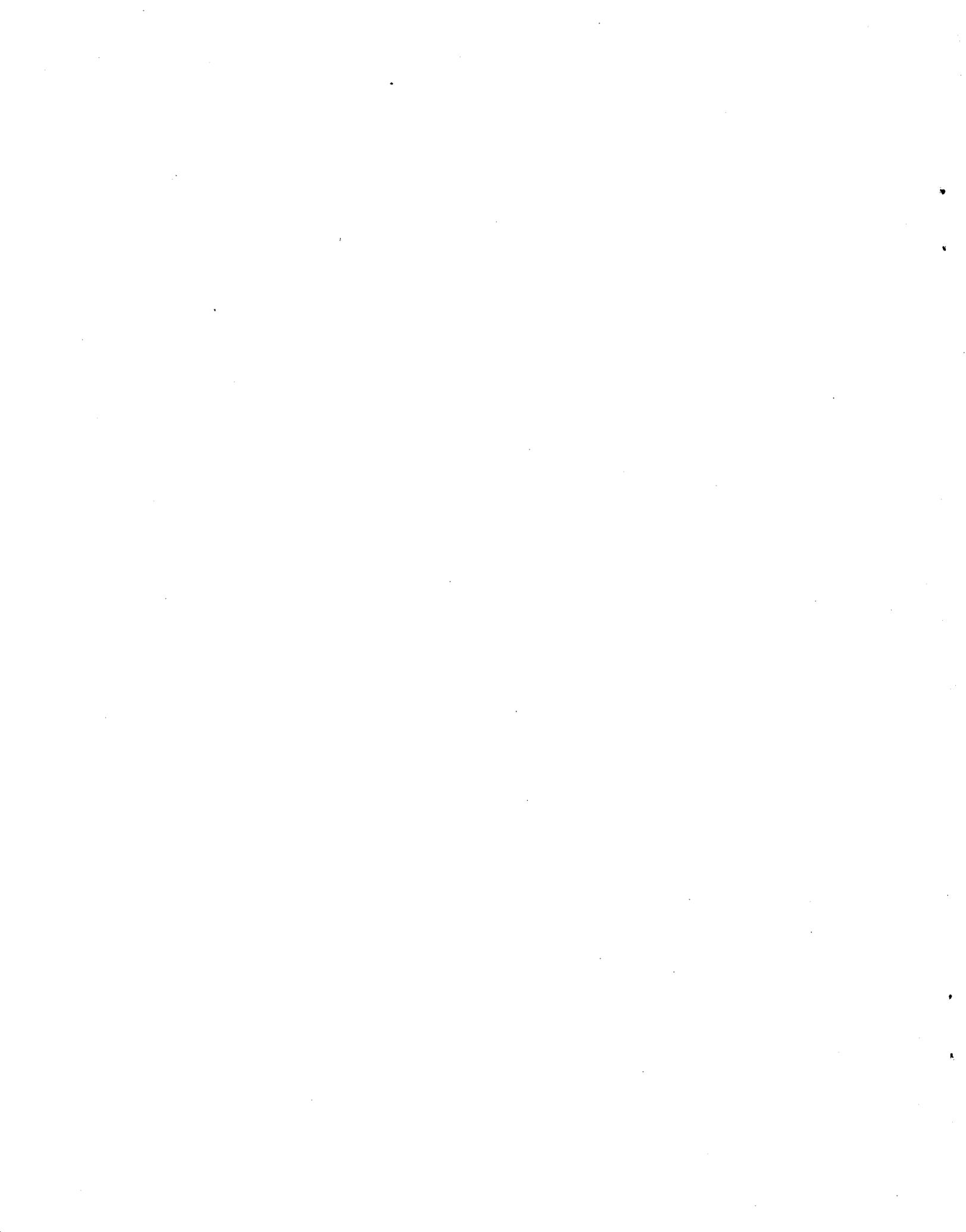


# **BUCKY Instruction Manual Version 3.3**

**James P. Smith**

MAY 1994







3 1176 01408 7317

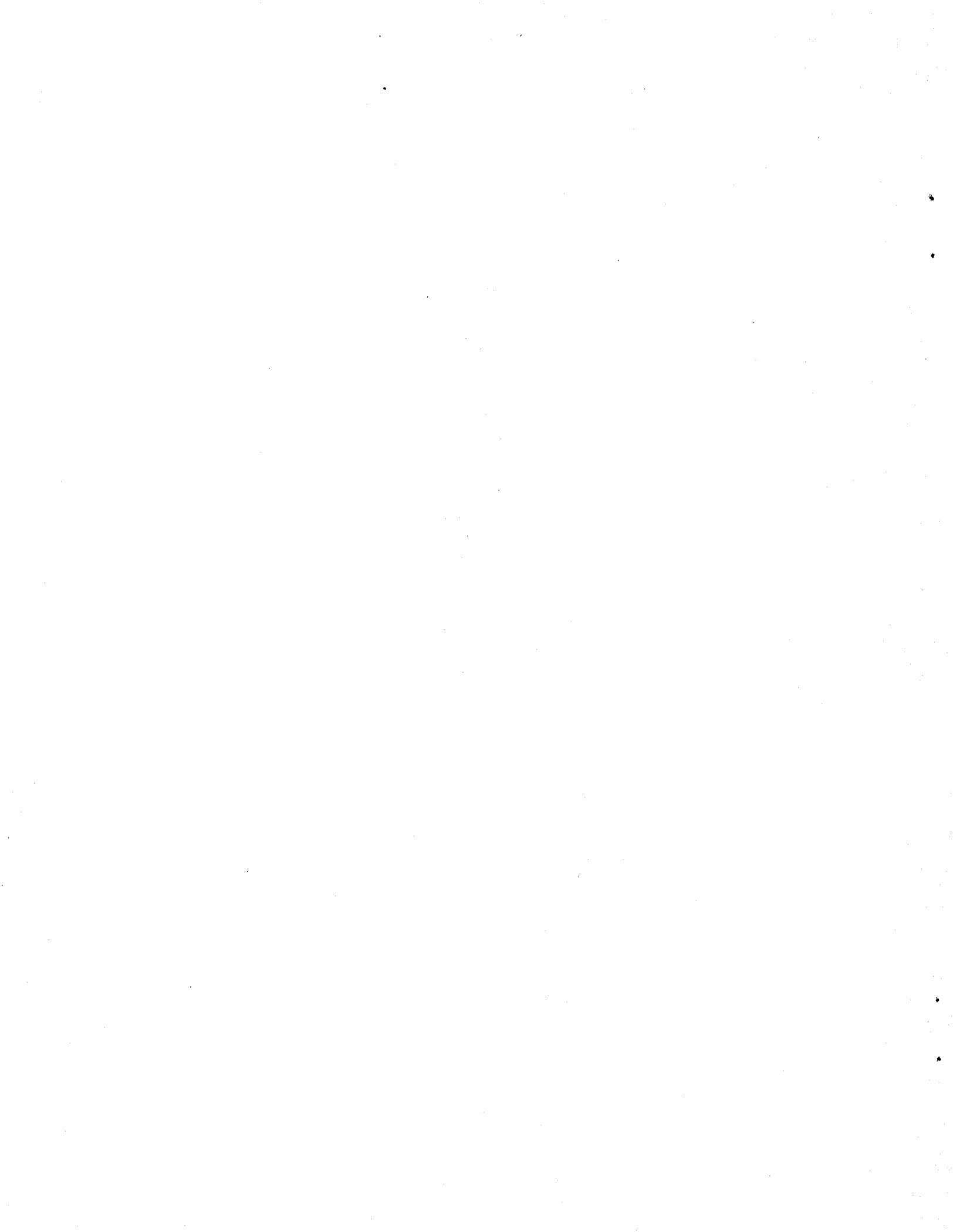
**NASA Technical Memorandum 104793**

# **BUCKY Instruction Manual Version 3.3**

**James P. Smith**  
*Lyndon B. Johnson Space Center*  
*Houston, Texas*



**National Aeronautics and  
Space Administration**



# Contents

<b>1</b>	<b>Introduction</b>	<b>1</b>
<b>2</b>	<b>BUCKY Program History</b>	<b>2</b>
<b>3</b>	<b>Installing BUCKY in a Computer Environment</b>	<b>3</b>
<b>4</b>	<b>Running BUCKY</b>	<b>4</b>
<b>5</b>	<b>BUCKY Problem Size Limitations</b>	<b>5</b>
<b>6</b>	<b>BUCKY Input</b>	<b>6</b>
6.1	Element Geometry . . . . .	6
6.2	Plane Stress Loading Conventions . . . . .	7
6.3	Plate Bending Loading Conditions . . . . .	7
<b>7</b>	<b>Interpreting the Output</b>	<b>8</b>
7.1	Output Data File - ftn08 . . . . .	8
7.2	BUCKY Logfile - ftn09 . . . . .	8
7.3	I-DEAS Files Written by BUCKY . . . . .	8
7.3.1	I-DEAS Universal File Model File - <i>IDEAS.UNV</i> . . . . .	8
7.3.2	I-DEAS Plane Stress Displacements File - <i>IDEAS.DIS</i> . . . . .	9
7.3.3	I-DEAS Plane Stress Stress File - <i>IDEAS.STR</i> . . . . .	9
7.3.4	I-DEAS Bending Analysis Displacement File - <i>IDEAS.BEN</i> . . . . .	9
7.3.5	I-DEAS Bending Analysis Stress File - <i>IDEAS.BST</i> . . . . .	10
7.3.6	I-DEAS Buckled Mode Shape File - <i>IDEAS.MOD</i> . . . . .	10
7.3.7	I-DEAS Plasticity Displacements File - <i>IDEAS.PLD</i> . . . . .	11
7.3.8	I-DEAS Plasticity Stress File - <i>IDEAS.PLS</i> . . . . .	11
7.3.9	I-DEAS Plasticity Strain File - <i>IDEAS.PLE</i> . . . . .	11
7.4	PATRAN Files Written by BUCKY . . . . .	12
7.4.1	PATRAN Neutral File - <i>PATRAN.PAT</i> . . . . .	12
7.4.2	PATRAN Plane Stress Displacements File - <i>PATRAN.DIS</i> . . . . .	12
7.4.3	PATRAN Plane Stress Stress File - <i>PATRAN.STR</i> . . . . .	12
7.4.4	PATRAN Bending Analysis Displacement File - <i>PATRAN.BEN</i> . . . . .	13
7.4.5	PATRAN Bending Analysis Stress File - <i>PATRAN.BST</i> . . . . .	14
7.4.6	PATRAN Buckled Mode Shape File - <i>PATRAN.MOD</i> . . . . .	14
7.4.7	PATRAN Plasticity Displacements File - <i>PATRAN.PLD</i> . . . . .	14
7.4.8	PATRAN Plasticity Stress File - <i>PATRAN.PLS</i> . . . . .	15
7.4.9	PATRAN Plasticity Strain File - <i>PATRAN.PLE</i> . . . . .	15
<b>8</b>	<b>BUCKY Data Cards</b>	<b>16</b>
<b>9</b>	<b>Examples</b>	<b>28</b>
9.1	Square Plate . . . . .	28
9.2	Square Plate with Cutout, Quarter Model . . . . .	30
9.3	Square Plate Under Hydrostatic Pressure Load . . . . .	33
9.4	Plastic Analysis of a Uniformly Loaded Cantilever Beam . . . . .	36
9.5	Axisymmetric Analysis of a Spherical Pressure Vessel . . . . .	38
<b>10</b>	<b>Error Messages</b>	<b>41</b>

<b>11 BUCKY Theoretical Supplement</b>	<b>46</b>
11.1 Introduction	46
11.2 Problem Statement	47
11.3 Minimum Potential Energy	47
11.4 Subparametric Mappings	48
11.5 Plate Buckling	49
11.6 Plate Bending	50
11.7 Plane Stress	50
11.8 Three-Dimensional Axisymmetric Analysis	51
11.9 Plane Stress Implementation	52
11.10 Plate Buckling Implementation	52
11.11 Plate Bending Implementation	53
11.12 Plastic Analysis	53
11.13 Axisymmetric Analysis Implementation	55
11.14 Conclusions	55

## List of Figures

1	Standard eight-noded BUCKY element. . . . .	6
2	Sign convention of applied edge loads. . . . .	6
3	Sign convention for applied loads and moments. . . . .	7
4	Four-element model of a square plate. . . . .	28
5	Quarter model of a square plate with an internal circular cutout. . . . .	31
6	von Mises stress distribution in a square plate with cutout. . . . .	32
7	Two element beam model for elastic-plastic analysis . . . . .	36
8	Axisymmetric sphere model. . . . .	38
9	Radial stress distribution in a spherical shell. . . . .	40
10	Circumferential stress distribution in a spherical shell. . . . .	40
11	Structure broken into a four-element model. . . . .	47
12	Subparametrically mapped element from physical to parametric space. . . . .	48
13	von Mises yield criterion. . . . .	54



# 1 Introduction

BUCKY is a program written to analyze structures with a high degree of accuracy. Specifically, BUCKY can analyze plates for their buckling behavior, as well as their behavior in plane stress and plate bending. In addition, BUCKY provides elastic-plastic solutions of isotropic plates in a state of plane stress. Three-dimensional problems can be treated using BUCKY's axisymmetric solution sequence. Using a higher order displacement function, known as the  $p$ -finite element method, BUCKY can be used to determine a plate's response to a variety of loading and boundary conditions. The very nature of  $p$ -finite elements allows for several traditional  $h$ -elements to be replaced by a single  $p$ -element. Additionally, the high order of the displacement functions enables the user to produce results that are more accurate than traditional  $h$ -finite elements.

In the current version of BUCKY (version 3.3), the user can choose between plane stress, plate bending (Kirchoff or Reissner-Mindlin theory), plate buckling, two-dimensional plane stress plasticity, and three-dimensional axisymmetric analysis. The current version assumes that the plate can vary its thickness linearly in the spatial coordinates. Additionally, the user can prescribe pressure loads, distributed edge moments, or edge tractions that are constant, linear, or quadratic in behavior. For plate bending, loads can also be applied in the form of concentrated point loads. BUCKY also supports orthotropic as well as isotropic material properties on an element level.

The BUCKY manual is subdivided into several main sections, including

- Program history
- Installing BUCKY
- Running the program
- Interpreting the output
- Examples
- Error messages
- Theoretical background

## 2 BUCKY Program History

BUCKY is currently up to version 3.1. Major upgrades are denoted by a higher first digit; minor upgrades are represented by the number after the decimal point in the version number. The brief program history is

- Version 1.0 - First BUCKY program. Fully functional two-dimensional plane stress and plate buckling analysis.  
August 1992
  
- Version 2.0 - Plate bending solution added. Linearly varying element thickness, quadratically varying loads (edges tractions and pressure forces) added.  
January 1993
  
- Version 2.1 - BORDER card added to make bending and buckling analysis have a different  $p$ -order from the plane stress case. Analysis is done out of core (disk based), so large problems may be done.  
July 1993
  
- Version 2.2 - Dependence upon disk-based solutions relieved somewhat. Data are stored on disk, but heavy calculations are done in core for faster execution.  
July 1993
  
- Version 2.3 - Much faster execution by optimizing constraint reductions. Performance increases of upwards of 1000% over the previous version.  
July 1993
  
- Version 3.0 - Major update. Distributed edge moments which may vary quadratically along the element edge were added. These moments come in the form of bending and twisting moments. The constraint reduction and application were modified to gain even more speed. The Cholesky decomposition was improved to decrease the required solution time. Plane stress plasticity solution has been added. This solution is performed with the tangent stiffness method.  
August 1993
  
- Version 3.1 - I-DEAS Universal File format added to output options to give further flexibility of graphical interpretation.  
October 1993
  
- Version 3.2 - Three-dimensional axisymmetric analysis added to library of solution sequences. I-DEAS to BUCKY translator added for pre-processing (node and element generation). Script file added to run BUCKY so that input file does not need to be named ftn07.  
November 1993
  
- Version 3.3 - Program converted from single to double precision in all respects. A new element "type" was added - BUCKY now supports the use of shear deformable plates based on Reissner-Mindlin plate theory.  
January 1994

## 3 Installing BUCKY in a Computer Environment

### Included Distribution Files

---

bucky\_v3.3.tar - UNIX tar format file

BUCKY\_v3.3/src - FORTRAN source files  
BUCKY\_v3.3/examples - BUCKY sample input files  
BUCKY\_v3.3/Makefile.unix - standard UNIX f77 compiler Makefile  
BUCKY\_v3.3/Makefile.unicos - Cray UNICOS cft77 compiler Makefile  
BUCKY\_v3.3/README - Information file  
BUCKY\_v3.3/ideas\_to\_bucky.f - I-DEAS to BUCKY translator  
BUCKY\_v3.3/bucky - UNIX script file to run BUCKY  
BUCKY\_v3.3/BUCKY\_v3.3.Manual.ps - BUCKY manual in postscript format

### Installation

---

#### Cray UNICOS Platforms

- 1) Copy bucky\_v3.3.tar to root directory
- 2) type 'tar xvf bucky\_v3.3.tar'
- 3) type 'cd BUCKY\_v3.3'
- 4) type 'cp Makefile.unicos Makefile'
- 5) type 'make'

#### Standard UNIX system

- 1) Copy bucky\_v3.3.tar to root directory
- 2) type 'tar xvf bucky\_v3.3.tar'
- 3) type 'cd BUCKY\_v3.3'
- 4) type 'cp Makefile.unix Makefile'
- 5) type 'make'

### Problems

---

Problems with installation or run-time errors should be directed to

James P. Smith  
NASA/Johnson Space Center  
Mail Code ES2  
Houston, TX 77058  
(713)-483-1242  
smithj@smd4.jsc.nasa.gov

## 4 Running BUCKY

There are two methods for running BUCKY. In the first, BUCKY can be called directly, or it can be run using a UNIX script file. In the first case, an input data file must be prepared to run BUCKY. The data file will be read corresponding to file input/output device 7. Thus, prior to running BUCKY, the file to be executed should be copied to the file corresponding to unit 7. In the current version of BUCKY, it is assumed that the file associated with the file device 7 is named ftn07, so to copy an input file to the proper location, type

```
cp file ftn07
```

Then, to run BUCKY, type

```
BUCKY
```

at the system prompt. The formats for the data cards in the input file are given in Section 8. As BUCKY runs, a log file is generated. Should an error occur during the execution of BUCKY, the log file will contain the reason for the error. Upon successful completion of BUCKY, an additional output file is generated with the data results from the problem solution. PATRAN and I-DEAS-readable output files are also prepared at this time.

In the second case, to run BUCKY we can use the supplied UNIX script named bucky. The script file will prompt the user for the input and output filenames. The input filename should be an existing file. The output file will be created upon completion. The script file will then execute BUCKY automatically, and, upon completion, the output file and a file named bucky.log are created.

## 5 BUCKY Problem Size Limitations

In the current version of BUCKY, problems are solved with a dimensioned array. The size of this array is given by a parameter `MATSIZ` in the main program. The current value for `MATSIZ` is 3000. An approximate formula for the required number of bytes for BUCKY is

$$mem = 8[9 * MATSIZ + (MATSIZ)^2].$$

There should be *mem* bytes of storage available in real or virtual memory for BUCKY to work properly.

The size of the problem solved with BUCKY is currently dictated by the value of `MATSIZ`. This parameter gives the largest unconstrained problem size. The application of constraints yields a smaller problem size, but, at this point, the code is not enhanced with advanced solving features. The total number of degrees of freedom (NDOF) associated with a problem is given by the expression

$$ndof_{total} = ANALTYPE * NUMELE * \left( \frac{p^2 + 3p + 6}{2} \right),$$

where `ANALTYPE` takes a value of 1 (Kirchoff plate bending), 2 (two-dimensional plane stress, plasticity, or three-dimensional axisymmetric problems), or 3 (Reissner-Mindlin plate bending); `NUMELE` is the number of elements in the problem; and *p* is the polynomial order of the solution. A proper combination of `ANALTYPE`, `NUMELE`, and *p* must be chosen to make sure that *ndof<sub>total</sub>* is equal to or below the parameter `MATSIZ`. Valid values for *p* are *p* = 4 to *p* = 8.

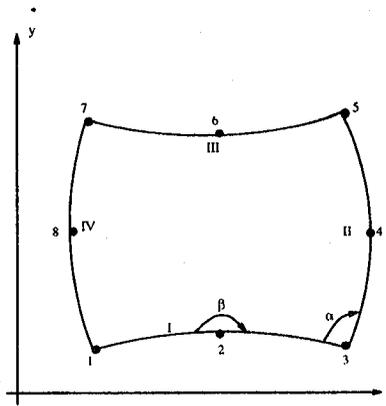


Figure 1: Standard eight-noded BUCKY element.

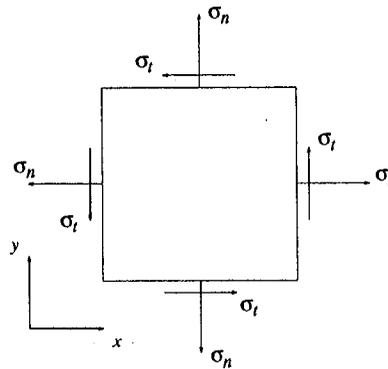


Figure 2: Sign convention of applied edge loads.

## 6 BUCKY Input

The following sections describe how to put a BUCKY model together for analysis.

### 6.1 Element Geometry

The geometry of the elements used in BUCKY are eight-noded subparametric elements. The nodes that are the basis of the data file give the geometry of the model. For each element, the element consists of eight local nodes, listed from 1 to 8, and four element sides, listed as I to IV. The element geometry is shown in Figure 1.

As a general rule, an edge should not have too much curvature, or numerical errors may arise. The following guidelines are good rules of thumb:

- Do not let the angle  $\beta$  for any edge be less than  $135^\circ$  or greater than  $225^\circ$ .
- Do not let the angle  $\alpha$  between two edges be greater than  $135^\circ$  or less than  $45^\circ$ .
- Side nodes should be near the center of the side; if the length of the side is  $L$ , then the side node should not be a distance  $L/4$  or less from either corner node on that side.
- Aspect ratios of the element should be less than 5 to 1 and absolutely no greater than 10 to 1; the aspect ratio is the greater of the width to height or height to width; a large aspect ratio can seriously degrade the quality of the computed results.

BUCKY models can also be prepared using the commercial program I-DEAS. In this case, I-DEAS can be used to prepare the model geometry, the element material properties, and the element thicknesses. The translator program supports the linearly varying thickness capabilities of BUCKY. The boundary conditions and loading parameters

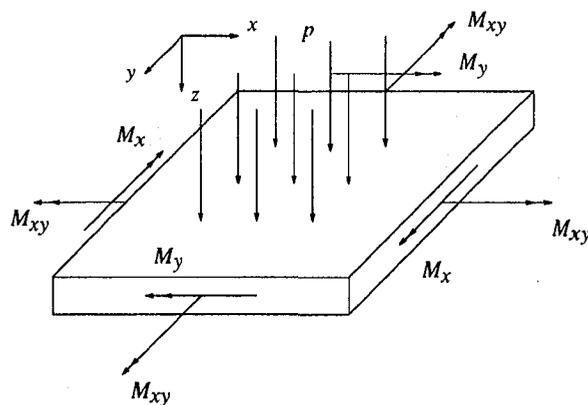


Figure 3: Sign convention for applied loads and moments.

must still be written in to the BUCKY data file. The I-DEAS to BUCKY translator is named `ideas_to_bucky`, and this program prompts the user for the I-DEAS Universal file. Upon completion, a BUCKY input file is generated.

## 6.2 Plane Stress Loading Conventions

The loading conventions are shown in Figure 2 for the plane stress analysis. As the normal and tangent vectors move around the element perimeter, the normal ( $\mathbf{n}$ ) is always directed away from the element edge. The tangent vector,  $\mathbf{t}$ , is always oriented  $90^\circ$  counterclockwise from the normal vector. For tensile stresses, if the stress is tensile, then  $\sigma_n$  is positive. If the load is compressive, then it acts in the direction opposite the normal, and its value should be less than one. By the same token, a positive shear load on any edge is in the direction of the tangent vector. Thus, to enter a shear load opposite the tangent vector, a negative value should be entered.

## 6.3 Plate Bending Loading Conditions

The loading conventions for the case of plate bending are shown in Figure 3. A positive value of the pressure and point load acts in the  $+z$  direction in the frame of reference of the plate. Also, this convention means that a pressure or point load is positive if it causes tension on the top side ( $z = +h/2$ ) of the plate. The bending moments  $M_x$  and  $M_y$  are positive if they produce tension in the upper surface of the plate. The twisting element  $M_{xy}$  is positive if its vector equivalent acts normally to the plate side.

## 7 Interpreting the Output

This section gives an explanation of the output files generated by BUCKY. There are two primary output files, plus the PATRAN and I-DEAS post-processing files. To access the data contained in either the PATRAN or I-DEAS related files, the PATRAN or IDEAS card must be supplied in the BUCKY data deck.

### 7.1 Output Data File - ftn08

This file contains the nodal and element information used during the execution of BUCKY. Also, depending on the type of analysis chosen, displacements, strains, and stresses from the analysis are given for the eight nodes plus the element center for each element. The displacements are given in the global  $(x, y)$  system for Cartesian two-dimensional analysis or in the global  $(r, z)$  system for axisymmetric cylindrical coordinate problems, and the stresses and strains correspond to the global  $(x, y)$  or  $(r, z)$  systems as well. If the BUCKLE option has been included in the data file (ftn07), then the last number in the file is the buckling factor. This number determines critical buckling load. If this number is equal to one, then the loads (stresses) input in the data file will cause the plate to buckle. If the buckling factor is below one, then this number times the input loads will produce buckling (the plate has already buckled). Finally, if the buckling factor is above one, then the plate has not buckled, and to buckle the plate requires that the loads be multiplied by the buckling factor. A negative number indicates a change in direction of the loads. For example, if the plate has an applied tensile rather than a compressive load, the buckling factor would be negative and the load would have to be in the opposite direction (making it compressive) to induce buckling.

If the BENDING option is included in the data deck, then a bending analysis of the plate subject to lateral pressure and distributed moment loads is performed. The output file contains information on the deformed configuration of the plate due to transverse pressure loads, point loads, and distributed edge moments. The out-of-plane displacement, normal stresses, and the shear stress are reported to the user at eight nodes in an element, plus the element center. The stresses computed at the nodes and center correspond to the maximum stresses. The strains vary linearly through the plate thickness, and the strain in the midplane due to bending is identically zero. The stresses at the outer fibers are equal and opposite in magnitude. The strains at the top fibers (at  $z = h/2$ ) are given in the output file.

Should a plastic analysis of the plate be requested, this file contains the displacements, stresses, and strains as the plate deforms plastically. In this case, either the total deformation is recorded or the user can specify that the results file be updated after each iteration or after a certain number of iterations (using the UPDATE card).

### 7.2 BUCKY Logfile - ftn09

This file simply contains an echo of the input file, any errors that may cease program operation, and messages about the execution of BUCKY.

### 7.3 I-DEAS Files Written by BUCKY

BUCKY can write the problem geometry to an I-DEAS Universal file for analysis. In addition, the displacement, stress, and strain data are also written to a universal file by specifying I-DEAS output in the BUCKY data deck.

#### 7.3.1 I-DEAS Universal File Model File - IDEAS.UNV

This file contains the model file for the plate. The plate element is subdivided into smaller elements for graphical analysis. The number of element subdivisions is given by the BUCKY card SUBDIV. There are SUBDIV\*SUBDIV elements created for each BUCKY element. The model is read into I-DEAS by issuing the following commands after entering I-DEAS and establishing a model file:

FE	(FE_Modeling_&_Analysis)
PO	(Post_Processing)
MA	(Manage_Models)
REA	(Read)
U	(Universal)
IDEAS.UNV	(Select defaults when prompted)

DR	(Draw)
AU	(Autoscale)

I-DEAS reads the problem in and draws it on the screen.

### 7.3.2 I-DEAS Plane Stress Displacements File - *IDEAS.DIS*

To view the deformed shape of the plate due to the plane stress analysis, the file *IDEAS.DIS* is used. To read this file, type

MAN	(Manage_Models)
REA	(Read)
U	(Universal)
IDEAS.DIS	
A	(Analysis_Dataset_Selection)
CU	(Current)
L	(Loadset_&_Type)
DISP	(Displacements)
DE	(Deformed_Geometry)
LIN	(Line)
EX	(Execute)

### 7.3.3 I-DEAS Plane Stress Stress File - *IDEAS.STR*

There are seven different quantities of stress that can be plotted in two-dimensional plane stress analysis:  $\sigma_x$ ,  $\sigma_y$ ,  $\tau_{xy}$ ,  $\sigma_1$ ,  $\sigma_2$ ,  $\sigma_E$ , and  $\tau_{max}$ , where  $\sigma_1$  and  $\sigma_2$  are the principal stresses,  $\sigma_E$  is the Von Mises equivalent stress, and  $\tau_{max}$  is the maximum shear stress. Similarly, if axisymmetric analysis is chosen, the following quantities can be plotted:  $\sigma_r$ ,  $\sigma_z$ ,  $\tau_{rz}$ ,  $\sigma_\theta$ ,  $\sigma_1$ ,  $\sigma_2$ , and  $\sigma_E$ . To read these quantities requires knowledge of the I-DEAS menu selection Data\_Component. To plot the results, type the following commands in I-DEAS:

MAN	(Manage_Models)
REA	(Read)
U	(Universal)
IDEAS.STR	
A	(Analysis_Dataset_Selection)
CU	(Current)
L	(Loadset_&_Type)
STRE	(Stress)
CO	(Contour)
DA	(Data_Component)
desired stress	
CONTI	(Continuous_Tone)
EX	(Execute)

### 7.3.4 I-DEAS Bending Analysis Displacement File - *IDEAS.BEN*

When a bending analysis is desired (specified by the BENDING card), BUCKY determines the deformed configuration as well as the stresses in the plate. The file *IDEAS.BEN* gives the deformed shape of the plate subjected to normal pressure loading. To view the deformed shape, type the following commands in I-DEAS:

MAN	(Manage_Models)
REA	(Read)
U	(Universal)

IDEAS.BEN	
A	(Analysis.Dataset.Selection)
CU	(Current)
L	(Loadset.&.Type)
DISP	(Displacements)
DE	(Deformed.Geometry)
LIN	(Line)
EX	(Execute)

The deformed shape can also be displayed using a contour plot. Since only one component of displacement is used in the bending analysis, we can type

CO	(Contour)
D	(Data.Component)
Z	(Z.Component)
CONTI	(Continuous.Tone)
E	(Execute)

### 7.3.5 I-DEAS Bending Analysis Stress File - IDEAS.BST

There are seven different quantities of stress that can be plotted:  $\sigma_x$ ,  $\sigma_y$ ,  $\tau_{xy}$ ,  $\sigma_1$ ,  $\sigma_2$ ,  $\sigma_E$ , and  $\tau_{max}$ . To plot the results, type the following commands:

MAN	(Manage.Models)
REA	(Read)
U	(Universal)
IDEAS.BST	
A	(Analysis.Dataset.Selection)
CU	(Current)
L	(Loadset.&.Type)
STRE	(Stress)
CO	(Contour)
DA	(Data.Component)
desired stress	
CO	(Continuous.Tone)
EX	(Execute)

### 7.3.6 I-DEAS Buckled Mode Shape File - IDEAS.MOD

Finally, the last file contains the mode shape corresponding to the critical buckling load of the plate. This file can be manipulated in one of two ways. First, if a deformed plot is desired, the user can search for the best perspective by changing the view from I-DEAS. Then, to read the data from *IDEAS.MOD*, follow the same procedure as with the previous displacement file, *IDEAS.DIS*, but with the new filename, *IDEAS.MOD*. Second, a contour plot may be easier to read. This requires the commands

MAN	(Manage.Models)
REA	(Read)
U	(Universal)
IDEAS.MOD	
A	(Analysis.Dataset.Selection)
CU	(Current)
L	(Loadset.&.Type)
DISP	(Displacements)

DE	(Deformed_Geometry)
LIN	(Line)
EX	(Execute)

### 7.3.7 I-DEAS Plasticity Displacements File - *IDEAS.PLD*

To view the deformed shape of the plate due to the plane stress analysis, the file *IDEAS.PLD* is used. To read this file, type

MAN	(Manage_Models)
REA	(Read)
U	(Universal)
IDEAS.PLD	
A	(Analysis_Dataset_Selection)
CU	(Current)
L	(Loadset_&_Type)
DISP	(Displacements)
DE	(Deformed_Geometry)
LIN	(Line)
EX	(Execute)

### 7.3.8 I-DEAS Plasticity Stress File - *IDEAS.PLS*

There are seven different quantities of stress that can be plotted:  $\sigma_x$ ,  $\sigma_y$ ,  $\tau_{xy}$ ,  $\sigma_1$ ,  $\sigma_2$ ,  $\sigma_E$ , and  $\tau_{max}$ , where  $\sigma_1$  and  $\sigma_2$  are the principal stresses,  $\sigma_E$  is the Von Mises equivalent stress, and  $\tau_{max}$  is the maximum shear stress. To plot the results, type the following commands:

MAN	(Manage_Models)
REA	(Read)
U	(Universal)
IDEAS.PLS	
A	(Analysis_Dataset_Selection)
CU	(Current)
L	(Loadset_&_Type)
STRE	(Stress)
CO	(Contour)
DA	(Data_Component)
desired stress	
CONTI	(Continuous_Tone)
EX	(Execute)

### 7.3.9 I-DEAS Plasticity Strain File - *IDEAS.PLE*

There are six different quantities of strain that can be plotted:  $\epsilon_x$ ,  $\epsilon_y$ ,  $\epsilon_{xy}$ ,  $\epsilon_1$ ,  $\epsilon_2$ , and  $\epsilon_{eq}$ , where  $\epsilon_1$  and  $\epsilon_2$  are the principal strains, and  $\epsilon_{eq}$  is the equivalent strain (similar to the Von Mises stress). To plot the results, type the following commands:

MAN	(Manage_Models)
REA	(Read)
U	(Universal)
IDEAS.PLE	
A	(Analysis_Dataset_Selection)
CU	(Current)

L	(Loadset.&.Type)
STRA	(Strain)
CO	(Contour)
DA	(Data.Component)
desired strain	
CONTI	(Continuous.Tone)
EX	(Execute)

## 7.4 PATRAN Files Written by BUCKY

BUCKY can write a PATRAN neutral file for the problem geometry as well as output files for the data, whether the data are displacements, strains, or stresses.

### 7.4.1 PATRAN Neutral File - *PATRAN.PAT*

This file contains the model file for the plate. The plate element is subdivided into smaller elements for graphical analysis. The number of element subdivisions is given by the BUCKY card SUBDIV. There are SUBDIV\*SUBDIV elements created for each BUCKY element. The model is read into PATRAN by issuing the following commands under the MODE menu:

5	(INTERFACE)
1	(NEUTRAL)
2	(INPUT MODEL)
PATRAN.PAT	
N	(OFFSET IDS)
Y	(PROCEED WITH FILE)

PATRAN then reads the problem in and draws it on the screen. To turn the element and node numbers off, type SET,LABELS,OFF, and then type PLOT to replot the model.

### 7.4.2 PATRAN Plane Stress Displacements File - *PATRAN.DIS*

To view the deformed shape of the plate due to the plane stress analysis, the file *PATRAN.DIS* is used. To read this file, type (under the MODE menu)

4	(RESULTS)
2	(EXTERNAL DATA)
1	(DEFORMED SHAPE)
PATRAN.DIS	
1	(UNDEFORM)
2	(DEFORM)

### 7.4.3 PATRAN Plane Stress Stress File - *PATRAN.STR*

There are seven different quantities of stress that can be plotted in two-dimensional plane stress problems:  $\sigma_x$ ,  $\sigma_y$ ,  $\tau_{xy}$ ,  $\sigma_1$ ,  $\sigma_2$ ,  $\sigma_E$ , and  $\tau_{max}$ , where  $\sigma_1$  and  $\sigma_2$  are the principal stresses,  $\sigma_E$  is the Von Mises equivalent stress, and  $\tau_{max}$  is the maximum shear stress. To read these quantities requires knowledge of which column of *PATRAN.STR* each is in. The columns associated with each stress are

- 1 -  $\sigma_x$  (Normal-x stress)
- 2 -  $\sigma_y$  (Normal-y stress)
- 3 -  $\tau_{xy}$  (Shear stress)
- 4 -  $\sigma_1$  (First principal stress)
- 5 -  $\sigma_2$  (Second principal stress)

- 6 -  $\sigma_E$  (Von Mises stress)
- 7 -  $\tau_{max}$  (Maximum shear stress)

In three-dimensional axisymmetric problems, the following stresses can be plotted:  $\sigma_r$ ,  $\sigma_z$ ,  $\sigma_\theta$ ,  $\sigma_{rz}$ ,  $\sigma_1$ ,  $\sigma_2$ , and  $\sigma_E$ . In the axisymmetric problem, the third principal stress is given by  $\sigma_\theta$ . The columns associated with each stress are

- 1 -  $\sigma_r$  (Radial stress)
- 2 -  $\sigma_z$  (Normal-z stress)
- 3 -  $\tau_{rz}$  (Shear stress)
- 4 -  $\sigma_\theta$  (Circumferential stress)
- 5 -  $\sigma_1$  (First principal stress in  $(r, z)$  plane)
- 6 -  $\sigma_2$  (Second principal stress in  $(r, z)$  plane)
- 7 -  $\sigma_E$  (Von Mises stress)

To plot the results, type the following commands under the MODE menu:

```

4          (RESULTS)
2          (EXTERNAL DATA)
4          (NODAL)
col        (column # of desired quantity)
1          (CONTOURS)
PATRAN.STR
1          (AUTO)
1 or 2     (CONTOUR) or (FRINGE)
7          (PLOT)
5          (DISPLAY LABELS)

```

#### 7.4.4 PATRAN Bending Analysis Displacement File - *PATRAN.BEN*

When a bending analysis is desired (specified by the BENDING card), BUCKY determines the deformed configuration as well as the stresses in the plate. The file *PATRAN.BEN* gives the deformed shape of the plate subjected to normal pressure loading. To view the deformed shape, type the following commands under the MODE menu:

```

4          (RESULTS)
2          (EXTERNAL DATA)
1          (DEFORMED SHAPE)
PATRAN.BEN
1          (UNDEFORM)
2          (DEFORM)

```

The deformed shape can also be displayed using a contour plot. Since only one component of displacement is used in the bending analysis, we can type (under the MODE menu)

```

4          (RESULTS)
2          (EXTERNAL DATA)
4          (NODAL)
3          (column number)
1          (CONTOURS)
PATRAN.BEN (or 1 for newfile followed by PATRAN.BEN)
1          (CONTOUR)
7          (PLOT)

```

#### 7.4.5 PATRAN Bending Analysis Stress File - *PATRAN.BST*

There are three different quantities of stress that can be plotted:  $\sigma_x$ ,  $\sigma_y$ , and  $\tau_{xy}$ . To read these quantities requires knowledge of which column of *PATRAN.BST* each is in. The columns associated with each stress are

- 1 -  $\sigma_x$  (Normal- $x$  stress)
- 2 -  $\sigma_y$  (Normal- $y$  stress)
- 3 -  $\tau_{xy}$  (Shear stress)

To plot the results, type the following commands under the MODE menu:

```

4                (RESULTS)
2                (EXTERNAL DATA)
4                (NODAL)
col              (column # of desired quantity)
1                (CONTOURS)
PATRAN.BST
1                (AUTO)
1 or 2          (CONTOUR) or (FRINGE)
7                (PLOT)
5                (DISPLAY LABELS)

```

#### 7.4.6 PATRAN Buckled Mode Shape File - *PATRAN.MOD*

Finally, the last file contains the mode shape corresponding to the critical buckling load of the plate. This file can be manipulated in one of two ways. First, if a deformed plot is desired, the user can search for the best perspective with the VIEW command within PATRAN. Then, to read the data from *PATRAN.MOD*, follow the same procedure as with the previous displacement file, *PATRAN.DIS*, but with the new filename, *PATRAN.MOD*. Second, a contour plot may be easier to read. This requires the commands (under the MODE menu)

```

4                (RESULTS)
2                (EXTERNAL DATA)
4                (NODAL)
3                (column number)
1                (CONTOURS)
PATRAN.MOD      (or 1 for newfile followed by PATRAN.MOD)
1                (CONTOUR)
7                (PLOT)

```

#### 7.4.7 PATRAN Plasticity Displacements File - *PATRAN.PLD*

To view the deformed shape of the plate due to the plane stress analysis, the file *PATRAN.PLD* is used. To read this file, type (under the MODE menu)

```

4                (RESULTS)
2                (EXTERNAL DATA)
1                (DEFORMED SHAPE)
PATRAN.PLD
1                (UNDEFORM)
2                (DEFORM)

```

#### 7.4.8 PATRAN Plasticity Stress File - PATRAN.PLS

There are seven different quantities of stress that can be plotted:  $\sigma_x$ ,  $\sigma_y$ ,  $\tau_{xy}$ ,  $\sigma_1$ ,  $\sigma_2$ ,  $\sigma_E$ , and  $\tau_{max}$ , where  $\sigma_1$  and  $\sigma_2$  are the principal stresses,  $\sigma_E$  is the Von Mises equivalent stress, and  $\tau_{max}$  is the maximum shear stress. To read these quantities requires knowledge of which column of *PATRAN.PLS* each is in. The columns associated with each stress are

- 1 -  $\sigma_x$  (Normal- $x$  stress)
- 2 -  $\sigma_y$  (Normal- $y$  stress)
- 3 -  $\tau_{xy}$  (Shear stress)
- 4 -  $\sigma_1$  (First principal stress)
- 5 -  $\sigma_2$  (Second principal stress)
- 6 -  $\sigma_E$  (Von Mises stress)
- 7 -  $\tau_{max}$  (Maximum shear stress)

To plot the results, type the following commands under the MODE menu:

```

4                (RESULTS)
2                (EXTERNAL DATA)
4                (NODAL)
col              (column # of desired quantity)
1                (CONTOURS)
PATRAN.PLS
1                (AUTO)
1 or 2          (CONTOUR) or (FRINGE)
7                (PLOT)
5                (DISPLAY LABELS)

```

#### 7.4.9 PATRAN Plasticity Strain File - PATRAN.PLE

There are six different quantities of strain that can be plotted:  $\epsilon_x$ ,  $\epsilon_y$ ,  $\epsilon_{xy}$ ,  $\epsilon_1$ ,  $\epsilon_2$ , and  $\epsilon_{eq}$ , where  $\epsilon_1$  and  $\epsilon_2$  are the principal strains, and  $\epsilon_{eq}$  is the equivalent strain (similar to the Von Mises stress). To read these quantities requires knowledge of which column of *PATRAN.PLE* each is in. The columns associated with each strains are

- 1 -  $\epsilon_x$  (Normal- $x$  strain)
- 2 -  $\epsilon_y$  (Normal- $y$  strain)
- 3 -  $\epsilon_{xy}$  (Shear strain)
- 4 -  $\epsilon_1$  (First principal strain)
- 5 -  $\epsilon_2$  (Second principal strain)
- 6 -  $\epsilon_{eq}$  (Equivalent strain)

To plot the results, type the following commands under the MODE menu:

```

4                (RESULTS)
2                (EXTERNAL DATA)
4                (NODAL)
col              (column # of desired quantity)
1                (CONTOURS)
PATRAN.PLS
1                (AUTO)
1 or 2          (CONTOUR) or (FRINGE)
7                (PLOT)
5                (DISPLAY LABELS)

```

## 8 BUCKY Data Cards

This section gives the data card formats for BUCKY. Each card is 80 columns wide and has 10 fields associated with it. Each field is 8 columns wide, and not all data cards utilize all 10 fields.

### # Card

The # card indicates that a comment is on the line. The inclusion of this card does not affect the execution of the problem; this card simply allows the user to input some information about the model within the data deck.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
# text									

The inclusion of the # card indicates that a comment follows on the line.

### AXIS3D Card

The AXIS3D card specifies that a three-dimensional axisymmetric analysis is requested. This type of solution, while modeling a three-dimensional problem, can be found using a two-dimensional approach.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
AIXS3D									

Plasticity solutions are currently not supported in the axisymmetric analysis.

### BEND Card

The BEND card specifies the bending boundary conditions for an edge on the element. Displacements and slopes are specified with the BEND card.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
BEND	EID	SID	IC1	IC2					
BEND	8	1	1	1					

EID           Element identification of the element the constraint is acting on (Integer>0)  
 SID           Side number the constraint is acting on for element EID (Integer>0)  
 IC1           Displacement boundary code. 1=no displacement, 0=free (Integer)  
 IC2           Slope boundary code. 1=no slope, 0=rotation (Integer)

SID must be a number between and including 1 and 4. The sides corresponding to SID are shown in Figure 1. If more than one side of element EID is constrained, a separate card must be used.

### BENDING Card

The BENDING card indicates that a bending analysis of the plate structure is desired. The plane stress and bending problems are decoupled from each other.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
BENDING									

The inclusion of the BENDING card indicates that a bending analysis of the plate is to be performed.

## BORDER Card

The BORDER card specifies the polynomial degree of the displacement functions for the plate bending and plate buckling solutions. A higher  $p$  value, which corresponds to the highest polynomial degree, gives a more precise result than a lower  $p$  value, but solution time is increased.

Field 1	Field2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
BORDER	$p$								
BORDER	8								

$p$  Polynomial degree (Integer>0)

The value of  $p$  is between 4 and 8. Any values of  $p$  below 4 will result in  $p = 4$ , and any  $p > 8$  will result in  $p = 8$ . If the BORDER card is not used to define the polynomial order for the plate bending or buckling problems, then the polynomial order defaults to the same as the plane stress solution.

## BUCKLE Card

The BUCKLE card indicates that the plate buckling solution is desired.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
BUCKLE									

The inclusion of the BUCKLE card indicates that a buckling solution of the plate is to be performed. If the BUCKLE card is not included in the data file, then only a plane stress solution is made.

## ELEMENT Card

The ELEMENT card specifies the geometry of the element. The geometry is determined by eight node points (specified by NODE cards).

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
ELEMENT	EID	NID1	NID2	NID3	NID4	NID5	NID6	NID7	NID8
ELEMENT	4	3	1	12	44	23	8	10	17

EID Identification number of the element (Integer>0)  
 NID1 Local node 1 of the element geometry (Integer>0)  
 NID2 Local node 2 of the element geometry (Integer>0)  
 NID3 Local node 3 of the element geometry (Integer>0)  
 NID4 Local node 4 of the element geometry (Integer>0)  
 NID5 Local node 5 of the element geometry (Integer>0)  
 NID6 Local node 6 of the element geometry (Integer>0)  
 NID7 Local node 7 of the element geometry (Integer>0)  
 NID8 Local node 8 of the element geometry (Integer>0)

The ELEMENT card must have a unique element EID field. That is, no other ELEMENT card may have the same EID value. The nodes are numbered in the local element system. The node numbering scheme is shown in Figure 1.

## ENDDATA Card

The ENDDATA card indicates the end of the data deck.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
ENDDATA									

The ENDDATA card is the last card in the data file.

## FORMULA Card

The FORMULA card allows traction loadings that vary linearly or quadratically along an element edge. Either normal or tangential (or both) tractions are defined with one FORMULA card per element edge.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
FORMULA	EID	SID	INOR	ISHR					
	$A_1$	$A_2$	$A_3$	$A_4$	$A_5$	$A_6$			
	$B_1$	$B_2$	$B_3$	$B_4$	$B_5$	$B_6$			
FORMULA	5	2	1	0					
	5.0	0.0	1.0	0.0	0.0	-2.0			

EID Element identification of the element the load is acting on (Integer>0)  
 SID Side number the load is acting on for element EID (Integer>0)  
 INOR Flag if coefficients  $A_i$  are present to define  $\sigma_n$  INOR>0 - first set of terms defines  $\sigma_n$ .  
 ISHR Flag if coefficients  $B_i$  are present to define  $\sigma_t$ . ISHR>0 - if INOR>0, the second set of terms defines  $\sigma_t$ , otherwise the first set defines  $\sigma_t$ .

SID must be a number between and including 1 and 4. The sides corresponding to SID are shown in Figure 1. The equations for the normal and shear stress on the edge,  $\sigma_n$  and  $\sigma_t$ , respectively, are

$$\sigma_n = A_1 + A_2x + A_3y + A_4x^2 + A_5y^2 + A_6xy$$

and

$$\sigma_t = B_1 + B_2x + B_3y + B_4x^2 + B_5y^2 + B_6xy$$

where the positive and negative directions for the tractions are defined by Figure 2. Also,  $x$  and  $y$  are defined in terms of the global Cartesian coordinates  $(x, y)$ , not some local coordinate system. If the three-dimensional axisymmetric solution is chosen, we write the stresses as

$$\sigma_n = A_1 + A_2r + A_3z + A_4r^2 + A_5z^2 + A_6rz$$

and

$$\sigma_t = B_1 + B_2r + B_3z + B_4r^2 + B_5z^2 + B_6rz$$

where  $r$  and  $z$  are defined in terms of the global cylindrical coordinates  $(r, z)$ .

If INOR is greater than 0, then the next line of the FORMULA card corresponds to  $\sigma_n$ . If INOR is not greater than 0 and ISHR is greater than 0, then the next line of the FORMULA card corresponds to  $\sigma_t$ . If both INOR and ISHR are greater than 0, then the second line of the FORMULA card corresponds to  $\sigma_n$ , and the third line corresponds to  $\sigma_t$ .

## IDEAS Card

The IDEAS card specifies that the graphical output information should be written in an I-DEAS compatible Universal File format. After BUCKY has completed an analysis, the output can be viewed using I-DEAS if this card is included in the input file.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
IDEAS									

## MATBIS Card

The MATBIS card specifies the isotropic elastic constants for a set of elements. The MATBIS card uses the given elastic constants for the bending analysis of the problem. If orthotropic properties are required, the MATBOR card should be used. Any element that does not have a MATBIS or MATBOR card associated with it uses the corresponding MATPIS or MATPOR card for that element.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
MATBIS	E1	E2	$E$	$\nu$					
MATBIS	6	9	10.E6	0.3					

- E1 First element in material property list (Integer>0)  
 E2 Second element in material property list (Integer)  
 E Value of the modulus of elasticity (Real>0.0)  
 $\nu$  Value of Poisson's ratio (Real>0.0)

The value of Poisson's ratio must lie in the range  $0 < \nu < 0.5$  and  $E$  is always a positive number. The first element number, E1, must be a valid element number and greater than zero. The second element number in the list, E2, does not need to be a valid element number. If E2 is less than or equal to E1, then only element E1 has the properties specified on the MATBIS card. Otherwise, all elements, from E1 to E2, have the material properties specified by the MATBIS card.

## MATBOR Card

The MATBOR card specifies orthotropic material constants for a list of elements in the bending analysis. If isotropic properties are used, then the MATBIS card is preferred. Any element that does not have a MATBIS or MATBOR card associated with it uses the corresponding MATPIS or MATPOR card for that element.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
MATBOR	E1	E2	$E_{xx}$	$E_{yy}$	$\nu_{xy}$	$G_{xy}$			
MATBOR	E1	E2	$E_{xx}$	$E_{yy}$	$\nu_{xy}$	$G_{xy}$	$G_{yz}$	$G_{zx}$	
MATBOR	3	5	11.E6	3.E6	0.21	4.3E6			

- E1 First element in material property list (Integer>0)  
 E2 Second element in material property list (Integer)  
 $E_{xx}$  Elastic constant in first direction (Real>0.0)  
 $E_{yy}$  Elastic constant in second direction (Real>0.0)  
 $\nu_{xy}$  Poisson's ratio (Real>0.0)  
 $G_{xy}$  Shear modulus (Real>0.0)  
 $G_{yz}$  Shear modulus (Real>0.0) requires SHEAR card  
 $G_{zx}$  Shear modulus (Real>0.0) requires SHEAR card

The first element in the list, E1, should be greater than zero and be an existing element number. E2 does not need to be a valid element number. If E2 is greater than or equal to E1, then only element E1 has the material properties specified by the MATBOR card. If E2 is greater than E1, then elements E1 to E2 have the properties given by the MATBOR card. When using traditional Kirchoff theory, only Fields 1 through 7 are used. Fields 8 and 9 are accessed only if the SHEAR card is included in the deck (Reissner-Mindlin theory).

## MATPIS Card

The MATPIS card specifies the isotropic elastic constants for a set of elements. The MATPIS card uses the given elastic constants for the plane stress analysis of the problem. If orthotropic properties are required, the MATPOR card should be used.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
MATPIS	E1	E2	$E$	$\nu$					
MATPIS	6	9	10.E6	0.3					

E1 First element in material property list (Integer>0)  
 E2 Second element in material property list (Integer)  
 E Value of the modulus of elasticity (Real>0.0)  
 $\nu$  Value of Poisson's ratio (Real>0.0)

The value of Poisson's ratio must lie in the range  $0 < \nu < 0.5$  and  $E$  is always a positive number. The first element number, E1, must be a valid element number and greater than zero. The second element number in the list, E2, does not need to be a valid element number. If E2 is less than or equal to E1, then only element E1 has the properties specified on the MATPIS card. Otherwise, all elements, from E1 to E2, have the material properties specified by the MATPIS card.

## MATPOR Card

The MATPOR card specifies orthotropic material constants for a list of elements in the plane stress or bending analysis. If isotropic properties are used, then the MATPIS card is preferred.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
MATPOR	E1	E2	$E_{xx}$	$E_{yy}$	$\nu_{xy}$	$G_{xy}$			
MATPOR	E1	E2	$E_{rr}$	$E_{zz}$	$\nu_{rz}$	$G_{rz}$	$\nu_{r\theta}$		
MATPOR	E1	E2	$E_{xx}$	$E_{yy}$	$\nu_{xy}$	$G_{xy}$	$G_{yz}$	$G_{zx}$	
MATPOR	3	5	11.E6	3.E6	0.21	4.3E6			

E1 First element in material property list (Integer>0)  
 E2 Second element in material property list (Integer)  
 $E_{xx}$  Elastic constant in first direction (Real>0.0)  
 $E_{yy}$  Elastic constant in second direction (Real>0.0)  
 $\nu_{xy}$  Poisson's ratio (Real>0.0)  
 $G_{xy}$  Shear modulus (Real>0.0)  
 $G_{yz}$  Shear modulus (Real>0.0) requires SHEAR card  
 $G_{zx}$  Shear modulus (Real>0.0) requires SHEAR card  
 $E_{rr}$  Elastic constant in first direction for axisymmetric analysis (Real>0.0)  
 $E_{zz}$  Elastic constant in second direction for axisymmetric analysis (Real>0.0)  
 $\nu_{rz}$  In-plane Poisson's ratio for axisymmetric analysis (Real>0.0)  
 $G_{rz}$  Out-of-plane shear modulus for axisymmetric analysis (Real>0.0)  
 $\nu_{r\theta}$  Out-of-plane Poisson's ratio for axisymmetric analysis (Real>0.0)

The first element in the list, E1, should be greater than zero and be an existing element number. E2 does not need to be a valid element number. If E2 is greater than or equal to E1, then only element E1 has the material properties specified by the MATPOR card. If E2 is greater than E1, then elements E1 to E2 have the properties given by the MATPOR card.

In the case of three-dimensional axisymmetric plane stress analysis, the strains are related to the stresses via the equations

$$\begin{aligned}\epsilon_{rr} &= -\nu_{rz} \frac{\sigma_{zz}}{E_{zz}} + \frac{\sigma_{rr}}{E_{rr}} - \nu_{r\theta} \frac{\sigma_{\theta\theta}}{E_{rr}}, \\ \epsilon_{\theta\theta} &= -\nu_{rz} \frac{\sigma_{zz}}{E_{zz}} - \nu_{r\theta} \frac{\sigma_{rr}}{E_{rr}} + \frac{\sigma_{\theta\theta}}{E_{rr}}, \\ \epsilon_{zz} &= \frac{\sigma_{zz}}{E_{zz}} - \nu_{rz} \frac{\sigma_{rr}}{E_{zz}} - \nu_{rz} \frac{\sigma_{\theta\theta}}{E_{zz}},\end{aligned}$$

and

$$\gamma_{rz} = \frac{\tau_{rz}}{G_{rz}}.$$

These are the equations for an anisotropic, stratified material.

## MOMENT Card

The MOMENT card allows distributed edge moments that vary linearly or quadratically along an element edge. This option is included for plate bending analysis and is not used in the plane stress or plate buckling solutions. The moments are defined as either acting normally to the edge or along the edge (twist). Either normal or tangential (or both) moments are defined with one FORMULA card per element edge.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
MOMENT	EID	SID	INOR	ITWST					
	A <sub>1</sub>	A <sub>2</sub>	A <sub>3</sub>	A <sub>4</sub>	A <sub>5</sub>	A <sub>6</sub>			
	B <sub>1</sub>	B <sub>2</sub>	B <sub>3</sub>	B <sub>4</sub>	B <sub>5</sub>	B <sub>6</sub>			
MOMENT	5	2	1	0					
	5.0	0.0	1.0	0.0	0.0	-2.0			

EID Element identification of the element the moment is acting on (Integer>0)  
 SID Side number the moment is acting on for element EID (Integer>0)  
 INOR Flag if coefficients A<sub>i</sub> are present to define M<sub>n</sub> INOR>0 - first set of terms defines M<sub>n</sub>.  
 ITWST Flag if coefficients B<sub>i</sub> are present to define M<sub>t</sub>. ITWST>0 - if INOR>0, the second set of terms defines M<sub>t</sub>, otherwise the first set defines M<sub>t</sub>.

SID must be a number between and including 1 and 4. The sides corresponding to SID are shown in Figure 1. The equations for the normal and twisting moment on the edge, M<sub>n</sub> and M<sub>t</sub>, respectively, are

$$M_n = A_1 + A_2x + A_3y + A_4x^2 + A_5y^2 + A_6xy$$

and

$$M_t = B_1 + B_2x + B_3y + B_4x^2 + B_5y^2 + B_6xy$$

where the positive and negative directions for the moments are defined by Figure 3. Also, x and y are defined in terms of the global coordinates (x, y), not some local coordinate system.

If INOR is greater than 0, then the next line of the MOMENT card corresponds to M<sub>n</sub>. If INOR is not greater than 0 and ITWST is greater than 0, then the next line of the MOMENT card corresponds to M<sub>t</sub>. If both INOR and ITWST are greater than 0, then the second line of the MOMENT card corresponds to M<sub>n</sub>, and the third line corresponds to M<sub>t</sub>.

## NODE Card

The NODE card specifies the locations and node identification number for the nodes which make up the problem geometry. In two-dimensional Cartesian analysis, the coordinates are given as  $(x, y)$  pairs. In three-dimensional axisymmetric problems, the coordinates are given as  $(r, z)$  pairs.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
NODE	NID	XLOC	YLOC						
NODE	NID	R	Z						
NODE	10	3.6	123.44						

NID Identification number of the node (Integer>0)  
 XLOC Global  $x$ -coordinate of the node in the Cartesian system (Real)  
 YLOC Global  $y$ -coordinate of the node in the Cartesian system (Real)  
 R Global  $r$ -coordinate of the node in the cylindrical system (Real)  
 Z Global  $z$ -coordinate of the node in the cylindrical system (Real)

The node identification numbers (NIDs) should be unique for each node. That is, no two NODE cards should have the same NID value.

## PATRAN Card

The PATRAN card specifies that the graphical output information should be written in a PATRAN compatible Neutral File format. After BUCKY has completed an analysis, the output can be viewed using PATRAN if this card is included in the input file.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
PATRAN									

## PLANE Card

The PLANE card indicates that a plane stress analysis of the plate structure is desired. The plane stress and bending problems are decoupled from each other. By default, if no analysis type is given in the input data file, a plane stress analysis is performed. However, if a plate bending solution (via the BENDING card) or a plasticity solution (PLASTIC card) is requested, no plane stress analysis is made unless PLANE is included.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
PLANE									

The inclusion of the PLANE card indicates that a plane stress analysis of the plate is to be performed. If the PLANE card is not included in the data file, then a plane stress analysis could be bypassed.

## PLASTIC Card

The PLASTIC card indicates that a plastic analysis of the plate structure is desired.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
PLASTIC									

The inclusion of the PLASTIC card indicates that a plastic analysis of the plate is to be performed. In the current version of BUCKY, only isotropic materials are supported for the plasticity solution.

## PLBOUND Card

The PLBOUND card specifies the plane stress boundary conditions for an element. Either the  $u$  ( $x$ -direction) or the  $v$  ( $y$ -direction) displacement in two-dimensional plane stress analysis or the  $u_r$  ( $r$ -direction) or the  $w$  ( $z$ -direction) displacement in three-dimensional axisymmetric for an element edge may be constrained.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
PLBOUND	EID	SID	IU	IV					
PLBOUND	5	3	1	0					

EID Element identification of the element the constraint is acting on (Integer>0)  
 SID Side number the constraint is acting on for element EID (Integer>0)  
 IU  $u$  displacement code. 1=no displacement, 0=free (Integer)  
 IV  $v$  displacement code. 1=no displacement, 0=free (Integer)

SID must be a number between and including 1 and 4. The sides corresponding to SID are shown in Figure 1. If more than one side of element EID is constrained, a separate card must be used.

## PLPNT Card

The PLPNT card specifies the plane stress boundary conditions for a node on the element. Either the  $u$  ( $x$ -direction) or the  $v$  ( $y$ -direction) displacement or both for the node may be constrained in two-dimensional plane stress analysis. In three-dimensional axisymmetric problems, either the  $u_r$  ( $r$ -direction) or the  $w$  ( $z$ -direction) displacement or both for the node may be constrained. This feature should be used to constrain out rigid body motion.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
PLPNT	EID	NID	IU	IV					
PLPNT	5	35	0	1					

EID Element identification of the element the constraint is acting on (Integer>0)  
 NID Node number the constraint is acting on for element EID (Integer>0)  
 IU  $u$  displacement code. 1=no displacement, 0=free (Integer)  
 IV  $v$  displacement code. 1=no displacement, 0=free (Integer)

## POINT Card

The POINT card allows for point loads to be applied at nodes in the plate bending analysis. The point load acts normally to the element face. A positive value of the point load yields a positive value of displacement.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
POINT	EID	NID	F						
POINT	3	12	100.3						

EID Element identification of the element the load is applied to (Integer>0)  
 NID Node number the load acts on element EID (Integer>0)  
 F Applied nodal force acting on node NID (Real)

## PORDER Card

The PORDER card specifies the polynomial degree of the displacement functions for the plane stress solution. A higher  $p$ , the maximum polynomial degree, gives a more precise result than a lower  $p$ , but solution time is increased.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
PORDER	$p$								
PORDER	6								

$p$  Polynomial degree (Integer>0)

The value of  $p$  is between 4 and 8. Any values of  $p$  below 4 will result in  $p = 4$ , and any  $p > 8$  results in  $p = 8$ .

## PRESS Card

The PRESS card defines the pressure field acting on the plate elements. It is used in the bending analysis of the plate. Due to the nature of the linear plate theory, the bending analysis is decoupled from the buckling and plane stress solutions. The pressure force is always normal to the plate surface.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
PRESS	EID	$P_0$	$P_1$	$P_2$	$P_3$	$P_4$	$P_5$		
PRESS	3	3.0	-1.0	2.5	0.0	0.0	0.0		

EID Identification number of the element (Integer>0)

The pressure field is given by the equation

$$p = P_0 + P_1x + P_2y + P_3x^2 + P_4y^2 + P_5xy \quad (1)$$

where a positive value of  $p$  indicates that the pressure load acts down on the plate element. Also,  $x$  and  $y$  are defined in terms of the global coordinates  $(x, y)$ , not some local element coordinate system.

## SHEAR Card

The SHEAR card specifies that the plate analysis should include the effects of shear deformation. The traditional BUCKY analysis uses the Kirchoff formulation for plate bending and plate buckling. The SHEAR card switches the analysis mode to Reissner-Mindlin plate theory. The program automatically assumes a shear correction factor of 5/6.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
SHEAR									

The inclusion of the SHEAR card indicates the use of Reissner-Mindlin plate theory for the plate bending or plate buckling solution.

## STEPS Card

The STEPS card indicates the number of load increments applied during the plastic analysis of the plate structure. The plate is analyzed at each iteration. For large problems, a high value of STEPS can be very time consuming.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
STEPS	NSTEP								
STEPS	13								

NSTEP      Number of load steps applied during the plastic deformation (Integer>0)

## STRESS Card

The STRESS card defines the loads applied to the edges of the plate elements. Stresses are defined in a normal and tangential coordinate system.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
STRESS	EID	SID	$\sigma_n$	$\sigma_t$					
STRESS	5	2	1000.0	-300.0					

EID            Element identification of the element the load is acting on (Integer>0)  
 SID            Side number the load is acting on for element EID (Integer>0)  
 $\sigma_n$         Normal stress acting along edge SID (Real)  
 $\sigma_t$         Shear stress acting along edge SID (Real)

SID must be a number between and including 1 and 4. The sides corresponding to SID are shown in Figure 1. The definitions of the loads,  $\sigma_n$  and  $\sigma_t$ , are applied according to Figure 2. Positive shear loads ( $\sigma_t$ ) on the STRESS card are in the direction of **t** in Figure 2. Positive normal loads on the STRESS card are in the direction of **n** in Figure 2.

## SUBDIV Card

The SUBDIV card specifies the number of subdivisions to be used for graphic output. A single BUCKY element is divided into the square of the number specified in the SUBDIV card.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
SUBDIV	SUB								
SUBDIV	4								

SUB            Number of divisions (Integer)

The value of SUB is between 1 and 9. Any values of SUB below 1 will result in SUB=1, and any SUB>9 results in SUB=9. The elements are divided into SUB\*SUB subelements for graphical output. For example, if SUB=4, then each element becomes a 4-by-4 element (16 total) mesh in the graphic output files.

## TANGENT Card

The TANGENT card specifies the tangent modulus of the plate elements for the plastic analysis. Elements may have independent values for the tangent modulus.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
TANGENT	E1	E2	$E_T$						
TANGENT	5	8	3.2E5						

E1 First element in thickness list (Integer>0)  
 E2 Second element in thickness list (Integer)  
 $E_T$  Tangent modulus of the material (Real>0.0)

The element list, defined by E1 and E2, gives the element numbers for which the tangent modulus is  $E_T$ . If E2 is less than E1, then only element E1 has the material properties specified in the TANGENT card.

## THICK Card

The THICK card specifies the thickness of the plate. The THICK card allows for linearly varying thickness of one or more elements.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
THICK	E1	E2	$C_1$	$C_2$	$C_3$				
THICK	5	8	.5	-.1	0.0				

E1 First element in thickness list (Integer>0)  
 E2 Second element in thickness list (Integer)

The element list, defined by E1 and E2, gives the thickness equation for elements E1 to E2. If E2 is less than E1, then only element E1 has the thickness given by the current THICK card. If E2 is greater than or equal to E1, then all of the elements from E1 to E2 will have a thickness given by the formula

$$h = C_1 + C_2x + C_3y$$

where  $x$  and  $y$  are the global coordinates (not parametric or element coordinates). The thickness of the element must be everywhere positive and nonzero. The THICK card is not required for axisymmetric problems.

## UPDATE Card

The UPDATE card indicates that during the plastic analysis of the plate structure, the output quantities of displacement and stress are to be updated to the output file at each iteration or a number of iterations. If the UPDATE card is not included, only the final results are given.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
UPDATE	INC								

INC Number of iterations required to be performed to update results (Integer)

The inclusion of the UPDATE card indicates that the deformation during the plastic analysis of the plate is updated at each iteration or after INC number of iterations. This leads to a much larger output data file once the analysis is completed. If INC is not included, the output is updated after each iteration. If INC is zero or negative, no updates will be provided. If INC is greater than 1, then the output is updated after each INC number of iterations.

## YIELD Card

The YIELD card specifies the yield strength of the plate elements for the plastic analysis. Elements may have independent values for the yield strength.

Field 1	Field 2	Field 3	Field 4	Field 5	Field 6	Field 7	Field 8	Field 9	Field 10
YIELD	E1	E2	$\sigma_0$						
YIELD	5	8	50000.0						

E1            First element in thickness list (Integer>0)  
 E2            Second element in thickness list (Integer)  
 $\sigma_0$         Yield strength of the material in simple tension (Real>0.0)

The element list, defined by E1 and E2, gives the element numbers for which the yield strength is  $\sigma_0$ . If E2 is less than E1, then only element E1 has the property defined in the YIELD card.

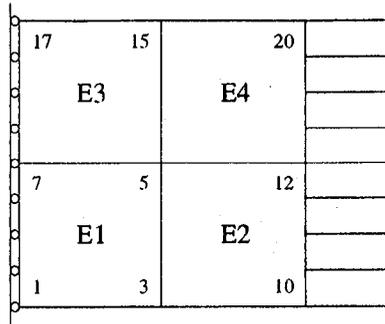


Figure 4: Four-element model of a square plate.

## 9 Examples

This section provides example data files for BUCKY. First, a simple plate is considered, followed by a quarter model of a plate with an internal circular cutout. In both cases, a buckling solution is performed. As an example of plate bending, a square plate subjected to a hydrostatic pressure load is studied. To demonstrate the elastic-plastic capabilities of BUCKY, a plastic analysis of a cantilevered beam is given. As an example of the axisymmetric cylindrical solver within BUCKY, a spherical pressure vessel is examined and the results are compared to the theoretical values.

### 9.1 Square Plate

The first example is a square simply supported plate, as shown in Figure 4. The plate is modeled with  $4 p = 6$  finite elements for both the plane stress and the plate buckling calculations. Rather than make the edges truly simply supported (no displacement or moment), only the geometric constraint (displacement) is enforced. The plane stress boundary conditions are as follows:

- Left edge cannot move in the  $x$ -direction
- One of the nodes of element 1, which lies along the model's centerline, cannot move in the  $y$ -direction. This takes out the rigid body motion of the plate while giving pure compression or tension of a square plate.

The plate is loaded in tension on the right side, so the buckling factor computed by BUCKY should be negative (tension cannot cause buckling). A buckling solution is requested via the BUCKLE card. The entire data file is shown below:

```
#
# Model of a simply supported square plate. Solve for buckling load.
# We can use Reissner-Mindlin plate theory if we uncomment the SHEAR card.
#
#SHEAR
PORDER      6
SUBDIV      4
BUCKLE
NODE        1      0.00    0.00
NODE        2      1.00    0.00
NODE        3      2.00    0.00
NODE        4      2.00    1.00
NODE        5      2.00    2.00
NODE        6      1.00    2.00
NODE        7      0.00    2.00
NODE        8      0.00    1.00
NODE        9      3.00    0.00
```

```

NODE      10      4.00      0.00
NODE      11      4.00      1.00
NODE      12      4.00      2.00
NODE      13      3.00      2.00
NODE      14      2.00      3.00
NODE      15      2.00      4.00
NODE      16      1.00      4.00
NODE      17      0.00      4.00
NODE      18      0.00      3.00
NODE      19      4.00      3.00
NODE      20      4.00      4.00
NODE      21      3.00      4.00
ELEMENT    1         1         2         3         4         5         6         7         8
ELEMENT    2         3         9        10        11        12        13         5         4
ELEMENT    3         7         6         5        14        15        16        17        18
ELEMENT    4         5        13        12        19        20        21        15        14
STRESS     2         2         1.0        0.0
STRESS     4         2         1.0        0.0
#
# Set up a boundary condition to avoid rigid body motion
#
PLBOUND    1         4         1         0
PLBOUND    3         4         1         0
PLPNT      1         6         0         1
THICK      1         4         .375
MATPIS     1         4        1.0E7        0.3
BEND       1         1         1         0
BEND       1         4         1         0
BEND       2         1         1         0
BEND       2         2         1         0
BEND       3         3         1         0
BEND       3         4         1         0
BEND       4         2         1         0
BEND       4         3         1         0
ENDDATA

```

An abbreviated output data file from the run finds that the buckling eigenvalue takes the value

```

*****
* BUCKLING FACTOR *
*****

```

BUCKLING FACTOR=-.318E+06

The reason for the negative sign in the BUCKLING FACTOR is that, in the sample problem, the plate has been loaded in tension. To cause buckling, the loads must compress the plate. Thus, the negative sign represents a direction change for the load from tension to compression. From [16], for simply supported square plates with a constant stress applied to opposite edges, we know that the critical buckling load for the plate is given by

$$\sigma_{cr} = \frac{\pi^2 E h^2}{3(1 - \nu^2) a^2},$$

where the material properties  $E$  and  $\nu$  are given by the MATPIS card;  $h$  is given by the THICK card; and  $a$  is the dimension of the plate. For this sample problem,  $a$  takes the value  $a = 4$ . Thus, the theoretical buckling load is

equal to

$$\sigma_{cr} = 317746,$$

which varies from the computational value by only 0.08%.

If we instead consider a shear deformable plate, we realize that the addition of the shear terms in the strain energy yields a softer structure. This in turn requires that the buckling stress  $\sigma_{cr}$  be reduced by some amount. For the plate considered, the thickness-to-width ratio is fairly large at 0.09375. For this value of the thickness-to-width ratio and for an aspect ratio of 1, we would expect to find a buckling load of approximately 92% of the non-shear deformable plate [15]. Thus, we would expect the computer model to find a buckling load near 292000. When using BUCKY to determine the buckling load, we find a value given by

```
*****
* BUCKLING FACTOR *
*****
```

```
BUCKLING FACTOR=-.278E+06
```

While this value for the buckling load is below that predicted by [15], it should be emphasized that the same value for the buckling factor was computed for  $p = 8$ , suggesting convergence of the results.

## 9.2 Square Plate with Cutout, Quarter Model

The next example is that of a square plate with a circular cutout in the interior of the plate, as shown in Figure 5. A quarter of the plate is modeled since the loading and boundary conditions are symmetric. Also, in the case of buckling, one should assume that, since the plate is being loaded in compression and since the loading is symmetric about both  $x$  and  $y$  axes, the first buckled mode shape is symmetric, which leads to the lowest buckling load. If the loads or boundary conditions were not symmetric, this would not be the case. So taking advantage of symmetry, the data file reads:

```
PORDER      6
BORDER      8
SUBDIV      4
#
# Dump the output to an I-DEAS file for post processing
#
IDEAS
BUCKLE
NODE        1      1.00      0.00
NODE        2      1.50      0.00
NODE        3      2.00      0.00
NODE        4      1.9239     .3827
NODE        5      1.8478     .7654
NODE        6      1.3858     .574
NODE        7      .9239      .3827
NODE        8      .9808      .1951
NODE        9      3.00      0.00
NODE       10      4.00      0.00
NODE       11      4.00      1.00
NODE       12      4.00      2.00
NODE       13      2.9239     1.3827
NODE       14      1.6310     1.0898
NODE       15      1.4142     1.4142
NODE       16      1.0607     1.0607
```

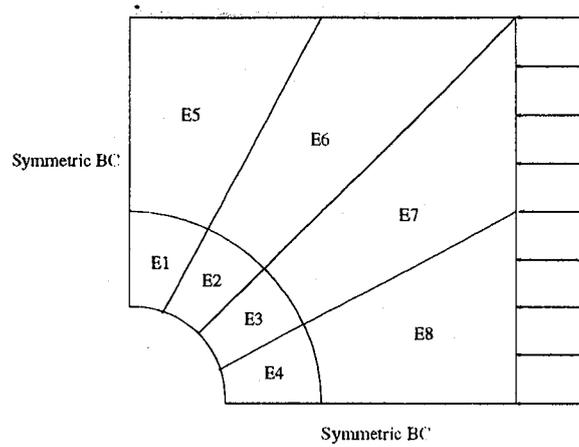


Figure 5: Quarter model of a square plate with an internal circular cutout.

NODE	17	.7071	.7071						
NODE	18	.8315	.5556						
NODE	19	4.00	3.00						
NODE	20	4.00	4.00						
NODE	21	2.7071	2.7071						
NODE	22	1.0898	1.6310						
NODE	23	.7654	1.8478						
NODE	24	.574	1.3858						
NODE	25	.3827	.9239						
NODE	26	.5556	.8315						
NODE	27	3.00	4.00						
NODE	28	2.00	4.00						
NODE	29	1.3827	2.9239						
NODE	30	.3827	1.9239						
NODE	31	0.00	2.00						
NODE	32	0.00	1.50						
NODE	33	0.00	1.00						
NODE	34	.1951	.9808						
NODE	35	1.00	4.00						
NODE	36	0.00	4.00						
NODE	37	0.00	3.00						
ELEMENT	1	33	34	25	24	23	30	31	32
ELEMENT	2	25	26	17	16	15	22	23	24
ELEMENT	3	17	18	7	6	5	14	15	16
ELEMENT	4	7	8	1	2	3	4	5	6
ELEMENT	5	31	30	23	29	28	35	36	37
ELEMENT	6	23	22	15	21	20	27	28	29
ELEMENT	7	15	14	5	13	12	19	20	21
ELEMENT	8	5	4	3	9	10	11	12	13
STRESS	7	3	-1.0	0.0					
STRESS	8	3	-1.0	0.0					
PLBOUND	1	4	1	0					
PLBOUND	5	4	1	0					
PLBOUND	4	2	0	1					
PLBOUND	8	2	0	1					
THICK	1	8	1.0						
MATPIS	1	8	1.0E7	0.3					

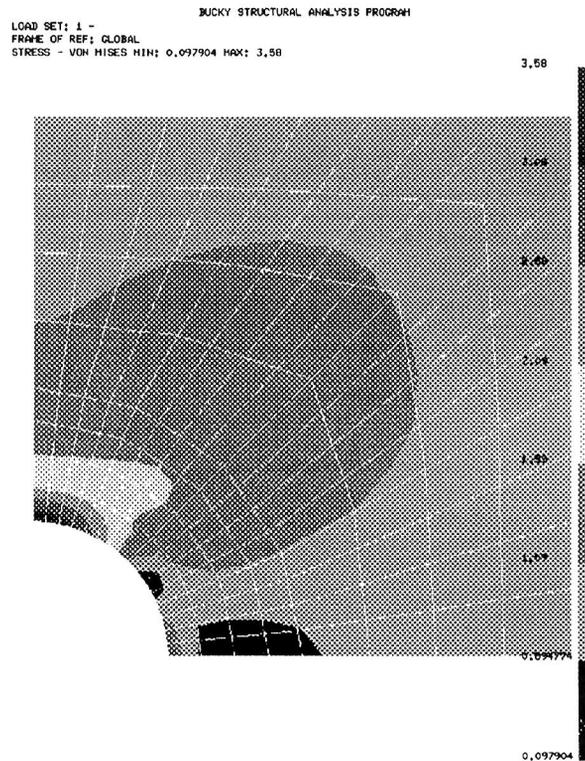


Figure 6: von Mises stress distribution in a square plate with cutout.

```

BEND      1      4      0      1
BEND      4      2      0      1
BEND      5      4      0      1
BEND      5      3      1      0
BEND      6      3      1      0
BEND      7      3      1      0
BEND      8      3      1      0
BEND      8      2      0      1
ENDDATA
  
```

In the file above, elements 1 and 5 are restricted to vertical translation only on edge IV. This edge is one of the lines of symmetry for the model, the other being edge II of elements 4 and 8, which are restricted to only moving in the  $x$ -direction. The bending boundary conditions correspond to a simply supported full model. However, since the model used is a quarter scale model, some alterations need to be made. Since the aforementioned edges are along lines of symmetry, symmetrical boundary conditions need to be imposed. These are displacement and slope along the edge. The displacement condition is satisfied automatically if the slope condition is utilized. The slope of the plate along the lines of symmetry must be zero for bending. Thus, for side II of elements 4 and 8 and side IV of elements 1 and 5, the slope is constrained to be zero. This will satisfy the symmetrical boundary conditions and lead to the lowest buckling load and mode for the plate. For the plane stress portion of the problem, a polynomial order of  $p = 6$  has been chosen. However, for the plate buckling analysis, the fidelity is increased with a higher polynomial order ( $p = 8$ ) through the BORDER card.

To show an example of the output capabilities, the von Mises stress distribution from the plane stress analysis is plotted using I-DEAS. Figure 6 show this distribution. Notice the fringes near the hole; these are indicative in cases of stress concentrations. Also, notice that the stress in this region is larger than the input stress.

### 9.3 Square Plate Under Hydrostatic Pressure Load

Using BUCKY version 4.0, the response of plates to pressure loads can be investigated. This section gives a sample BUCKY input file for the case of a square simply supported plate that is subject to a linearly varying pressure load. This is the special case of hydrostatic pressure loads. The same plate used in the example of Section 9.1 is used here. The input deck for this analysis is

```
#
# Full model of a square plate with a linearly varying hydrostatic load.
# We can include the effects of transverse shear deformation by uncommenting
# the SHEAR card entry.
#
PORDER      8
SUBDIV      4
BENDING
#
# Use shear deformable plates
#
#SHEAR
NODE        1      0.00   0.00
NODE        2      1.00   0.00
NODE        3      2.00   0.00
NODE        4      2.00   1.00
NODE        5      2.00   2.00
NODE        6      1.00   2.00
NODE        7      0.00   2.00
NODE        8      0.00   1.00
NODE        9      3.00   0.00
NODE       10      4.00   0.00
NODE       11      4.00   1.00
NODE       12      4.00   2.00
NODE       13      3.00   2.00
NODE       14      2.00   3.00
NODE       15      2.00   4.00
NODE       16      1.00   4.00
NODE       17      0.00   4.00
NODE       18      0.00   3.00
NODE       19      4.00   3.00
NODE       20      4.00   4.00
NODE       21      3.00   4.00
ELEMENT      1      1      2      3      4      5      6      7      8
ELEMENT      2      3      9     10     11     12     13     5      4
ELEMENT      3      7      6      5     14     15     16     17     18
ELEMENT      4      5     13     12     19     20     21     15     14
THICK        1      4      .375
MATPIS       1      4     1.0E6     0.3
BEND         1      1      1      0
BEND         1      4      1      0
BEND         2      1      1      0
BEND         2      2      1      0
BEND         3      3      1      0
BEND         3      4      1      0
BEND         4      2      1      0
BEND         4      3      1      0
PRESS        1      0.0     1.0
```

```

PRESS      2      0.0    1.0
PRESS      3      0.0    1.0
PRESS      4      0.0    1.0
ENDDATA

```

In the data deck, a bending only solution is requested by the BENDING card. The pressure load, defined by the PRESS card, suggests that there is no constant component of the pressure and the pressure varies linearly with  $x$ . The pertinent data from the BUCKY run are shown below:

Element	Node	Z-disp.	Sigma-x	Sigma-y	Tau-xy
1	1	-.74489E-08	.38721E-01	.28948E-01	-.35275E+02
	2	-.49027E-12	.35787E-01	.11780E+00	-.28158E+02
	3	-.16803E-08	-.15428E-01	-.94797E-01	-.67955E+01
	4	.31149E-03	.48644E+02	.53051E+02	-.35236E+01
	5	.43068E-03	.65400E+02	.65326E+02	.34991E-01
	6	.27794E-03	.35810E+02	.40874E+02	-.21140E-03
	7	-.14823E-07	.36584E-01	.90827E-01	-.27523E-01
	8	-.69993E-08	.16355E-02	.41951E-03	-.23408E+02
	C	.20021E-03	.26436E+02	.32451E+02	-.18398E+02
2	3	-.50663E-08	-.35790E-01	-.15257E+00	-.67938E+01
	9	-.79859E-09	.83295E-01	.27658E+00	.26302E+02
	10	-.38995E-08	.14616E-01	.11284E-01	.53242E+02
	11	-.41117E-08	.38379E+00	.11574E+00	.31054E+02
	12	.10071E-07	-.33953E+00	-.19051E+00	-.38612E-01
	13	.34507E-03	.70298E+02	.56391E+02	.38591E-02
	5	.43072E-03	.64900E+02	.64943E+02	.43519E-01
	4	.31149E-03	.48637E+02	.53050E+02	-.35242E+01
	C	.25193E-03	.54038E+02	.48016E+02	.18072E+02
3	7	.45674E-08	-.50101E-01	-.31118E-01	-.25719E-01
	6	.27796E-03	.35813E+02	.40884E+02	.95495E-03
	5	.43070E-03	.65314E+02	.65176E+02	.35226E-01
	14	.31149E-03	.48642E+02	.53050E+02	.35199E+01
	15	.33879E-08	-.49871E-01	-.12788E+00	.67674E+01
	16	.16800E-09	.36531E-01	.12028E+00	.28157E+02
	17	-.13435E-09	-.28200E-02	.33582E-02	.35298E+02
	18	-.17554E-09	-.34464E-03	-.31173E-03	.23410E+02
	C	.20021E-03	.26438E+02	.32451E+02	.18397E+02
4	5	.43068E-03	.65024E+02	.65135E+02	.33437E-01
	13	.34504E-03	.70296E+02	.56381E+02	-.25838E-02
	12	-.20221E-07	-.20317E+00	-.93715E-02	-.41627E-01
	19	-.13344E-07	.38360E+00	.11518E+00	-.31051E+02
	20	-.13396E-07	.79401E-01	.42919E-01	-.53208E+02
	21	.93919E-10	.83718E-01	.27914E+00	-.26302E+02
	15	-.11491E-07	.34852E-01	-.69179E-01	.67535E+01
	14	.31147E-03	.48640E+02	.53049E+02	.35209E+01

C .25193E-03 .54034E+02 .48011E+02 -.18071E+02

In the sample case above, we can compare the computational solution to known textbook solutions. From classical theory, we know that at the center of the plate, the plate deflection is given by the expression [17]

$$w_{cen} = .00203 \frac{q_0 a^4}{D}$$

This is the expression for the theoretical plate deflection for the center of a plate subject to a hydrostatic (linearly increasing) pressure load on its surface. In the example problem, the parameter  $a$  is the span of the plate;  $D$  is the modulus of rigidity of the plate; and  $q_0$  is the maximum value of load. The modulus of rigidity is related to the material and physical properties of the plate. For an isotropic plate of constant thickness,  $D$  is constant and given by

$$D = \frac{Eh^3}{12(1-\nu^2)}$$

The values of  $E$  and  $\nu$  are given by the MATPIS card. The thickness, which is constant in this case, is given by the THICK card, and the parameter  $a$  takes the value  $a = 4$ . Finally, from the PRESS card, we have prescribed a pressure load of

$$p(x, y) = x$$

on the plate. At  $x = 4$ ,  $p$  has its maximum of  $p = 4$ . Thus, the quantity  $q_0$  takes the value  $q_0 = 4$ , and the central deflection is found to be

$$w_{cen} = .00043045.$$

By comparison, the central deflection in the plate model is given by the deflection at node 5. We see that at node 5, the deflection takes the value

$$w_{node5} = .00043068,$$

which represents negligible error.

For the same loading, the classical expression for the stresses can also be compared to the computational results. In the case above, it was found that at node 6 of element 1 the x and y stresses,  $\sigma_x$  and  $\sigma_y$ , take the values

$$\sigma_x = 35.81 \quad \text{and} \quad \sigma_y = 40.874.$$

From reference [17], we find that

$$\sigma_x = 6M_x/h^2 = 6\beta_1 q_0 a^2/h^2 \quad \text{and} \quad \sigma_y = 6M_y/h^2 = 6\beta_2 q_0 a^2/h^2,$$

where

$$\beta_1 = .0132 \quad \text{and} \quad \beta_2 = .0149$$

for the point corresponding to node 6. Thus, we find that the theoretical stresses are given by

$$\sigma_x = 36.045 \quad \text{and} \quad \sigma_y = 40.687,$$

whereas the computational values are

$$\sigma_{x_{node6}} = 35.810 \quad \text{and} \quad \sigma_{y_{node6}} = 40.874.$$

If we include the effects of shear deformation into the problem, then we find that the new central deflection is given by

$$w_{node5} = .00048444,$$

which is 12.48% larger than the non-shear deformable central deflection. Similarly, the stresses at node 6 in the model are slightly larger with the inclusion of shear deformations. For this location, we find that the stresses are

$$\sigma_{x_{node6}} = 39.705 \quad \text{and} \quad \sigma_{y_{node6}} = 44.354.$$

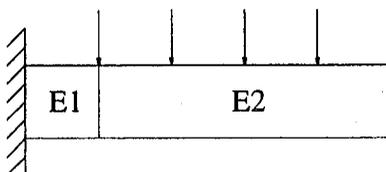


Figure 7: Two-element beam model for elastic-plastic analysis.

#### 9.4 Plastic Analysis of a Uniformly Loaded Cantilever Beam

To show the capabilities of BUCKY in plasticity problems, a cantilever beam subjected to a uniformly distributed pressure load is studied. The plasticity solution is an iterative plane stress solution. The finite element model is comprised of only two elements, as shown in Figure 7. To effectively trace the plastic growth, two elements are used, with the smaller one near the beam base being the same order as the beam thickness. The input deck for this analysis is

```
#
# Elastic-plastic analysis of a uniformly loaded cantilever
#
PORDER      6
UPDATE      10
PLASTIC
NODE         1      0.0      0.0
NODE         2      1.0      0.0
NODE         3      2.0      0.0
NODE         4      2.0      1.0
NODE         5      2.0      2.0
NODE         6      1.0      2.0
NODE         7      0.0      2.0
NODE         8      0.0      1.0
NODE         9      6.0      0.0
NODE        10     10.0      0.0
NODE        11     10.0      1.0
NODE        12     10.0      2.0
NODE        13      6.0      2.0
ELEMENT      1      1      2      3      4      5      6      7      8
ELEMENT      2      3      9     10     11     12     13     5      4
STRESS       1      3     -505.
STRESS       2      3     -505.
STRESS       1      1      505.
STRESS       2      1      505.
YIELD        1      2     50000.
#
# Specify a 20 step plastic analysis
#
STEPS        20
TANGENT      1      2      1.0E0
MATPIS       1      2     10.0E6      0.0
PLBOUND      1      4      1      1
THICK        1      2      1.0
ENDDATA
```

The output from this run is quite extensive. To reduce the amount of out generated, only the displacements are given for the outer element. It should be noted that the applied load given in the STRESS cards is slightly above

the static collapse pressure of the beam. The exact collapse pressure is 1000 psi (or 500 psi on each face). The applied pressure in this example is 1010 psi, a 1% increase over the theoretical collapse pressure. Pressures much higher than this will result in ever-increasing tip deflections that are unbounded. The output for the second element (displacements only) is

\*\*\* PLASTICITY ITERATION # 1 COMPLETE \*\*\*

\*\*\* PERCENTAGE OF LOAD APPLIED = 65.48% \*\*\*

Element	Node	X-disp.	Y-disp.
2	3	-.81764E-02	-.10073E-01
	9	-.15528E-01	-.62242E-01
	10	-.16534E-01	-.12795E+00
	11	.44650E-11	-.12793E+00
	12	.16534E-01	-.12795E+00
	13	.15528E-01	-.62242E-01
	5	.81764E-02	-.10073E-01
	4	-.22120E-12	-.10054E-01
	C	.11969E-12	-.62222E-01

\*\*\* PLASTICITY ITERATION #10 COMPLETE \*\*\*

\*\*\* PERCENTAGE OF LOAD APPLIED = 81.01% \*\*\*

Element	Node	X-disp.	Y-disp.
2	3	-.10373E-01	-.12956E-01
	9	-.19464E-01	-.78517E-01
	10	-.20708E-01	-.16083E+00
	11	.55711E-11	-.16080E+00
	12	.20708E-01	-.16083E+00
	13	.19464E-01	-.78517E-01
	5	.10373E-01	-.12956E-01
	4	-.22696E-12	-.12939E-01
	C	.19595E-12	-.78492E-01

\*\*\* PLASTICITY ITERATION #20 COMPLETE \*\*\*

\*\*\* PERCENTAGE OF LOAD APPLIED = 98.27% \*\*\*

Element	Node	X-disp.	Y-disp.
2	3	-.16826E-01	-.23165E-01
	9	-.27840E-01	-.11963E+00
	10	-.29347E-01	-.23639E+00
	11	.68262E-11	-.23637E+00
	12	.29347E-01	-.23639E+00
	13	.27840E-01	-.11963E+00
	5	.16826E-01	-.23165E-01
	4	-.18119E-12	-.23158E-01
	C	.27599E-12	-.11960E+00

\*\*\* PLASTICITY ITERATION #21 COMPLETE \*\*\*

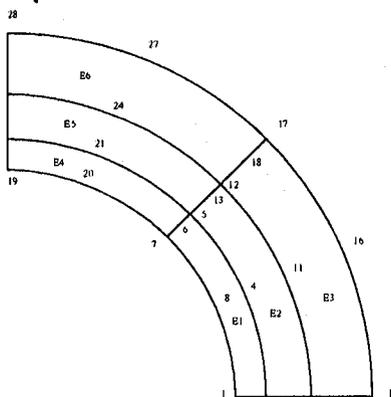


Figure 8: Axisymmetric sphere model.

\*\*\* PERCENTAGE OF LOAD APPLIED = 100.00% \*\*\*

Element	Node	X-disp.	Y-disp.
2	3	-.18274E-01	-.25590E-01
	9	-.29472E-01	-.12833E+00
	10	-.31005E-01	-.25173E+00
	11	.69644E-11	-.25170E+00
	12	.31005E-01	-.25173E+00
	13	.29472E-01	-.12833E+00
	5	.18274E-01	-.25590E-01
	4	-.17072E-12	-.25587E-01
	C	.29341E-12	-.12830E+00

## 9.5 Axisymmetric Analysis of a Spherical Pressure Vessel

A traditional benchmark for finite element programs is the examination of the characteristics of spherical pressure vessels under some internal pressure. The exact solution is easily found [18]. The finite element model of the sphere is a six-element model. We can take advantage of symmetry and model only one hemisphere of the vessel. In addition, because of axisymmetry, we model only a 90° segment of the sphere. The finite element model is shown in Figure 8. A pressure  $p_0$  of 3600 psi acts on the interior of the sphere. The inner radius of the tank is  $a = 1.735$  in., and the outer radius is given by  $b = 1.85$  in. The input deck for this analysis is

```
#
# Axisymmetric analysis of a spherical pressure vessel under pressure
#
AXIS3D
PORDER      6
IDEAS
SUBDIV      6
NODE        1   1.735   0.
NODE        2   1.7425  0.
NODE        3   1.75    0.
NODE        4   1.617   .670
NODE        5   1.237   1.237
NODE        6   1.232   1.232
NODE        7   1.227   1.227
NODE        8   1.603   .664
```

```

NODE      9      1.77      0.
NODE     10      1.79      0.
NODE     11      1.654     .685
NODE     12      1.266     1.266
NODE     13      1.252     1.252
NODE     14      1.82      0.
NODE     15      1.85      0.
NODE     16      1.709     .708
NODE     17      1.308     1.308
NODE     18      1.287     1.287
NODE     19      0.       1.735
NODE     20      .664     1.603
NODE     21      .670     1.617
NODE     22      0.       1.75
NODE     23      0.     1.7425
NODE     24      .685     1.654
NODE     25      0.       1.79
NODE     26      0.       1.77
NODE     27      .708     1.709
NODE     28      0.       1.85
NODE     29      0.       1.82
ELEMENT   1       7       8       1       2       3       4       5       6
ELEMENT   2       5       4       3       9      10      11      12      13
ELEMENT   3      12      11      10      14      15      16      17      18
ELEMENT   4      19      20       7       6       5      21      22      23
ELEMENT   5      22      21       5      13      12      24      25      26
ELEMENT   6      25      24      12      18      17      27      28      29
MATPIS    1       6     28.5E6    0.3
#
# We are using symmetry about z=0, so include symmetric boundary conditions
# for the elements along this line
#
PLBOUND   1       2       0       1
PLBOUND   2       2       0       1
PLBOUND   3       2       0       1
#
# Include constraints against radial motion along the r=0 line.  BUCKY
# automatically enforces it, so it is really superfluous
#
PLBOUND   4       4       1       0
PLBOUND   5       4       1       0
PLBOUND   6       4       1       0
STRESS    1       1     -3600.    0.
STRESS    4       1     -3600.    0.
ENDDATA

```

Since six elements have been used, the output is extensive. However, we can compare the normal and tangential stresses in the elements to the theoretical predictions. According to [18], the radial and circumferential stresses in a spherical pressure vessel are given by

$$\sigma_r(r) = -\frac{p_0 a^3}{(a^3 - b^3)} \left(1 - \frac{b^3}{r^3}\right) \quad \text{and} \quad \sigma_\theta(r) = -\frac{p_0 a^3}{(a^3 - b^3)} \left(1 + \frac{b^3}{2r^3}\right).$$

The equations for  $\sigma_r$  and  $\sigma_\theta$  were derived in a spherical coordinate system with symmetry. Thus, the stresses are a function of  $r$  only. In the cylindrical system, displacements and stresses are functions of two field variables,  $r$  and

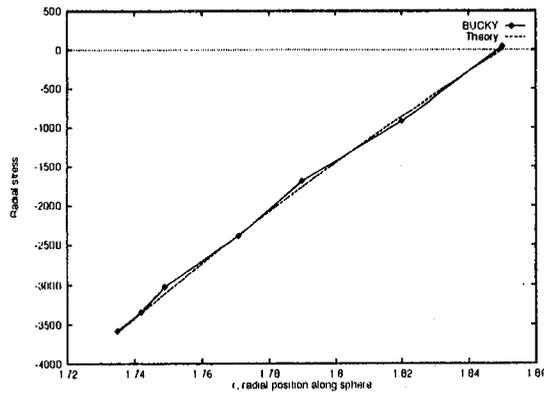


Figure 9: Radial stress distribution in a spherical shell.

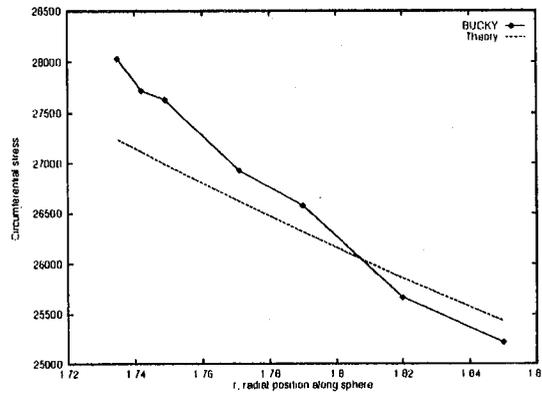


Figure 10: Circumferential stress distribution in a spherical shell.

z. However, for this problem, the principal stresses,  $\sigma_1$  and  $\sigma_2$  in the  $r - z$  plane, are equivalent to the radial and circumferential stresses in the spherical system. Additionally, the stress  $\sigma_\theta$  computed by BUCKY is theoretically equivalent to the principal stress representing the circumferential stress ( $\sigma_1$ ).

Figure 9 shows the radial stress distribution along a straight line joining nodes 5, 6, 7, 12, 13, 17, and 18 and the stresses corresponding to the theoretical prediction at those nodes. Figure 10 shows the circumferential stress distribution along the same straight line. From Figure 9, we see excellent agreement of the radial stress along the shell wall. The similarities in the circumferential stress, given by Figure 10, are not as appealing. However, the relative error between the theoretical and numerical result is small.

## 10 Error Messages

This section gives a complete list of all the error messages BUCKY is capable of posting. Along with the messages are reasons for and ways to correct the problem.

**\*\*\* ERROR 1 \*\*\* UNRECOGNIZED INPUT CARD** - BUCKY found a data card that is not in the BUCKY library of valid input cards. Check the spelling of the card or delete card.

**\*\*\* ERROR 2 \*\*\* ILLEGAL NODE NUMBER IN NODE CARD** - A node must have an identification number of at least the value one. A zero or negative value for the node number will cause this error. Change the node number to a positive value.

**\*\*\* ERROR 3 \*\*\* ILLEGAL NODE NUMBER IN ELEMENT CARD** - An element has a node in its connectivity data that is not a valid node number. Check the ELEMENT card to make sure the proper node is specified. If so, modify or create a new NODE card to create this missing node.

**\*\*\* ERROR 4 \*\*\* ILLEGAL ELEMENT NUMBER IN ELEMENT CARD** - An element must have an identification number of at least the value one. A zero or negative value for the element number will cause this error. Change the element number to a positive value.

**\*\*\* ERROR 5 \*\*\* ILLEGAL ELEMENT NUMBER IN STRESS CARD** - The element referenced in the STRESS card is nonexistent. Either create the missing element with the ELEMENT card or change the element number in the STRESS card to an existing element.

**\*\*\* ERROR 6 \*\*\* ILLEGAL SIDE NUMBER IN STRESS CARD** - An element has four sides, labeled from one to four. If this error occurs, a number less than one or greater than four was entered as the side number. Supply a number greater than zero and less than five.

**\*\*\* ERROR 7 \*\*\* ILLEGAL ELEMENT NUMBER IN PLBOUND CARD** - The element referenced in the PLBOUND card is nonexistent. Either create the missing element with the ELEMENT card or change the element number in the PLBOUND card to an existing element.

**\*\*\* ERROR 8 \*\*\* ILLEGAL SIDE NUMBER IN PLBOUND CARD** - An element has four sides, labeled from one to four. If this error occurs, a number less than one or greater than four was entered as the side number. Supply a number greater than zero and less than five.

**\*\*\* ERROR 9 \*\*\* ILLEGAL CONSTRAINT CODE IN PLBOUND CARD** - The constraint codes for this card must be zero or one. This error indicates neither a zero nor one value was entered for the IU condition. Enter the appropriate number for the IU condition.

**\*\*\* ERROR 10 \*\*\* ILLEGAL CONSTRAINT CODE IN PLBOUND CARD** - The constraint codes for this card must be zero or one. This error indicates neither a zero nor one value was entered for the IV condition. Enter the appropriate number for the IV condition.

**\*\*\* ERROR 11 \*\*\* ILLEGAL POLYNOMIAL ORDER IN PORDER CARD** - Valid polynomial orders for BUCKY are from four to eight. Specify a value from four to eight.

**\*\*\* ERROR 12 \*\*\* ILLEGAL ELEMENT NUMBER IN THICK CARD** - The element referenced in the THICK card is nonexistent. Either create the missing element with the ELEMENT card or change the element number in the THICK card to an existing element.

**\*\*\* ERROR 13 \*\*\* ILLEGAL ELEMENT NUMBER IN MATPOR CARD** - The element referenced in the MATPOR card is nonexistent. Either create the missing element with the ELEMENT card or change the element number in the MATPOR card to an existing element.

**\*\*\* ERROR 14 \*\*\* ILLEGAL ELEMENT NUMBER IN MATPIS CARD** - The element referenced in the MATPIS card is nonexistent. Either create the missing element with the ELEMENT card or change the element number in the MATPIS card to an existing element.

**\*\*\* ERROR 15 \*\*\* ILLEGAL ELEMENT NUMBER IN BEND CARD** - The element referenced in the BEND card is nonexistent. Either create the missing element with the ELEMENT card or change the element number in the BEND card to an existing element.

**\*\*\* ERROR 16 \*\*\* ILLEGAL SIDE NUMBER IN BEND CARD** - An element has four sides, labeled from one to four. If this error occurs, a number less than one or greater than four was entered as the side number. Supply a number greater than zero and less than five.

**\*\*\* ERROR 17 \*\*\* ILLEGAL CONSTRAINT CODE IN BEND CARD** - This error corresponds to Field 4 of the BEND card, which refers to the displacement boundary condition. Valid values for IC1 are zero and one. This error indicates neither of these values was used. Enter a zero or one in the proper field to rectify the error.

**\*\*\* ERROR 18 \*\*\* ILLEGAL CONSTRAINT CODE IN BEND CARD** - This error corresponds to Field 4 of the BEND card, which refers to the slope boundary condition. Valid values for IC2 are zero and one. This error indicates neither of these values was used. Enter a zero or one in the proper field to rectify the error.

**\*\*\* ERROR 21 \*\*\* ILLEGAL SUBDIVISIONS IN SUBDIV CARD** - The number of valid element subdivisions is from one to eight. Any other number of subdivisions is invalid.

**\*\*\* ERROR 22 \*\*\* NODE IN ELEMENT IS INVALID** - A nonexistent node was found in the element connectivity information for the current element. Create a new NODE card or change the element connectivity to remedy the error.

**\*\*\* ERROR 23 \*\*\* APPLIED LOAD ACTS ON NON-EXISTENT ELEMENT** - A STRESS card was found which acts on a nonexistent element. Check the STRESS card and change the EID or create a new ELEMENT card with the same EID.

**\*\*\* ERROR 24 \*\*\* PLANE STRESS BOUNDARY CONDITION ACTS ON NON-EXISTENT ELEMENT** - A PLBOUND card was found which acts on a nonexistent element. Check the PLBOUND card and change the EID or create a new ELEMENT card with the same EID.

**\*\*\* ERROR 25 \*\*\* BENDING BOUNDARY CONDITION ACTS ON NON-EXISTENT ELEMENT** - A BEND card was found which acts on a nonexistent element. Check the BEND card and change the EID or create a new ELEMENT card with the same EID.

**\*\*\* ERROR 26 \*\*\* ILLEGAL ELEMENT NUMBER IN PLPNT CARD** - The element referenced in the PLPNT card is nonexistent. Either create the missing element with the ELEMENT card or change the element number in the PLPNT card to an existing element.

**\*\*\* ERROR 27 \*\*\* ILLEGAL CONSTRAINT CODE IN PLPNT CARD** - Valid constraint codes for IU or IV are zero and one. This error indicates neither of these values was found for either IU or IV. Change the field in question to a zero or one.

**\*\*\* ERROR 28 \*\*\* PLANE STRESS POINT CONSTRAINT ACTS ON NON-EXISTENT ELEMENT** - A PLPNT card was found which acts on a nonexistent element. Check the PLPNT card and change the EID or create a new ELEMENT card with the same EID.

**\*\*\* ERROR 29 \*\*\* PLANE STRESS POINT CONSTRAINT ACTS ON NON-EXISTENT NODE** - A PLPNT card was found which acts on a nonexistent node. Check the PLPNT card and change the NID or create a new NODE card with the same NID.

**\*\*\* ERROR 33 \*\*\* NUMBER OF NODES EXCEEDS ALLOWABLE AMOUNT** - BUCKY has a predetermined limit on the number of nodes allowed for a problem, and this error indicates the model has exceeded the memory requirements. Reduce the number of nodes in the model.

**\*\*\* ERROR 34 \*\*\* NUMBER OF ELEMENTS EXCEEDS ALLOWABLE AMOUNT** - BUCKY has a predetermined limit on the number of elements allowed for a problem, and this error indicates the model has exceeded the memory requirements. Reduce the number of elements in the model.

**\*\*\* ERROR 35 \*\*\* ILLEGAL ELEMENT NUMBER IN FORMULA CARD** - The element referenced in the FORMULA card is nonexistent. Either create the missing element with the ELEMENT card or change the element number in the FORMULA card to an existing element.

**\*\*\* ERROR 36 \*\*\* ILLEGAL SIDE NUMBER IN FORMULA CARD** - An element has four sides, labeled from one to four. If this error occurs, a number less than one or greater than four was entered as the side number. Supply a number greater than zero and less than five.

**\*\*\* ERROR 37 \*\*\* NUMBER OF ELEMENTS WITH STRESS CARDS EXCEEDS ALLOWABLE AMOUNT** - BUCKY has a predetermined limit on the number of STRESS cards allowed for a problem, and this error indicates the model has exceeded the memory requirements. Reduce the number of STRESS cards in the model.

**\*\*\* ERROR 38 \*\*\* NUMBER OF PLANE STRESS BOUNDARY CARDS EXCEEDS ALLOWABLE AMOUNT** - BUCKY has a predetermined limit on the number of PLBOUND cards allowed for a problem, and this error indicates the model has exceeded the memory requirements. Reduce the number of PLBOUND cards in the model.

**\*\*\* ERROR 39 \*\*\* NUMBER OF BENDING BOUNDARY CARDS EXCEEDS ALLOWABLE AMOUNT** - BUCKY has a predetermined limit on the number of BEND cards allowed for a problem, and this error indicates the model has exceeded the memory requirements. Reduce the number of BEND cards in the model.

**\*\*\* ERROR 40 \*\*\* NUMBER OF PLANE STRESS POINT CONSTRAINT CARDS EXCEEDS ALLOWABLE AMOUNT** - BUCKY has a predetermined limit on the number of PLPNT cards allowed for a problem, and this error indicates the model has exceeded the memory requirements. Reduce the number of PLPNT cards in the model.

**\*\*\* ERROR 41 \*\*\* TWO NODE CARDS HAVE THE SAME NODE ID** - NODE cards must have unique node numbers, so a repeated number will result in this error. Change or delete one of the NODE cards.

**\*\*\* ERROR 42 \*\*\* TWO ELEMENT CARDS HAVE THE SAME ELEMENT ID** - ELEMENT cards must have unique element numbers, so a repeated number will result in this error. Change or delete one of the ELEMENT cards.

**\*\*\* ERROR 44 \*\*\* ILLEGAL ELEMENT NUMBER IN MATBOR CARD** - The element referenced in the MATBOR card is invalid. The element number should be a positive number. Change the element identification number to a nonzero positive value.

**\*\*\* ERROR 45 \*\*\* ILLEGAL ELEMENT NUMBER IN MATBIS CARD** - The element referenced in the MATBIS card is invalid. The element number should be a positive number. Change the element identification number to a nonzero positive value.

**\*\*\* ERROR 46 \*\*\* ILLEGAL ELEMENT NUMBER ON PRESS CARD** - The element referenced in the PRESS card is not valid. The element number must have a positive value.

**\*\*\* ERROR 47 \*\*\* ILLEGAL POLYNOMIAL ORDER IN BORDER CARD** - Valid polynomial orders for BUCKY are from four to eight. Specify a value from four to eight.

**\*\*\* ERROR 48 \*\*\* ILLEGAL ELEMENT NUMBER ON MOMENT CARD** - The element referenced in the MOMENT card is invalid. Valid element numbers are positive numbers. If the element referenced in the MOMENT card is zero or negative, this error will be posted. Change the element number to a positive value.

**\*\*\* ERROR 49 \*\*\* ILLEGAL SIDE NUMBER IN MOMENT CARD** - An element has four sides, labeled from one to four. If this error occurs, a number less than one or greater than four was entered as the side number. Supply a number greater than zero and less than five.

**\*\*\* ERROR 50 \*\*\* ILLEGAL ELEMENT NUMBER IN YIELD CARD** - The element referenced in the YIELD card is invalid. The element number should be greater than zero. Change the element number in this card to a positive value.

**\*\*\* ERROR 51 \*\*\* ILLEGAL NUMBER OF LOAD STEPS IN STEPS** - The number of load step increments for the plastic analysis must be greater than zero. Specify a nonzero or nonnegative value.

**\*\*\* ERROR 52 \*\*\* ILLEGAL ELEMENT NUMBER IN TANGENT CARD** - The element referenced in the TANGENT card is invalid. The element number should be greater than zero. Change the element number in this card to a positive value.

**\*\*\* ERROR 53 \*\*\* ILLEGAL ELEMENT NUMBER IN POINT CARD** - The element referenced in the POINT card is invalid. The element number should be greater than zero. Change the element number in this card to a positive value.

**\*\*\* ERROR 54 \*\*\* POINT LOAD ACTS ON NON-EXISTENT ELEMENT** - Either create the missing element with the ELEMENT card or change the element number in the POINT card to an existing element.

**\*\*\* ERROR 55 \*\*\* PRESSURE LOAD ACTS ON NON-EXISTENT ELEMENT** - Either create the missing element with the ELEMENT card or change the element number in the PRESSURE card to an existing element.

**\*\*\* ERROR 56 \*\*\* MOMENT ACTS ON NON-EXISTENT ELEMENT** - Either create the missing element with the ELEMENT card or change the element number in the MOMENT card to an existing element.

**\*\*\* ERROR 100 \*\*\* BAD JACOBIAN** - A bad Jacobian is one where the Jacobian is zero or negative. This is due to poor element geometry such as corner and side angles being too large or too small. Another cause of a bad Jacobian may be improper node connectivity information. Check the ELEMENT and NODE cards for improper data.

**\*\*\* ERROR 101 \*\*\* NON-POSITIVE THICKNESS** - The thickness of a plate element must be everywhere positive, and this error indicates a zero or negative thickness was encountered. Check the THICK card for negative values in the global domain of the element.

**\*\*\* ERROR 102 \*\*\* NON-POSITIVE RADIUS** - The radius of an element must always be positive, and this error indicates a negative radius was encountered. Check the input geometry and be sure there are no negative values in the  $R$  column of the NODE cards.

**\*\*\* ERROR 200 \*\*\* ERROR IN DECOMPOSITION, CHECK CONSTRAINTS** - For the most part, you will find this error if you have a modeling problem. Check the geometry through PATRAN or some other means and also the boundary conditions. This error is called when a negative or zero is found on the main diagonal of the stiffness matrix, signifying a non-positive definite matrix. The two most common reasons for this error are poor geometry and improper constraints, such as neglecting to constrain out the rigid body motion of the structure. Make sure the structure is constrained in both  $x$  and  $y$  directions for plane stress and is sufficiently constrained in plate bending (at least one edge fixed against displacement and another fixed against rotation).

**\*\*\* ERROR 201 \*\*\* ITERATIONS EXCEEDED MAXIMUM, EIGENVALUE MAY BE IN ERROR** - BUCKY uses the inverse power method to determine the buckling load of a loaded plate. Should the eigenvalue ever fail to converge to the criteria specified in the program, this error will be printed and the calculated eigenvalue may be incorrect. Every effort has been made to assure convergence.

**\*\*\* ERROR 202 \*\*\* DECOMPOSITION ERROR IN LEAST SQUARES FIT** - During the plastic analysis, the stresses and strains are computed at the Gauss points within the element. However, the Gauss points are not a

preferable location for the output of the stresses and strains. BUCKY performs a nonlinear least squares fit of the data at the Gauss points to come up with an equation for the stresses and strains anywhere in the plate. If this error occurs, a problem was found with the data and the data in the output may be incorrect.

**\*\*\* ERROR 203 \*\*\* DIFFICULTY CONVERGING IN LEAST SQUARES FIT** - The nonlinear least squares analysis is an iterative process. The routine should converge within only a few steps; but if for some reason it does not converge after a certain number of steps, this message will be printed. The output data may have some inaccuracies in them.

**\*\*\* ERROR 999 \*\*\* MEMORY REQUIREMENTS EXCEEDED** - BUCKY has a predetermined core memory usage requirement. If the model exceeds this requirement, then an error is posted. Two things dictate the size of the problem: the mesh and the polynomial order. If a high polynomial order is being used, reduce the  $p$  level by one or two to see if the problem fits in core memory. Otherwise, the mesh may need to be less refined such that the number of elements is reduced.

## 11 BUCKY Theoretical Supplement

### 11.1 Introduction

Plate structures are a common component in aerospace structures. Whether it is a piece of aircraft or a satellite, virtually every structure incorporates the use of plates in its design. For example, aircraft wings utilize plates for the skin. In addition, floor panels within aircraft, or even the Space Shuttle, are comprised of plates. These components take loads, and the structural engineer must show, with confidence, that the loads do not damage the substructure or lead to catastrophic failure of the entire system itself.

Failure of a system can be in terms of fracture, plastic yielding, or buckling. Indeed, often the buckling load is used as a limit load for the structure. To prevent against such failures, a medium through which the analysis of plates can be made must be known. Finite element methods provide an approximation to the behavior of the plates in a system. Furthermore, for complex structures, finite elements are often the only viable method of solution.

In the past decade, the field of finite element theory has been somewhat split into two areas of concentration. The traditional approach to finite element theory, called the  $h$ -version, has been to establish a library of low-order (linear) elements and to apply meshing techniques to approach the exact solution to the governing differential equations. Recently, research has been done into the effectiveness of keeping the mesh fixed and increasing the order of the solution. Called the  $p$ -version of the finite element method, it reduces the error associated with the trial solution by increasing the order of the solution well beyond the linear estimations of  $h$ -finite elements.

The  $p$ -finite element method applied to plates is not well developed. The plane stress portion of the problem has found its way to commercial codes; but in general, the plate bending problem is not often seen, and the plate buckling solution has never been seen before in other codes. Plate bending, in some cases, has been dropped in favor of full three-dimensional approaches. However, if a plate is expanded from Kirchoff plate theory to a fully three-dimensional problem, the expense skyrockets. Thus, there is some merit in studying the plate bending problem in the smaller two-dimensional space.

In the field of buckling of structures, work with  $h$ -version finite elements has been done with success, and some of it has gained acceptance into widely available commercial programs. However, with the exception of the author, relatively little work has been done extending the  $p$ -version of the finite element method to thin plate buckling. In linear problems of two- and three-dimensional elasticity, the  $p$ -version of the finite element method has proven to be exceptional, and we would expect the attractive qualities to carry over to plate buckling.

The problem of plate buckling is a two-part analysis. In the first part, the stress distribution throughout the plate structure must be found. This step involves solving a plane stress problem for a plate undergoing distributed edge tractions (shear and normal stresses). The solution of such a problem has already been implemented in several commercial  $p$ -finite element software packages. Next, using the stresses from the plane stress analysis, the plate buckling solution proceeds. A plate will buckle if the stresses in the plate element are large enough to cause geometrically nonlinear deflections of the plate. Thus, if the edge loads are below the buckling load, the plate only deforms in the plane of the plate. If the edge loads have a magnitude equal to the buckling load, then the plate deforms out-of-plane, and large deformations ensue. Theoretically, the deflections and, thus, the stresses are unbounded, so we wish to stay below the critical buckling load, and we solve for the minimum buckling load of the plate. This second part, which yields the critical buckling factor, is an eigenvalue problem.

The plate bending problem is somewhat easier in that it is only a one-part solution method. In linear theory, there is a decoupling of the membrane and bending action, so the plate bending problem does not require the stress information from a plane stress analysis.

The computer program BUCKY introduced here performs the two-part buckling solution for plates of arbitrary shapes. For buckling, textbook solutions and tabular information of the critical buckling load exist for rectangular, elliptical, circular, triangular, and skewed plates, but plates with irregular shapes must be solved using an approximate method. BUCKY solves a given problem by mapping the irregular plate to a square domain, where the analysis is made easier. In addition to the geometrical mapping features of BUCKY, other highlights include

- Polynomial trial solutions of degree 4 to 8
- Isotropic and orthotropic plate material properties
- Quadratically varying edge tractions
- Quadratically varying pressure loads for plate bending analysis

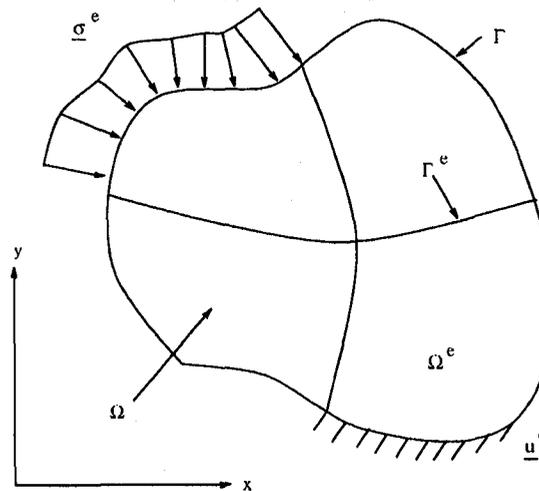


Figure 11: Structure broken into a four-element model.

- Linearly varying element thicknesses
- Extensive solution output
- Graphical post-processing support.

Providing a full set of analysis modes, the user may choose between performing plane stress, plate bending, plate buckling, or plane stress plasticity solutions. Furthermore, any combination of the solutions can be requested without problem.

In this paper, we discuss some of the features of BUCKY. Also, we present some of the underlying theory to plane stress and plate buckling analysis, as well as finite element theory. Plate bending shares many common components with plate buckling, so its discussion follows closely with the buckling material. Orthotropic material properties are discussed, since their importance is presently seen with advanced composite materials. Finally, some example problems are solved, and some remarks about the  $p$ -version finite element as it is applied to plate buckling conclude this work.

## 11.2 Problem Statement

For buckling, suppose we wish to find the response of the plate structure of Figure 11 under some loading situation. The plate is defined over the region  $\Omega$ , and it is broken into several elements  $\Omega^e$ . The plate boundary is given by  $\Gamma$ , and the element boundaries are given by  $\Gamma^e$ . It follows that  $\Gamma^e \in \Gamma$  and  $\Omega^e \in \Omega$ .

Stresses in the form of normal and shear tractions act on the elements  $\Omega^e$  at the boundaries  $\Gamma^e$ . Denoted by  $\sigma^e$ , these are known quantities. In addition, there may be pressure forces acting perpendicularly to the plate surface. To constrain the plate against rigid body motion, we impose the boundary conditions  $\mathbf{u}^e$  on  $\Gamma^e$ . Next, we perform a plane stress analysis on  $\Omega$  subject to the sum total of the applied stresses  $\sigma$ .

In the case of plate bending, we have pressure forces  $p(x, y)$  acting normally to the plate surface. Again, the plate has certain prescribed boundary conditions  $\mathbf{u}_e$  on  $\Gamma^e$ . We wish to find the response of the body  $\Omega$  to the set of loads  $p(x, y)$ .

## 11.3 Minimum Potential Energy

Using the theorem of minimum potential energy, approximate solutions to difficult problems can be found. Indeed, the entire basis of finite elements is grounded in energy theorems. The principal behind minimum potential energy is that, given a particular problem, an approximate solution is guessed. With this approximate solution comes an inherent error. To reduce the error, the total energy is minimized and the minimized function is set to zero. For the plane stress problem, this gives a linear system of equations, and for the buckling solution, an eigenvalue problem is created.

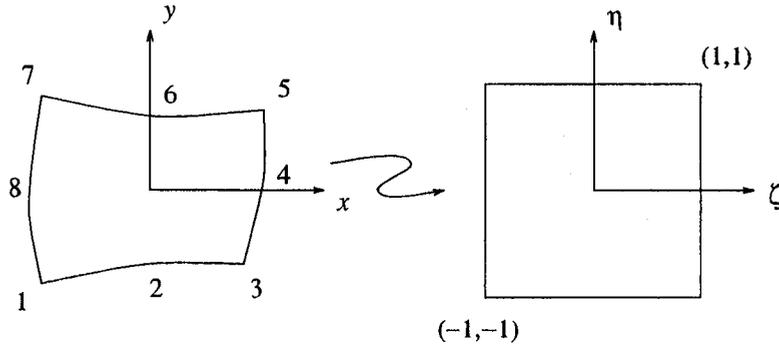


Figure 12: Subparametrically mapped element from physical to parametric space.

Traditional  $h$ -convergent finite elements utilize energy theorems by selecting certain shape functions as the approximate solutions. These shape functions are nodally based such that, when minimized, the solution vector is a vector of nodal displacements and rotations. On the other hand, the method used here is not nodally based. Rather, displacements are not found at the nodes, so the solution vector does not contain nodal parameters. Instead, coefficients to an assumed displacement function are found, which in turn defines the displacements anywhere in the element, including the nodes. Regardless of which method is followed, the potential energy can be written as

$$\Pi = U - T = e, \tag{2}$$

where  $U$  is the strain energy of the plate;  $T$  is the work of the applied loads;  $\Pi$  is the energy; and  $e$  is the error associated with the approximate displacement function. The strain energy of the plate is

$$U = \frac{1}{2} \int_V \{\epsilon\}^T \{\sigma\} dV. \tag{3}$$

where  $\{\sigma\}$  is the stress and  $\{\epsilon\}$  is the strain in the body. In (2),  $T$  is a vector for linear static analysis (plane stress and plate bending) and a matrix for buckling analysis.

To solve a given problem, (2) is minimized with respect to the coefficients of the assumed displacement function. If the coefficients of the displacement function are contained in the vector  $\{a\}$ , then minimizing (2) requires

$$\delta \Pi = \left( \frac{\partial U}{\partial a_i} - \frac{\partial T}{\partial a_i} \right) da_i = 0. \tag{4}$$

As mentioned earlier, depending on how  $T$  is defined, (4) gives either a linear system or an eigenvalue problem.

One aspect of either method ( $h$ -convergent finite elements or the approach used here) is that the system produced by (4) is singular as it stands. Before either the nodal displacements ( $h$ -finite element) or the displacement function coefficients can be found, constraint conditions must be imposed. Boundary conditions can be added to (2) with Lagrange multipliers. By writing the constraints  $R_i$  such that  $R_i = 0$ , no energy is added to (2), and equation (2) is

$$\bar{\Pi} = \Pi + \lambda_i R_i = U - T + \lambda_i R_i. \tag{5}$$

Now equation (5) is minimized such that

$$\delta \bar{\Pi} = \left[ \frac{\partial U}{\partial a_i} - \frac{\partial T}{\partial a_i} + \frac{\partial}{\partial a_i} (\lambda_i R_i) \right] da_i + \frac{\partial}{\partial \lambda_i} (\lambda_i R_i) d\lambda_i = 0. \tag{6}$$

### 11.4 Subparametric Mappings

To integrate the energy functional easily, a subparametric mapping to a new domain is made. The new domain is generally a square for two-dimensional analysis. A mapping of an eight-noded shape in a Cartesian  $(x, y)$  system to the  $(\zeta, \eta)$  domain is shown in Figure 12. This mapping is done to easily integrate the strain energy and the work of the loads, which are done using Gauss integration techniques. The mapping functions are written as

$$x = m_1(\zeta, \eta) \quad \text{and} \quad y = m_2(\zeta, \eta). \quad (7)$$

Using information on the shape of the element, the functions  $m_1$  and  $m_2$  are easily found. With the mapping functions known, the strain energy can be integrated in the  $(\zeta, \eta)$  domain using numerical integration. Gauss quadrature allows integration of functions by simply evaluating functions at certain points, called Gauss points, rather than actually integrating using a technique such as Simpson's Rule or Romberg integration.

The subparametric mapping allows us to rewrite the strain energy. Recalling the definition for strain energy in Cartesian coordinates, the strain energy in the  $(\zeta, \eta)$  domain becomes

$$U = \frac{1}{2} \int_V \{\epsilon\}^T \{\sigma\} dV = \frac{1}{2} \int_{\hat{V}} \{\epsilon\}^T \{\sigma\} |J| d\hat{V}$$

where  $|J|$  is the Jacobian, and  $d\hat{V}$  is the infinitesimal volume in the mapped domain. In two-dimensional analysis,  $d\hat{V} = d\zeta d\eta dh$ , where  $h$  is the plate thickness.

## 11.5 Plate Buckling

The derivation of the buckling theory in this section can be found in [16] and [8]. The results of these references are given here. For an orthotropic plate undergoing compressive and shear loads in its mid-plane, the governing differential equation can be shown to be [8]

$$D_x \frac{\partial^4 w}{\partial x^4} + 2H \frac{\partial^4 w}{\partial x^2 \partial y^2} + D_y \frac{\partial^4 w}{\partial y^4} = N_x \frac{\partial^2 w}{\partial x^2} + 2N_{xy} \frac{\partial^2 w}{\partial x \partial y} + N_y \frac{\partial^2 w}{\partial y^2} \quad (8)$$

where  $w$  is the out-of-plane displacement of the plate. Notice that ordinary plate theory is used and that  $w$  is a function of  $x$  and  $y$  only. The parameters  $N_x$ ,  $N_y$ , and  $N_{xy}$  are the compressive and shear loads, with dimensions of force per unit length.  $N_x$  corresponds to a compressive load in the  $x$ -direction;  $N_y$  is the  $y$ -direction load; and  $N_{xy}$  is the shear load acting on the plate boundary.  $N_x$ ,  $N_y$ , and  $N_{xy}$  are unknowns and must be found to compute the critical buckling loads. The orthotropic material constants of the plate,  $D_x$ ,  $D_y$ , and  $H$ , are given by

$$D_x = \frac{E_x h^3}{12}, \quad D_y = \frac{E_y h^3}{12}, \quad H = D_{xy} + 2G_{xy}, \quad D_{xy} = \frac{E_{xy} h^3}{12}, \quad \text{and} \quad G_{xy} = \frac{Gh^3}{12}. \quad (9)$$

Exact solutions to (8) are few, so approximate solutions are necessary. Equation (8) can be recast using the theory of minimum potential energy, giving

$$\Pi = U_b - T_b \quad (10)$$

where

$$U_b = \frac{1}{2} \int_{\Omega} \left[ D_x \left( \frac{\partial^2 w}{\partial x^2} \right)^2 + 2D_{xy} \frac{\partial^2 w}{\partial x^2} \frac{\partial^2 w}{\partial y^2} + D_y \left( \frac{\partial^2 w}{\partial y^2} \right)^2 + 4G_{xy} \left( \frac{\partial^2 w}{\partial x \partial y} \right)^2 \right] d\Omega \quad (11)$$

and

$$T_b = \frac{1}{2} \int_{\Omega} \left[ N_x \left( \frac{\partial w}{\partial x} \right)^2 + 2N_{xy} \frac{\partial w}{\partial x} \frac{\partial w}{\partial y} + N_y \left( \frac{\partial w}{\partial y} \right)^2 \right] d\Omega. \quad (12)$$

Equation (10) results due to the existence of a strain energy function. This limits the analysis to materials with material symmetry (i.e., the material is not completely anisotropic). The expression  $U_b$  is the strain energy due to bending of the plate. Equation (12) corresponds to the geometrically nonlinear portion of the plate deflection. For example, the strain in the  $x$ -direction for the plate is

$$\epsilon_x = -z \frac{\partial^2 w}{\partial x^2} + \frac{du}{dx} + \frac{1}{2} \left( \frac{\partial w}{\partial x} \right)^2 \quad (13)$$

where large deformations are assumed. The first expression on the right-hand side of (13) is the strain due to bending, and it relates to (11). The last two expressions of (13) are the nonlinear components of the strain in the plate. In plate buckling theory, it is assumed that the plate is inextensible in  $u$  and  $v$ . Thus,  $du/dx = 0$ , and the geometrically nonlinear strain energy is given by (12).

To compute the values of  $N_x$ ,  $N_y$ , and  $N_{xy}$ , we take the first variation of (10) and set it equal to zero, or

$$\delta\Pi = 0. \quad (14)$$

Setting the first variation of (10) to zero results in an eigenvalue problem involving  $N_x$ ,  $N_y$ , and  $N_{xy}$ . If the edge loads are related, then we have an eigenvalue problem in terms of only  $N_x$ ,  $N_y$ , or  $N_{xy}$ .

## 11.6 Plate Bending

The plate bending problem is simpler than the plate buckling problem. Rather than resulting in an eigenvalue problem, the plate bending problem is inherently a linear system, assuming linear elasticity theory is used. The differences between plate bending and plate buckling are small. The main distinction is the governing differential equation. The governing equation for plate buckling, equation (8), contains expressions for the mid-plane loads  $N_x$ ,  $N_y$ , and  $N_{xy}$ . In plate bending, we make the assumption that these mid-plane loads are nonexistent. The work done on the system comes from transverse pressure loads and distributed edge moments. Denoted by  $p$ , these pressures act perpendicularly to the plate surface. Thus, if the plate is acted on by pressure forces alone, we can rewrite (8) for plate bending, such that

$$D_x \frac{\partial^4 w}{\partial x^4} + 2H \frac{\partial^4 w}{\partial x^2 \partial y^2} + D_y \frac{\partial^4 w}{\partial y^4} = p. \quad (15)$$

In terms of the strain energy of bending, the left-hand side of (15) corresponds to the bending strain energy of (11). However, we no longer include the energy (12) in the plate bending analysis, but instead replace it with the work of the pressure forces  $p$ , such that

$$T_b = \int_{\Omega} p w d\Omega. \quad (16)$$

In the case where distributed edge moments act on the plate, the work of the moments is given by the integral over the edge of the moment times the slope along that edge.

## 11.7 Plane Stress

Unfortunately, the buckling problem cannot be solved directly for irregularly shaped plates. In the case of rectangular plates, the integration involved with (11) and (12) is straightforward. Also, in this case, the edge loads are easily definable on the domain  $\Omega$ , so the integration of (12) is not problematic. However, once the plate tends from regularity, the behavior of the loads  $N_x$ ,  $N_y$ , and  $N_{xy}$  inside  $\Omega$  vary spatially. Thus, we must know how  $N_x$ ,  $N_y$ , and  $N_{xy}$  vary within  $\Omega$  so that we can incorporate their effects. To do this, we must perform a plane stress analysis of the plate structure.

In the case of plane stress analysis of a plate, the potential energy of the system is given by

$$\Pi = U_p - T_p \quad (17)$$

where

$$U_p = \frac{1}{2} \int_{-h/2}^{h/2} \int_{\Omega} \left[ E_x \left( \frac{du}{dx} \right)^2 + E_y \left( \frac{dv}{dy} \right)^2 + 2E_{xy} \frac{du}{dx} \frac{dv}{dy} + G_{xy} \left( \frac{du}{dy} + \frac{dv}{dx} \right)^2 \right] d\Omega dz \quad (18)$$

and

$$T_p = \int_{-h/2}^{h/2} \int_{\Gamma} \sigma \cdot \mathbf{u} d\Gamma dz. \quad (19)$$

Using (18) and (19) and taking the first variation of (31) yields the deformed configuration of the plate in response to the applied loads. Once the deformed configuration is found, the stresses on  $\Omega$  can be found, and subsequently,  $N_x$ ,  $N_y$ , and  $N_{xy}$  can be scaled accordingly to one another. At any point in the plate,  $N_x = \sigma_x h$ ,  $N_y = \sigma_y h$ , and  $N_{xy} = \sigma_{xy} h$ .

## 11.8 Three-Dimensional Axisymmetric Analysis

Three-dimensional problems can be solved using a two-dimensional approach if axisymmetry is imposed. Imagine a body of revolution in the cylindrical  $(r, \theta, z)$  coordinate system. If the body has identical geometry, boundary conditions, and loading conditions for all values of  $\theta$ , the problem is said to be one of axisymmetry. In this case, the  $\theta$  dependence is safely dropped, and the problem becomes one of two dimensions in the  $(r, z)$  plane.

In the case of cylindrically symmetric problems, there are two degrees of freedom associated with a structure. Denoted by  $u_r(r, z)$  and  $u_z(r, z)$ , these are displacements in the radial and longitudinal directions, respectively. Furthermore, of the six available components of stress, four are nonzero:  $\sigma_r$ ,  $\sigma_z$ ,  $\sigma_\theta$ , and  $\tau_{rz}$ . In all cases, the  $\theta$  dependence disappears from the problem. In the case of axisymmetry, the nonzero strains are

$$\epsilon_r = \frac{\partial u_r}{\partial r}, \quad \epsilon_\theta = \frac{u_r}{r}, \quad \epsilon_z = \frac{\partial u_z}{\partial z}, \quad \text{and} \quad \gamma_{rz} = \frac{\partial u_z}{\partial r} + \frac{\partial u_r}{\partial z}. \quad (20)$$

The stresses  $\{\sigma\}^T = \{\sigma_r \ \sigma_\theta \ \sigma_z \ \tau_{rz}\}$  are related to the strains  $\{\epsilon\}^T = \{\epsilon_r \ \epsilon_\theta \ \epsilon_z \ \gamma_{rz}\}$  via the constitutive relation

$$\{\sigma\} = [C]\{\epsilon\} \quad (21)$$

where

$$[C] = \begin{bmatrix} E_{11} & E_{12} & E_{13} & E_{14} \\ E_{12} & E_{22} & E_{23} & E_{24} \\ E_{13} & E_{23} & E_{33} & E_{34} \\ E_{14} & E_{24} & E_{34} & E_{44} \end{bmatrix}. \quad (22)$$

In the case of axisymmetric stratified material, the material constants  $E_{14}$ ,  $E_{24}$ , and  $E_{34}$  in (22) are identically zero (see Zienkiewicz [20]). We can also write the material constants out explicitly as

$$[C] = \frac{E_{zz}}{(1 + \nu_{r\theta})(1 - \nu_{r\theta} - 2n\nu_{rz}^2)} \begin{bmatrix} 1 - \nu_{r\theta}^2 & n\nu_{rz}(1 + \nu_{r\theta}) & n\nu_{rz}(1 + \nu_{r\theta}) & 0 \\ n\nu_{rz}(1 + \nu_{r\theta}) & n(1 - \nu_{rz}^2) & n(\nu_{r\theta} + n\nu_{rz}^2) & 0 \\ n\nu_{rz}(1 + \nu_{r\theta}) & n(\nu_{r\theta} + n\nu_{rz}^2) & n(1 - n\nu_{rz}^2) & 0 \\ 0 & 0 & 0 & m(1 + \nu_{r\theta})(1 - \nu_{r\theta} - 2n\nu_{rz}^2) \end{bmatrix}, \quad (23)$$

where  $n = E_{rr}/E_{zz}$  and  $m = G_{rz}/E_{zz}$ .

We recall that we can write the strain energy  $U$  in the form

$$U = \frac{1}{2} \int_V \{\epsilon\}^T [C] \{\epsilon\} dV = \frac{1}{2} \int_\theta \int_z \int_r \{\epsilon\}^T [C] \{\epsilon\} r dr dz d\theta, \quad (24)$$

where the variable  $r$  has been included as a product of a coordinate transformation from Cartesian to cylindrical. Since  $\{\epsilon\}$  and  $[C]$  are independent of  $\theta$ , (24) involves an integration in two variables,  $r$  and  $z$ . Inserting (20) and (22) into (24) yields

$$\begin{aligned} U = & \pi \int_z \int_r \left[ E_{11} \left( \frac{\partial u_r}{\partial r} \right)^2 + 2E_{12} \frac{u_r}{r} \frac{\partial u_r}{\partial r} + 2E_{13} \frac{\partial u_r}{\partial r} \frac{\partial u_z}{\partial z} + E_{22} \frac{u_r^2}{r^2} + 2E_{23} \frac{u_r}{r} \frac{\partial u_z}{\partial z} \right. \\ & \left. + E_{33} \left( \frac{\partial u_z}{\partial z} \right)^2 + E_{44} \left( \frac{\partial u_z}{\partial r} + \frac{\partial u_r}{\partial z} \right)^2 \right] r dr dz. \end{aligned} \quad (25)$$

We can integrate (25) in the parametric  $(\zeta, \eta)$  domain as before by selecting displacement functions

$$u_r = a_{ij} \zeta^i \eta^j \quad \text{and} \quad u_z = b_{ij} \zeta^i \eta^j \quad (26)$$

and mapping functions

$$r = m_1(\zeta, \eta) \quad \text{and} \quad z = m_2(\zeta, \eta). \quad (27)$$

The cylindrical axisymmetric development of the work of the applied loads progresses in much the same way as for the two-dimensional plane stress case. Writing the stresses as

$$\sigma = \sigma_r e_r + \sigma_z e_z, \quad (28)$$

the work of  $\sigma$  is given by

$$T = \int_{\Gamma} \int_{\theta} \sigma \cdot \mathbf{u} r d\theta d\Gamma = 2\pi \int_{\Gamma} \sigma \cdot \mathbf{u} r d\Gamma, \quad (29)$$

where

$$\mathbf{u} = u_r \mathbf{e}_r + u_z \mathbf{e}_z. \quad (30)$$

Thus, (29) differs from the plane stress case by an inclusion of the variable  $r$  in the integration.

## 11.9 Plane Stress Implementation

The  $p$ -version of the finite element method for plane stress is well documented, so only the features of BUCKY which typify the excellent accuracy of  $p$ -finite elements are included here. Currently, an eight-noded quadrilateral plate element is used for analysis. The nodes define the element geometry only. With eight nodes, edge curvature can be easily implemented. While the element has eight defining nodes, the order of the element's displacement function varies from  $p = 4$  to  $p = 8$ . Using the serendipity functions for the displacement, the degrees of freedom associated with the element vary from 17 ( $p = 4$ ) to 47 ( $p = 8$ ) for each component of displacement. Hence, in plane stress analysis, one BUCKY element has as many degrees of freedom as a dozen or more low-order  $h$ -finite elements.

Plate elements may have constant thickness or linearly varying thickness in the element, such as

$$h^e = C_1 + C_2 x + C_3 y \quad (x, y) \in \Omega^e. \quad (31)$$

With a linearly varying thickness, simple three-dimensional problems can be modeled as two-dimensional plates. The plate material of BUCKY elements is such that individual elements can have unique orthotropic or isotropic properties. Some plates, such as ribbed and corrugated panels, have different bending characteristics than for plane stress. In this case, the plate is not assumed to be isotropic, but orthotropic. The new material constants are usually related to the moments of inertia of the rib sections.

Edge tractions in BUCKY are quite flexible and amenable to general loading configurations. Stresses are defined in normal/tangential components on an element boundary  $\Gamma^e$ , and can vary quadratically over the element boundary, such that by

$$\sigma_n^e = A_1 + A_2 x + A_3 y + A_4 x^2 + A_5 y^2 + A_6 xy \quad (x, y) \in \Omega^e \quad (32)$$

and

$$\sigma_t^e = B_1 + B_2 x + B_3 y + B_4 x^2 + B_5 y^2 + B_6 xy \quad (x, y) \in \Omega^e \quad (33)$$

where  $\sigma_n^e$  and  $\sigma_t^e$  denote the element normal and shear stress on  $\Gamma^e$ .

We require that the total energy of the system be positive definite, so we must constrain the plate against rigid body motion. BUCKY allows for two types of constraints: edge and point constraints. Both forms of constraints are rigid; that is, no elastic boundary conditions are currently supported. The edge boundary condition constrains an entire element edge in the desired direction, rather than at the nodes, as  $h$ -finite element theory does. The point constraint constrains only a node against motion. This type of constraint is analogous to the point constraints found in the  $h$ -finite element theory. This type of constraint does not constrain the entire edge.

Once a BUCKY plane stress analysis is complete, the plate response can be investigated through BUCKY's output files or through graphical means. BUCKY output files include information on nodal displacements and stresses (including principal and von Mises stresses). In addition to information at the eight defining nodes, the response at the element centers are computed as well. For a graphical view of the plate response, BUCKY supports the commercial program PATRAN, a pre- and post-processor for many of today's standard finite element software packages. When writing the PATRAN file, BUCKY divides each BUCKY element into a number of specified subelements (the user chooses the number). By breaking the large elements into smaller elements, PATRAN receives a better scattering of data with which to show the plate response. At most, 1 BUCKY element can be broken into a 9-by-9 matrix of 81 subelements, with 4 corner nodes each, to enhance the study of the plate.

## 11.10 Plate Buckling Implementation

As mentioned earlier, it is believed that BUCKY is the first attempt to extend the  $p$ -version finite element method to plate buckling. Every attempt has been made to assess the accuracy of BUCKY in this new realm. Unfortunately, the number of known solutions for plates with irregular shapes, boundary conditions, or loading patterns is few,

but for known solutions, BUCKY correlates extremely well. Furthermore, buckling solutions of plates with varying thickness are virtually nonexistent. Recall that one of the capabilities of BUCKY is the implementation of a linearly varying element thickness. The effects of plates with taper are not largely known, so BUCKY provides an excellent test bed for such an investigation.

In some cases, the bending behavior of plates is much different than the behavior when the plate is in a state of plane stress. In bending, the effects of the ribs dominate if the plate is ribbed, and the isotropic material takes on orthotropic properties. It is for this reason that BUCKY allows for a separate set of material constants for bending. As in plane stress, both isotropic and orthotropic properties are available.

The boundary conditions for the buckling analysis involve the plate deflection and slope. Either the displacement or the first derivative (or both) of the displacement can be set to zero. With the choice of boundary conditions, plates with simply supported, clamped, and free edges can be modeled, as well as those with edges free to translate but not rotate. In addition, element continuity conditions must be satisfied by matching displacements and slopes of adjoining elements.

After the buckling eigenvalue is found, BUCKY reports it to the output files and writes a PATRAN file for the buckled mode shape. The buckling eigenvalue corresponds to the applied stresses. For example, if the applied stresses on the body are described by the set  $\sigma_s$ , and the buckling factor is  $\lambda$ , then it takes a state of stress  $\lambda\sigma_s$  to buckle the structure. If  $\lambda$  is less than or equal to one, then the plate has buckled, and  $\sigma_s$  should be reduced or the plate redesigned to prohibit buckling. If  $\lambda$  is greater than unity, then the plate has not buckled and its design can be optimized.

### 11.11 Plate Bending Implementation

BUCKY can solve the plate bending problem quickly and with very few elements. The plate can be subjected to a quadratically varying pressure force of the form

$$p = C_1 + C_2x + C_3y + C_4x^2 + C_5y^2 + C_6xy \quad (x, y) \in \Omega^e. \quad (34)$$

In addition, the plate can have quadratically varying edge moments in the form of (34). These moments are pure bending moments along the edge or a distributed twisting moment. Portions of the plate may be loaded while other plate elements are unloaded, simulating a real structure. The boundary conditions associated with the plate bending problem are those of the plate buckling solution. In addition, BUCKY allows for the same isotropic or orthotropic material properties as with the buckling problem.

Once BUCKY solves the given problem, the user can view the output graphically or in columnar format. BUCKY outputs the out-of-plane displacements of the plate due to the pressure loads, and it also computes the stresses in the plate elements. However, because the stresses are assumed to vary linearly through the element thickness, as seen with (13), we can question the strain calculations. For convenience, BUCKY computes the strain and stresses at the extreme outer fibers. This gives the highest stress value in the plate cross section.

### 11.12 Plastic Analysis

As we progress through the solution of the plane stress problem, it becomes apparent that a single pass is not sufficient to completely describe the plastic state of a structure. Thus, we conclude that plasticity solutions using the  $p$ -finite element method (or the  $h$ -method for that matter) require iterative procedures. During each iteration, we must compute the stresses and determine if they violate the yield function. If there are region in which the stresses violate the yield criteria, then actions must be taken to quantify the effects of plasticity. In this section, we develop the algorithm used to solve the plasticity problem in general, regardless of method, and to present the associated yield function.

In finite element analysis, there are several widely accepted methods for plastic analysis. One, the initial stiffness method, uses the same stiffness matrix for the structure at all times. By determining the response of the plate to a load, the stresses produced by the loads can be transformed into prestresses. These prestresses, in turn, give the structure certain plastic-like characteristics. The method used in this analysis, however, is the tangent stiffness method, a much more intuitive approach. As the loads are applied, a region of the structure may yield. If the material properties, such as the strength, change in those regions, we can describe plastic behavior. For metals, a stress-strain curve shows that, as the metal goes plastic, large strain increases result for smaller load increments. We

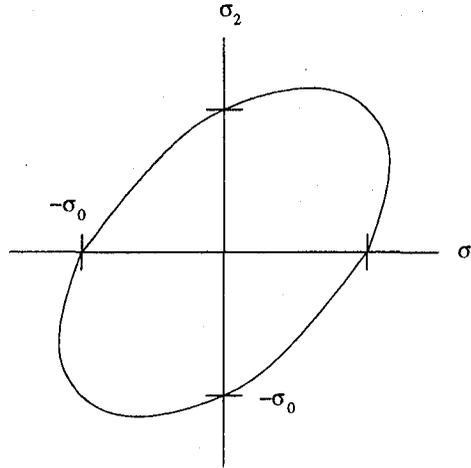


Figure 13: von Mises yield criterion.

can quantify this in terms of a tangent modulus of elasticity, which is significantly less than the Young's modulus for the material. The new tangent modulus is used where the material is plastic.

For a yield function, perhaps the easiest to implement is the von Mises yield function, shown in Figure 13. It is much easier to calculate the yield function on the von Mises ellipse rather than determining where on the yield function we lie, such as in the case of the von Mises condition. Thus, for this analysis, we use the von Mises yield function, given by

$$f_m = J_2 - k^2 \quad (35)$$

where  $k$  is the yield limit in simple shear. At yield, the yield function  $f_m$  is identically zero. For plane stress,  $J_2$ , in terms of principal stresses, is given by

$$J_2 = (\sigma_1^2 - \sigma_1\sigma_2 + \sigma_2^2)/3. \quad (36)$$

Thus, the calculation of  $J_2$  is a simple task for the chosen yield function.

Next, we write the algorithm used in the tangent stiffness method. This involves recalculation of the element stiffness matrix with each iteration. While this is computationally intensive, it is a necessity of the method.

1. For the first cycle, assume  $E$  at the current integration point to take the value of Young's modulus. Form  $U$  and  $T$  for the full set of loads  $\{F\}$ .
2. Compute the displacements, strains, and stresses at each quadrature point by solving the linear problem:

$$[K]\{\Delta d\}_i = \{F\}$$

3. Compute  $\{d\}_i = \{d\}_{i-1} + \{\Delta d\}_i$  for each integration point. Compute  $\epsilon_i = \epsilon_{i-1} + \Delta\epsilon_i$  and  $\sigma_i = \sigma_{i-1} + \Delta\sigma_i$ . For the first cycle, the  $(i-1)$  terms are zero. If on the first cycle, scale the results to bring the structure to initial yield. Set the remaining load increment to  $\Delta F = (F - F_{yield})/nsteps$ .
4. For the quadrature points that have gone plastic, set the modulus of elasticity  $E$  at those points to  $E = E_T$ . Recompute the stiffness matrix,  $[K]_{new}$  of the element.
5. Next solve the problem  $[K]_{new}\{\Delta d\}_i = \{\Delta F\}$  and go to step 3.
6. Stop when  $F_{yield} + \sum \Delta F = F$

The most expensive steps computationally are 1 and 4, where the original and new stiffness matrices are computed. Next, the actual linear solution takes place with less effort. The search for yield violations is relatively inexpensive. It should be noted that there should be a sufficient number of steps to adequately discretize the load increments. Ideally, an infinite number of steps would provide the exact solution. If too few steps are taken, then the relationship between the force and displacement deviates from the exact solution, as shown in Figure 3. Notice that a greater force is required to achieve a displacement than for the exact case.

### 11.13 Axisymmetric Analysis Implementation

Through the use of axisymmetry, three-dimensional stress problems can be converted into a two-dimensional analysis. BUCKY has the capability to model three-dimensional axisymmetric structures in the cylindrical coordinate system. The formulation of the problem is similar to that of the plane stress analysis, except that the field variable  $x$  in the two-dimensional plane stress implementation is replaced by the variable  $r$  in the cylindrical coordinate system. Likewise, the variable  $z$  replaces  $y$ .

One difficulty with the implementation of axisymmetric analysis is the careful maintenance of the problem geometry. Because BUCKY can calculate displacements, strains, and stresses anywhere in a structure, geometry resting on the centerline ( $r = 0$ ) should be avoided. It is well established in linear elasticity that at  $r = 0$ , a singularity arises in the stress conditions. One way to circumvent this problem is to offset geometry only slightly, creating a slight gap in the problem. However, there is logic built into BUCKY such that if a zero radius is found, a point inside the element is used that is close to the desired point. This yields finite values of stresses.

### 11.14 Conclusions

The main emphasis of this paper has been to introduce the  $p$ -version finite element method to the field of thin plate problems. While the two-dimensional theory for plane stress is well integrated into commercial programs, little or no work has been done to extend the  $p$ -version finite element method to thin plate bending and buckling.

In the context of thin plate buckling, currently the only recourse for finding the buckling loads of regularly shaped plates has been through tabulated data and  $h$ -finite element models. For irregularly shaped plates, the  $h$ -finite element method has been used with some success. BUCKY provides an alternative to this previous method. The results are exceptional for even a low-fidelity mesh. Likewise, the exceptional accuracy of  $p$ -methods has proven valuable to plate bending problems. The flexibility of the  $p$ -method allows for problems to be modeled on a continuum type domain, with intricate loading situations.

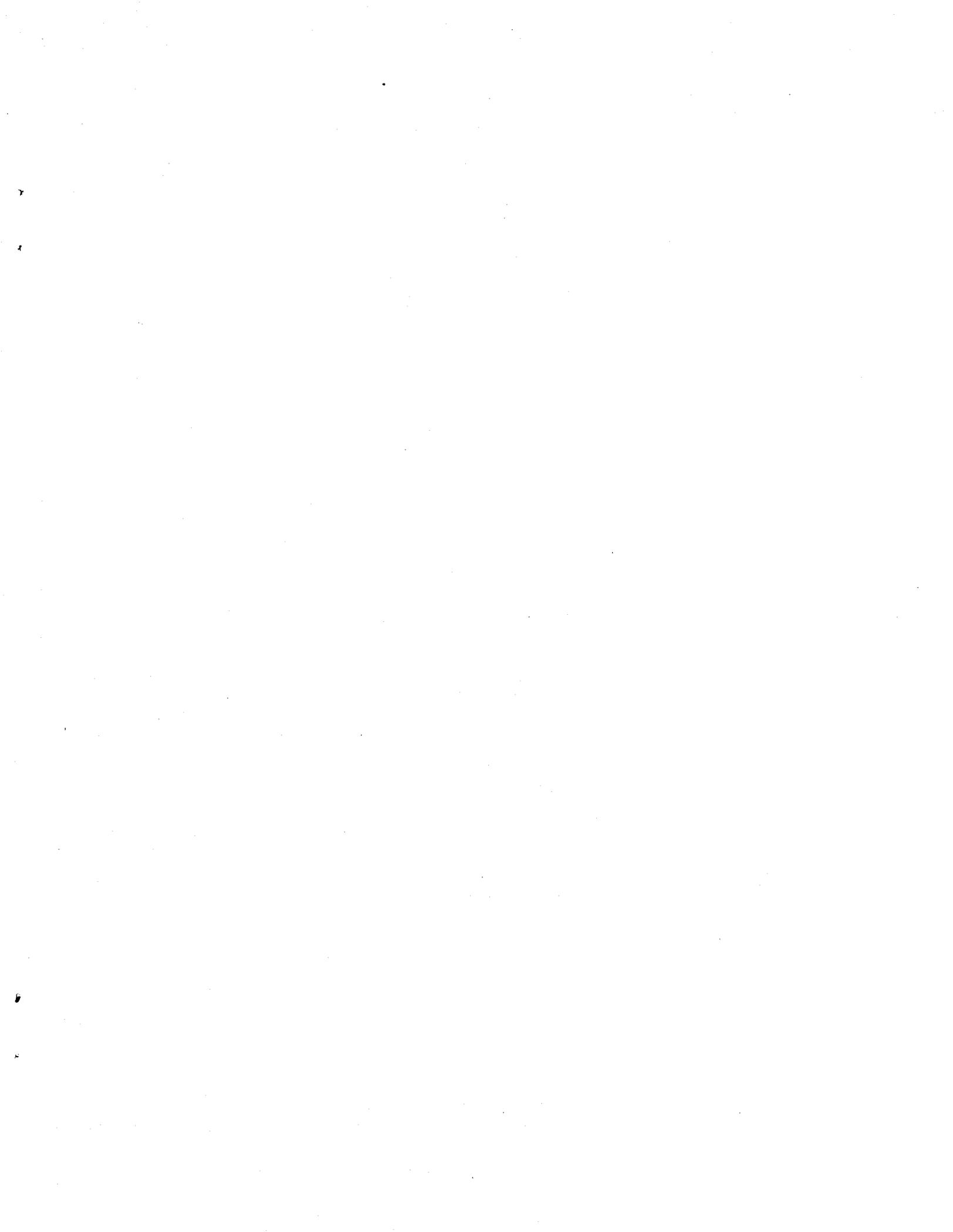
The previous section demonstrates the low error associated with the  $p$ -version finite element method when applied to plate buckling. The same can be said about thin plate bending, although no numerical examples were presented. With a  $p$ -level of 8, the small number of equations needing to be solved yields a highly accurate solution for the buckling load and mode shape. Indeed, for the few known solutions to the thin plate buckling problem, BUCKY and the theoretical solutions correlate very well. The plate buckling problem is more expensive to solve than the linear plane stress or plate bending problem, so a reduced mesh size is optimal when doing a buckling analysis; if we reduce the number of elements in the model, we reduce the costs associated with computational effort. While the plane stress problem involves two displacements ( $u$  and  $v$ ), it is a simple linear problem not requiring expensive data manipulation. However, the buckling analysis is an eigenvalue problem, which is computationally more intensive and, therefore, expensive. Thus, we want to be able to quickly and accurately predict the buckling load of the plate. The  $h$ -finite element method requires a well-defined mesh to model the mode shapes of the plate. The size of the problem becomes prohibitively more expensive to solve. If we can instead use only a few elements, then the number of equations to solve decreases. We can increase the accuracy by simply increasing the polynomial level of the trial solution. While the solution time increases, it does not increase dramatically if the mesh is held fixed.

The quick solution time and the exceptional accuracy of the  $p$ -version of the finite element method makes BUCKY and the  $p$ -finite element method an invaluable tool for the structural engineer. When applied to small plate structures, BUCKY provides a quick determination of the response of a plate subject to a variety of loads. BUCKY's strongpoint, as traditional  $h$ -finite element codes, is its ability to predict the response of irregularly shaped plates in plane stress, plate bending, and plate buckling.

## References

- [1] *PATRAN Plus Users Manual*, PDA Engineering, Costa Mesa, California, 1988.
- [2] Burnett, David S., *Finite Element Analysis*, Addison-Wesley, Reading, Mass., 1987.
- [3] Column Research Committee of Japan, *Handbook of Structural Stability*, Corona, Tokyo, 1971.
- [4] Cook, Robert D., *Concepts and Applications of Finite Element Analysis*, John Wiley & Sons, New York, 1981.

- [5] Eisley, Joe G., *Mechanics of Elastic Structures*, Prentice Hall, Englewood Cliffs, New Jersey, 1989.
- [6] Jones, Robert M., *Mechanics of Composite Materials*, McGraw-Hill, New York, 1975.
- [7] Mar, James W., and Lagace, Paul A., *Advanced Composites*, Massachusetts Institute of Technology, Cambridge, Massachusetts, 1989.
- [8] McFarland, D., et al, *Analysis of Plates*, Spartan Books, New York, 1972.
- [9] Peterson, R. E., *Stress Concentration Factors*, John Wiley & Sons, New York, 1974.
- [10] Roark, Raymond J., and Young, Warren C., *Formulas for Stress and Strain*, McGraw-Hill Book Co., Inc., New York, 1982.
- [11] Smith, James P., "Stability Analysis of Plates Using Ritz Techniques and Lagrange Constraints," JSC-25240, July 1991.
- [12] Smith, James P., "General Plate Stability Using High Order Techniques," AIAA 92-2283, presented at the 33<sup>rd</sup> AIAA SDM Conference, Dallas, Texas, April 1992.
- [13] Smith, James P., "COMPPAP Instruction Manual," JSC-25938, July 1992.
- [14] Smith, James P., "On Solving the Irregularly-Shaped Plate Buckling Problem," AIAA 93-0014, presented at the 31<sup>st</sup> Aerospace Sciences Meeting, Reno, Nevada, January 1993.
- [15] Stein, M., "Nonlinear Theory of Plates and Shells Including the Effects of Transverse Shearing," *AIAA Journal*, Vol. 24, No. 9, Sept. 1986, pp. 1537-1544.
- [16] Timoshenko, S., and Gere, J., *Theory of Elastic Stability*, McGraw-Hill Book Co., Inc., New York, 1961.
- [17] Timoshenko, S., and Woinowsky-Kreiger, S., *Theory of Plates and Shells*, McGraw-Hill Book Co., Inc., New York, 1959.
- [18] Timoshenko, S., and Goodier, J., *Theory of Elasticity*, McGraw-Hill Book Co., Inc., New York, 1970.
- [19] Wilkinson, J. H., *The Algebraic Eigenvalue Problem*, Clarendon Press, Oxford, 1965.
- [20] Zienkiewicz, O. C., *The Finite Element Method*, McGraw-Hill Book Co., Inc., New York, 1977.

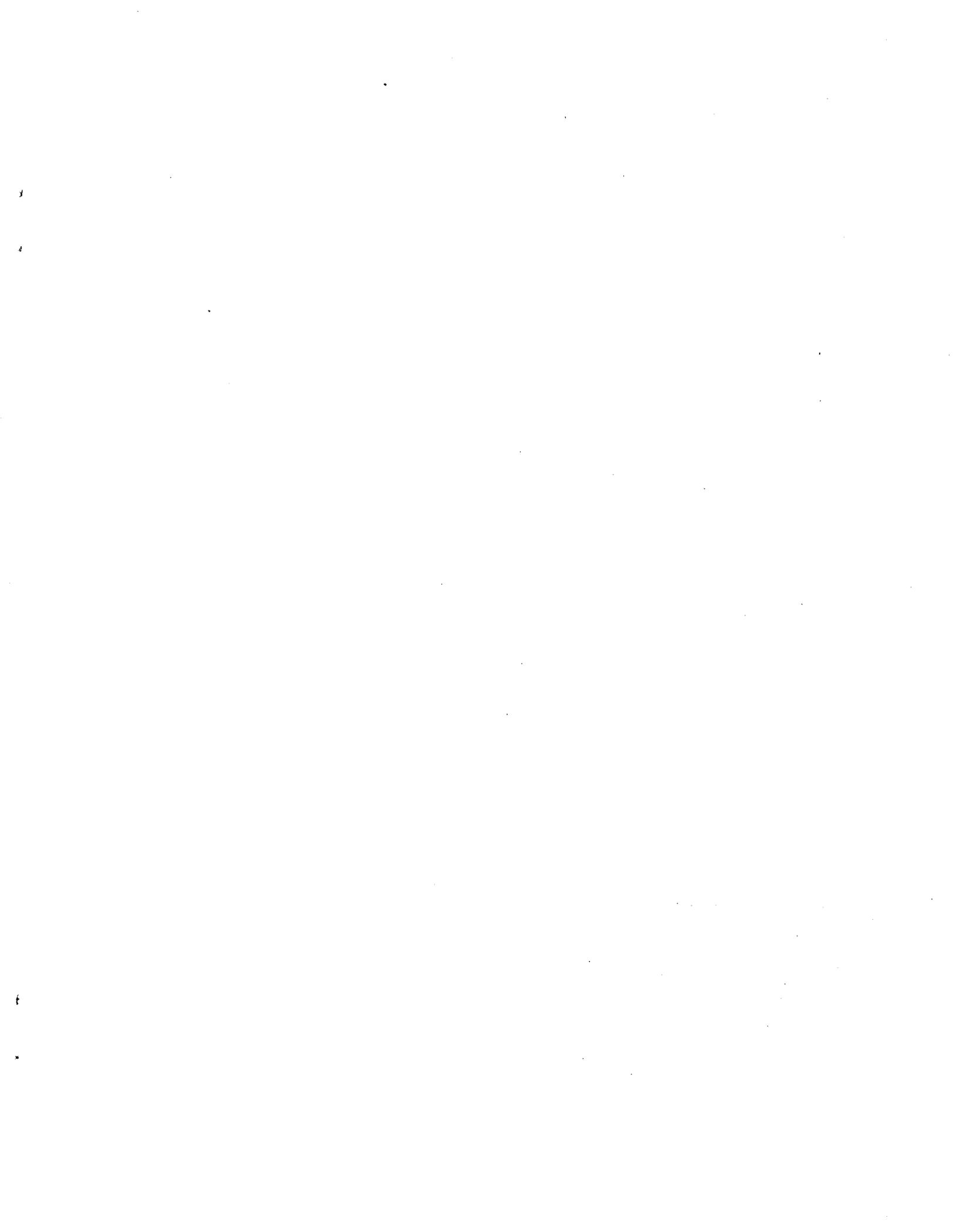


# REPORT DOCUMENTATION PAGE

*Form Approved*  
**OMB No. 0704-0188**

Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.

1. AGENCY USE ONLY ( <i>Leave blank</i> )	2. REPORT DATE <b>May 1994</b>	3. REPORT TYPE AND DATES COVERED <b>Technical Memorandum (TM)</b>	
4. TITLE AND SUBTITLE  <b>BUCKY Instruction Manual version 3.3</b>		5. FUNDING NUMBERS	
6. AUTHOR(S)  <b>James P. Smith</b>		8. PERFORMING ORGANIZATION REPORT NUMBER  <b>S-766</b>	
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)  <b>NASA Johnson Space Center Houston, TX 77058</b>		10. SPONSORING / MONITORING AGENCY REPORT NUMBER  <b>NASA TM-104793</b>	
9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES)  <b>National Aeronautics and Space Administration Washington, D.C. 20546</b>		11. SUPPLEMENTARY NOTES	
12a. DISTRIBUTION / AVAILABILITY STATEMENT <b>National Technical Information Service 5285 Port Royal Road Springfield, VA 22161 (703) 487-4600</b>		12b. DISTRIBUTION CODE  <b>Subject Category: 61</b>	
13. ABSTRACT ( <i>Maximum 200 words</i> )  <b>The computer program BUCKY is a p-version finite element package for the solution of structural problems. The current version of BUCKY solves the 2-D plane stress, 3-D plane stress plasticity, 3-D axisymmetric, Mindlin and Kirchoff plate bending, and buckling problems. The p-version of the finite element method is a highly accurate version of the traditional finite element method. Example cases are presented to show the accuracy and application of BUCKY.</b>			
14. SUBJECT TERMS  <b>Finite Element Method, Computer Programs</b>		15. NUMBER OF PAGES <b>62</b>	16. PRICE CODE
17. SECURITY CLASSIFICATION OF REPORT <b>Unclassified</b>	18. SECURITY CLASSIFICATION OF THIS PAGE <b>Unclassified</b>	19. SECURITY CLASSIFICATION OF ABSTRACT <b>Unclassified</b>	20. LIMITATION OF ABSTRACT <b>UL</b>



NASA Technical Library



3 1176 01408 7317