Two-Dimensional Computer Modeling of Joints and Fractures in Continua

Royd R. Nelson

Brigham Young University - Provo

Follow this and additional works at: https://scholarsarchive.byu.edu/etd

Part of the Civil and Environmental Engineering Commons

BYU ScholarsArchive Citation

https://scholarsarchive.byu.edu/etd/3462

This Thesis is brought to you for free and open access by BYU ScholarsArchive. It has been accepted for inclusion in All Theses and Dissertations by an authorized administrator of BYU ScholarsArchive. For more information, please contact scholarsarchive@byu.edu, ellen_amatangelo@byu.edu.
TWO-DIMENSIONAL COMPUTER MODELING OF JOINTS AND FRACTURES IN CONTINUA

A Thesis
Presented to the
Department Of Civil Engineering
Brigham Young University

In Partial Fulfillment
of the Requirements for the Degree
Master of Science

by
Royd R Nelson
December 1983
This thesis, by Royd R. Nelson, is accepted in its present form by the department of Civil Engineering of Brigham Young University as satisfying the thesis requirement for the degree of Master of Science.

Steven E. Benzley
Committee Chairman

S. Olani Durrant
Committee Member

7 December 83
Date

Henry N. Christiansen
Department Chairman
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>ACKNOWLEDGEMENTS</td>
<td>iv</td>
</tr>
<tr>
<td>LIST OF FIGURES</td>
<td>v</td>
</tr>
<tr>
<td><strong>CHAPTER</strong></td>
<td></td>
</tr>
<tr>
<td>1. INTRODUCTION</td>
<td>1</td>
</tr>
<tr>
<td>2. LOW SHEAR ELEMENT SLIP PLANES</td>
<td>4</td>
</tr>
<tr>
<td>3. JOINTED MEDIA MODEL</td>
<td>25</td>
</tr>
<tr>
<td>4. MINE CAVITY ROOF DEFLECTION ANALYSIS</td>
<td>43</td>
</tr>
<tr>
<td>5. SUMMARY AND RECOMMENDATIONS</td>
<td>55</td>
</tr>
<tr>
<td><strong>BIBLIOGRAPHY</strong></td>
<td>57</td>
</tr>
<tr>
<td><strong>APPENDIXES</strong></td>
<td></td>
</tr>
<tr>
<td>A. PRESCRUBS USER'S MANUAL</td>
<td>60</td>
</tr>
<tr>
<td>B. SCRUBS USER'S</td>
<td>89</td>
</tr>
<tr>
<td>C. SLIP PLANE ADDITION CODE LISTINGS</td>
<td>106</td>
</tr>
<tr>
<td>D. JOINTED MEDIA CODE LISTING</td>
<td>113</td>
</tr>
</tbody>
</table>
ACKNOWLEDGEMENTS

The author wishes to express his appreciation to Dr. Steven E. Benzley and Dr. S. Olani Durrant for their many hours of patient assistance in preparation of this thesis. Special thanks goes to Dr. Benzley for providing this research opportunity and for his continued help throughout the course of this work.

Thanks is also extended to the Brigham Young University faculty and to many friends and family members who have provided support during the preparation of this work.

Finally, the author wishes to express his love and appreciation to his devoted wife, Kathy, for her patience, understanding, and love.
LIST OF FIGURES

Figure
1. Low shear slip plane. .................. 4
2. Location of added slip plane row. .......... 9
3. Effect of slip plane addition on nodal numbering 12
4. Effect on nodal numbering of 8-noded elements . 15
5. Shifting plates example .................. 18
6. Deformed shape of shifting plates example ... 18
7. Effect of varying G of slip plane on deflection 19
8. Beam example. ......................... 20
9. Finite element mesh for beam example. .... 21
10. Deformed meshes for various numbers of slip planes. .............. 22
11. Representative elementary volume containing regularly spaced fractures. ........ 26
12. Assumed joint stiffness behaviour in shear. . . . 32
13. Loading and deformed shape of simple shear example .............. 38
14. Effect of varying Gs of jointed material on deflection. ............... 39
15. Loading and deformed shape of beam example. . . 41
16. Experimental centrifuge model .................. 45
17. Mine cavity roof deflection analysis mesh ... 47
18. Surface profiles above mined region ........ 50
19. Mine cavity roof deflection profiles .......... 50
20. Comparison of Sandia code JAC and SCRUBS ... 53
CHAPTER 1

INTRODUCTION

Accurate prediction of the continuum response of geologic materials has traditionally been a challenging task. Being out of view, the widely varying strata of soils makes accurate geometric modeling difficult, and the difficulty of obtaining undisturbed test samples, along with the commonly nonhomogeneous nature of soils, presents problems in accurately determining the soil's material parameters. Another problem often encountered is the presence of cracks and fissures in geologic materials. Fault lines which occur can also alter the continuum response, and the joints and cracks present in such materials reduced their strength considerably. Although the difficulty in predicting the geometric and soil properties will always be a problem, determining the location of joints and cracks, and the incorporation of their effects into the mathematical model, can help lead to better and more accurate prediction of soil and rock mechanics problems.
The purpose of this thesis is to implement into a two-dimensional finite element code the capability of modeling joints and cracks in materials. This work is a continuation of work done by Benzley [1] to improve computer modeling of subsidence prediction. The capabilities added in this work, however, can be used for any fractured or jointed material problem, including nongeologic applications.

Two different methods of modeling cracks will be investigated. The first method involves adding a slip plane along specified boundaries to model the joints individually. The details of how this method was added to the code, along with the effects of these slip planes on examples, will be discussed in the following chapter.

The second method incorporates a continuum description for jointed media modified from work by Thomas. [2] This description introduces a new material model in which the material is idealized as a regularly fractured media having both elastic and plastic slip at the joints. The details of this method will be outlined in Chapter 3.

Chapter 4 will demonstrate the capabilities of these methods by comparing their effects on a mine cavity roof deflection problem. A second comparison of the slip plane capability will be made with the
prediction of another code with a slip plane capability on a similar problem. Finally, the conclusions and recommendations reached in this thesis will be discussed in Chapter 5.

The finite element code updated to include this model is a two-dimensional, elastic-plastic code called SCRUBS. The basic finite element equations and procedures used in developing this code can be found in References 3 and 4. A user's manual for SCRUBS which includes the capabilities added in this work is located in Appendix B. An interactive preprocessor called PRESCRUBS, developed by Long [5], is available for setting up the SCRUBS data file. An updated PRESCRUBS manual is also included in and can be found in Appendix A.
The modeling of joints and fractures has been incorporated into many finite element codes by the use of sliding interfaces or slip planes. The implementation of these interfaces into the codes can involve somewhat complex formulations and often increase the computing time of these programs considerably. A slip plane formulation was added to the SCRUBS code as part of this thesis. A "low shear element" slip plane was developed for this purpose, and was chosen because of its ease in implementation into the existing code, and its limited effect on the run time of the program. To use this method, the user simply adds a very thin row of elements into his model wherever a slip plane is desired, as shown in Figure 1.

Figure 1. Low Shear Slip Plane.
This row of elements is then defined as a low shear material as will be described in the following section. This formulation simply imposes regions of minimal shear resistance along the slip plane element row.

Low Shear Formulation

"In standard plane strain computer programs which employ a generalized form of Hooke's law, the relationship between stresses and strains is expressed as

\[
\begin{bmatrix}
\sigma_{xx} \\
\sigma_{yy} \\
\sigma_{xy}
\end{bmatrix}
= \frac{E}{(1+\nu)(1-2\nu)}
\begin{bmatrix}
(1-\nu) & \nu & 0 \\
\nu & (1-\nu) & 0 \\
0 & 0 & (1-2\nu)/2
\end{bmatrix}
\begin{bmatrix}
\varepsilon_{xx} \\
\varepsilon_{yy} \\
\gamma_{xy}
\end{bmatrix}
\] (2.1)

Problems obviously occur in this formulation when Poisson's ratio becomes .5. Another disadvantage to this formulation is that, for a material that has very little shear strength, as G approaches 0, E approaches 0 and the equations for stress and strain break down. In a manner similar to Clough and Woodward [6] we can circumvent the above problems by writing the stress strain relations in terms of the bulk modulus, K, and the shear modulus, G. Doing this,
\[
\begin{pmatrix}
\sigma_{xx} \\
\sigma_{yy} \\
\sigma_{xy}
\end{pmatrix} =
\begin{bmatrix}
K + 4/3 G & K - 2/3 G & 0 \\
K - 2/3 G & K + 4/3 G & 0 \\
0 & 0 & G
\end{bmatrix}
\begin{pmatrix}
\varepsilon_{xx} \\
\varepsilon_{yy} \\
\gamma_{xy}
\end{pmatrix}
\]

where \( K = E/3(1-2v) \)
\( G = E/2(1+v) \)

The above equation allows \( G \) to approach 0 and still maintain a valid stress strain relationship." [1] This formulation was incorporated into the SCRUBS program by simply allowing for the entering of the bulk modulus, \( K \), as a negative number, in place of Young's modulus, and entering the shear modulus \( G \) in place of Poisson's ratio. The negative sign on the bulk modulus is the flag which causes the elasticity matrix to be created using equation 2.2 rather than equation 2.1. To model a material as a low shear material, the shear modulus is simply entered as a number very low relative to the bulk modulus. This model is not limited to use in low shear materials, but can be utilized any time the material is described by \( K \) and \( G \) rather than \( E \) and \( \nu \).

PRESRUBS was also modified to allow for low shear slip plane. A new solution control parameter is required which indicates whether or not any slip planes exist. If so, the user is automatically prompted for the bulk and shear moduli for the slip plane material.
It should be noted that this formulation requires a separate boundary flag for each discrete slip plane.

Slip Plane Element Row Addition

The geometric modeling for SCRUBS is done on an interactive mesh generator called QMESH. This program accepts the perimeter data for specified regions of the geometry, and automatically creates and numbers the finite element mesh. Boundary conditions are imposed by assigning flags to desired lines or line segments which can later (in PRESCRUBS) be coupled to specified displacement or force criteria. In order to define low shear element rows as described earlier, the user would have to enter perimeter data for each row. A new capability was therefore added to the SCRUBS program which automatically creates this thin row of elements. With this new upgrade, any line can be defined as a slip plane by simply assigning it a boundary flag in QMESH, thus saving considerable preprocessing time. (Readers not familiar with the use of QMESH should consult Reference 7.)
Code Modifications

Both the PRESCRUBS and SCRUBS programs required modifications in order to incorporate the automatic slip line generation capability.

PRESCRUBS was modified to include a new boundary code defining a slip plane boundary. The program then determines all nodes on the boundary information received from QMESH. The user must also input the thickness desired for the element row to be created, and a flag which indicates the general angle of the slip plane. Detailed instructions for user input can be found in the PRESCRUBS user's manual located in Appendix A.

In order to include this slip row addition, it was necessary to add a new set of subroutines to the SCRUBS program. The major subroutine added is called SLIPP. The SLIPP subroutine also makes use of three supplementary routines, CORNN, MAKEEL, and NEWNOD.

Many problems were encountered in developing this capability, and an outline of the development of these routines is worthwhile. The data read in by the SLIPP routine is as follows:

1. The number of slip planes
2. Thickness for each slip plane
3. The angle flag for each slip plane
The angle flag indicates the angle of the slip plane and is 1 for angles greater than $45^\circ$, and 2 for angles less than $45^\circ$.

Determining the position of the new row of slip elements is relatively easy. A new node is created for every node in the slip plane node list by adding the thickness of the slip plane to the X or Y coordinate of the slip plane node. The thickness is added to the X coordinate if the slip plane angle is greater than $45^\circ$, and it is added to the Y coordinate if the angle is less than $45^\circ$. These new coordinates form a line parallel to the specified slip plane thickness. Figure 2 illustrates where the new line would be added for two different slip plane angles.

![Figure 2. Location of Added slip plane row.](image-url)
The main problem in the development lies in the numbering of the new nodes created. The maximum difference in node numbers for any element is directly related to the bandwidth of the stiffness matrix. The higher the bandwidth, the more storage space there is required to store the total stiffness matrix. Too high of a bandwidth causes the stiffness matrix dimensioning of the program to be exceeded.

The initial numbering of the mesh is done in QMESH. This program has many different numbering schemes provided to help optimize the bandwidth of the initial problem. Since there is no way of knowing which of these numbering schemes was used, it is impossible to duplicate the QMESH numbering once inside the SCRUBS program. The slip plane addition therefore required either a complete renumbering as done in QMESH, or an algorithm which kept the bandwidth reasonable regardless of the initial numbering scheme. In order to minimize the addition of code to SCRUBS, the latter of these two options was selected.

The algorithm used to number the new nodes is relatively simple. A new node is created for the first node in the slip plane node list, and is assigned a node number one greater than its corresponding slip node number. The node numbers in the connectivity array, the
coordinate arrays, the boundary condition arrays, and the slip plane node list array all must then be updated by adding one to all node numbers contained in these arrays which are greater than the slip plane node being considered. This process is then repeated until a new node has been added for each node in the slip plane node list.

The coordinate of each new node has the same coordinate as its corresponding slip node, except with the specified thickness added to the X or Y coordinate as described earlier. Also, if any slip plane node has a specified boundary condition, the new node created is assigned the same boundary condition. Figure 3 shows a simple mesh for 3 different numbering schemes including the numbering before and after the slip plane row addition.
Figure 3. Effect of slip plane addition on nodal numbering.
As seen in Figure 3, this numbering algorithm does provide a new numbering with a reasonably small maximum node number difference for all three numbering schemes shown; however, it is also apparent that the increase in node difference is not independent of the original numbering sequence. For sequence 1 the maximum node difference increased from 7 to 12, for sequence 2 from 11 to 12 and for sequence 3 from 6 to 7. This indicates that the slip plane line addition algorithm works best when the numbering is done perpendicular to the slip plane rather than parallel to it. This should be kept in mind when modeling large problems where bandwidth limitation is important.

After the nodal numbering is complete, the new row of slip elements must also be included in the connectivity array. This is accomplished in subroutine MAKEEL by creating a new element for each adjoining pair of slip plane nodes. These new elements are numbered by just adding new elements at the end of the connectivity array; the new element numbers would then be the numbers immediately succeeding the former maximum element number.

Subroutine CORNN is included to allow renumbering of higher order elements, and simply determines which slip plane nodes lie at the corners of an element. This
information is required by both SLIPP and MAKEEL to provide proper renumbering. For example, with 8-noded elements, subroutine SLIPP adds 2 nodes for each corner node, and only one for mid-side nodes. MAKEEL also adds one new element for each set of consecutive corner nodes along the slip plane, rather than every adjoining nodal pair. Figure 4 shows the renumbering of a simple 8-noded finite element mesh. This figure illustrates that the problem of increased bandwidth is compounded when using higher order elements. The use of this method with these elements in problems with large numbers of elements can be impractical.
Figure 4. Effect on nodal numbering of 8-noded elements.
Subroutine NEWNOD is required due to the change in nodal numbering. The function of this routine is simply to determine the updated node number of any given node. This is done by storing the original coordinate array, and comparing the coordinate of an altered node with the original coordinate array to determine its original node number. This routine is used in several places in SLIPP to update node numbers after the addition of new nodes. NEWNOD is also accessed by other routines in the original SCRUBS program in order to update node numbers of load application points, singular points, summed reaction nodes, and boundary impact nodes.

It is important to keep in mind when inputting node numbers for these points in PRESCRUBS, that they must correspond to node numbers in the original mesh created by QMESH. The use of NEWNOD updates these numbers to their new node numbers so that the loads, singular points, etc., are applied in their proper positions.

The SLIPP program also uses NEWNOD to create a nodal mapping of old to new node numbers. This mapping is provided in the SCRUBS.LIS output file along with a listing of the new elements added in the connectivity. The mapping was included to help the user in comparing an altered SCRUBS output mesh with the original mesh created by QMESH.
A code listing of each these added subroutines is included in Appendix C.

EXAMPLES

Two example problems will be presented which demonstrate the use of this new addition. The following sections will describe each of these examples in detail.

Shifting Plates Example

The first example is a model of two plates sliding past one another. Each plate is 2 inches thick and 5 inches long. One plate is lying flat on a surface and is fixed in the X and Y directions, and the second plate lies directly above it. To demonstrate the capability of the low shear slip plane, the boundary between the plates is modeled as a slip plane, and a horizontal load of 10 lbs. is applied. An illustration of the modeling of this example is shown in Figure 5.
The plate material is modeled as steel and the "slip" material is given a bulk modulus of $10^7$ psi and a shear modulus of 400 psi. As can be seen in Figure 6, the deformations along the line of slip are very high compared to the deformation in the plate material. This was the effect desired for the slip plane addition.

The problem was run several times using varying values for the shear modulus $G$. Figure 7 shows the displacement of the top plate as a function of $G$, and clearly shows the linear relationship of this
displacement to the shear modulus.

Figure 7. Effect of varying G of slip plane on deflection.

It should be noted that the resistance to the load must be transferred totally through the slip plane. This method therefore doesn't describe a true slip because the slip plane is actually an elastic material and it does transfer load; however, the model does adequately predict a boundary layer weakening along the slip, plane which was the purpose of the development of this model.
Beam Example

The second example considered is that of a simply supported beam loaded with a concentrated load at midspan and having a varying number of horizontal slip planes as shown in Figure 8.

![Beam example diagram](image)

Figure 8. Beam example.

The beam modeled was 4 inches deep, 10 inches long and 1 inch thick. The material was modeled as steel ($E=3\times10^7$ psi, $v = .3$) and the downward load was 20 lbs. Slip planes ($K=100000$ psi, $G = 40$ psi) were placed every inch through the thickness. The symmetrical nature of the problem made possible the use of only half the model to create the mesh. The mesh used and the boundary conditions imposed are shown in Figure 9.
Figure 9. Finite element mesh for beam example.

This problem was analyzed 4 times, beginning with no slip planes and adding one slip plane in each succeeding analysis. Figure 10 illustrates the exaggerated deflected shapes of the four cases analyzed. TABLE 1 shows the deflection at midspan for each of the four cases, and demonstrates well the weakening effect of the addition of slip planes.

<table>
<thead>
<tr>
<th>NUMBER OF SLIP PLANES</th>
<th>DEFLECTION AT MIDSPAN</th>
<th>SCRUBS</th>
<th>HAND METHOD</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>.279 X 10^-4</td>
<td>.313 X 10^-4</td>
</tr>
<tr>
<td>1</td>
<td></td>
<td>.105 X 10^-3</td>
<td>.125 X 10^-4</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>.228 X 10^-3</td>
<td>.200 X 10^-3</td>
</tr>
<tr>
<td>3</td>
<td></td>
<td>.468 X 10^-3</td>
<td>.500 X 10^-3</td>
</tr>
</tbody>
</table>
This weakening in bending is due to the inability of the slip plane to transfer shear, making the problem behave as though there were separate beams between each slip plane. For comparison, the prediction of the midspan deflection using the classical approach was also included in TABLE 1, with the deflection being predicted by the following equation:

\[ \delta = \frac{PL^3}{48EI} \]  

(2.3)
All parameters in this equation remain constant, except the moment of inertia $I$, which is computed as the sum of the moments of inertia of individual beams bounded by slip planes. For example, in case 4 it would be computed as 4 times the moment of inertia of one beam 1 inch high and 1 inch thick, whereas with no slip planes $I$ is simply computed for one beam 1 inch thick and 4 inches high. The comparison of these results shows that the model is quite accurate and that the slip plane model used acts very much like a region of zero shear.

SUMMARY

The use of this low shear material does not accurately model a true slip plane because it does not allow any slip, but rather only elastic deformations. This formulation also does not allow free opening of the crack or joint which may be inaccurate in some applications. However, for mainly compressive problems with relatively small deformations, the method adequately describes the weakening of the jointed material.
Although at present, the use of this low shear element capability is currently limited to elastic deformations along the plane of slip, the incorporation of a plastic failure model for the slip plane material is possible. The Drucker-Prager [8] yield criteria would be especially applicable for this purpose and would be recommended for any further study in this area.
A second method of modeling the effects of cracks and fractures in geologic materials is to develop a new material model for the jointed material based on continuum theory. Using this method, joints in the materials are implemented by allowing for the effects of jointing on the material response rather than discrete models for each joint. The continuum description for this material model will be presented in the following sections. This formulation is taken from work done by Thomas [2], but is modified to neglect the joint dilation response, and to include only the two dimensional or axisymmetric continua.

The model is composed of two parts: a continuum approximation in which the joint displacements are average through the material, and a continuum description based on linear behavior of the base material, and nonlinear shear at the joints.
Continuum Approximation

"The continuum model used for this material is based on the published work of Moreland. [9-11] Consider a "representative elementary volume" containing regularly spaced parallel fractures as sketched in Figure 11. The orientation of this joint set is characterized by a unit normal vector \( \mathbf{n} \) with respect to fixed \( x_1, x_2, x_3 \) coordinate axes. The spacing between fractures is denoted by \( \delta \). Additional unit vectors \( \mathbf{s} \) and \( \mathbf{t} \) in the plane of the joints are introduced such that \( \mathbf{s}, \mathbf{t}, \) and \( \mathbf{n} \) form a local cartesian coordinate system.

Figure 11. Representative elementary volume containing regularly spaced fractures.
It is assumed that the relative motion at the interface of the \( r \)th fracture at position \( x_r \) can be measured by a jump "dilitation" vector \( u_r^d(x_r) \) normal to the fracture plane, and a jump slip displacement vector \( u_r^s(x_r) \) parallel to the fracture plane. The netjump displacements for \( R \) fractures in the representative volume will then be

\[
R \bar{u}^d(x) = \sum_{r=1}^{R} u_r^d(x_r)
\]

\[
R \bar{u}^s_r(x) = \sum_{r=1}^{R} u_r^s(x_r)
\]

(3.1)

where \( \bar{u}^d \) and \( \bar{u}^s \) are average displacements and \( x \) is any position in the element. The continuous displacement fields \( u_d \) and \( u_s \) with respect to the \( x_1, x_2, x_3 \) axes are introduced,

\[
u^d(x + d \bar{n}) = u^d(x) + \left( \frac{u^d}{\delta} \right) d \bar{n}
\]

\[
u^s(x + d \bar{n}) = u^s(x) + \left( \frac{u^s}{\delta} \right) d \bar{n}
\]

(3.2)

where

\[
\bar{n}^d = |\bar{n}| \bar{n} = u^d \bar{n}
\]

\[
\bar{n}^s = |\bar{n}| \bar{\nu} = u^s \bar{\nu}
\]

(3.3)

In equation (3.3) the direction of slip displacement is in the direction of the unit vector \( \bar{\nu} \). The total displacements can be written as

\[
u(x) = u^s(x) + u^d(x) + u^*(x)
\]

(3.4)
where \( \mathbf{u}^b \) is the displacement field of the intact material between fractures. From equation (3.4), the strain decomposition is taken to be

\[
\mathbf{\epsilon} = \mathbf{\epsilon}^b + \mathbf{\epsilon}^d + \mathbf{\epsilon}'
\]  

(3.5)

The dilatation and slip strains are defined in terms of the continuous displacement,

\[
2\mathbf{\epsilon}^d = \mathbf{u}^d \nabla + (\mathbf{u}^d \nabla)^T
\]

\[
2\mathbf{\epsilon}' = \mathbf{u}' \nabla + (\mathbf{u}' \nabla)^T
\]  

(3.6)

where \( \nabla \) is the gradient operator with respect to the \( x_1, x_2, x_3 \) axes. Equation (3.6) can be reduced by decomposing \( \mathbf{u}^d \) and \( \mathbf{u}' \) into components in the local coordinate system.

\[
\mathbf{u}^d = |\mathbf{u}^d| \mathbf{n} = \mathbf{u}^d \mathbf{n}
\]

\[
\mathbf{u}' = |\mathbf{u}'| \mathbf{\hat{s}} = \mathbf{u}' \mathbf{\hat{s}}
\]  

(3.7)

Since both \( \mathbf{u}^d \) and \( \mathbf{u}' \) have nonzero gradients only in the \( n \) direction, equation (3.6) becomes

\[
\mathbf{\epsilon}^d = \frac{\partial \mathbf{u}^d}{\partial n} (\mathbf{n} \otimes \mathbf{n})
\]

\[
\mathbf{\epsilon}' = \frac{1}{2} \frac{\partial \mathbf{u}'}{\partial n} (\mathbf{s} \otimes \mathbf{n} + \mathbf{n} \otimes \mathbf{s})
\]  

(3.8)
From equation (3.2)
\[ \frac{\partial u^d}{\partial n} = \frac{\partial d}{\partial n} \]
\[ \frac{\partial u^*}{\partial n} = \frac{\partial u^*}{\partial n} \]
(3.9)

so the final form for the strains is
\[ d^d = \frac{\partial d}{\partial n} (n \otimes n) \]
\[ d^* = \frac{\partial u^*}{\partial n} (s \otimes n + n \otimes s) \]
(3.10)

Constitutive Model

We will first introduce the components of a stress tensor \( T \) and a total strain tensor \( E \) which refer to the local \( g, t, n \) coordinate system. If \( \sigma \) and \( e \) are stress and strain tensors in the \( x_1, x_2, x_3 \) coordinate system, the transformation equations are
\[ T_{nn} = g n \cdot n \]
\[ T_{ns} = g n \cdot s \]
\[ T_{ts} = g t \cdot t \]
\[ T_{tn} = g n \cdot t \]
(3.11)

and
\[ E_{nn} = e n \cdot n \]
\[ E_{ns} = e n \cdot s \]
\[ E_{ts} = e t \cdot t \]
\[ E_{tn} = e n \cdot t \]
(3.12)
Intact Material

In the present formulation, the intact material is assumed to behave as a linear elastic solid, having a strain rate

$$\dot{\varepsilon}^b = \frac{\sigma}{2G} - \left(\frac{K - \frac{2}{3}G}{6KG}\right)(\text{tr} \dot{\varepsilon}) \mathbf{I}.$$  \hspace{1cm} (3.13)

Joint Dilation

The main purpose of using this model is to predict the weakening effects of slip along fractures and joints in materials. For many problems the fracture opening is small relative to the fracture spacing. For these problems, the joint dilation normal to the crack is also relatively small for cracks in compression; therefore the dilation response will be neglected; thus

$$\varepsilon^d = 0$$ \hspace{1cm} (3.14)

The only condition imposed on the joint dilation is that a joint cannot support a tensile load; therefore

$$T_{nn} = 0, \quad \tau^d \geq 0$$ \hspace{1cm} (3.15)
Joint Shear

The joint shear stress-displacement behavior is assumed to be elastic perfectly plastic. In the elastic range

$$\dot{\omega} = \frac{T_{es}}{G_s}$$  \hspace{1cm} (3.16)

where $G_s$ is an elastic modulus of slip determined for a single joint. From equation (3.10) the slip strain rate is

$$\dot{\varepsilon}^s = \frac{T_{es}}{2\delta G_s} (s \otimes n + n \otimes s)$$  \hspace{1cm} (3.17)

The onset of plastic behavior is assumed to be governed by a linear Mohr-Coulomb criterion, based on a scalar "slip function", $F$, defined as

$$F = |T_{ns}| + \mu T_{nn} - C_o$$  \hspace{1cm} (3.18)

where $\mu$ is the coefficient of friction and $C_o$ is the cohesion. The joint behavior is elastic for $F<0$ and plastic for $F>0$. The stress-displacement behavior in shear is shown in Figure 12.
Solution of Constitutive Equations

The SCRUBS finite element program uses an incremental method where the stresses and strains are known at the last stress increment, the current incremental strains are known, and only the current incremental stresses are needed.

From equation (3.5), the elastic strain rate can be defined as

\[ \dot{\varepsilon} = \dot{\varepsilon}^b + \dot{\varepsilon}^d + \dot{\varepsilon}^e \]  (3.19)
Using the constitutive relations in equations (3.13), (3.14), and (3.17), the elastic strain rate [Eq 3.19] becomes

\[
\dot{\varepsilon} = \frac{\dot{\gamma}}{2G} - \left(\frac{K - \frac{2}{3}G}{6KG}\right) (\text{tr } \dot{\gamma}) \mathbf{I} \\
+ \frac{T_{\text{m}}}{2\xi G_s} (\mathbf{s} \otimes \mathbf{n} + \mathbf{n} \otimes \mathbf{s})
\]

(3.20)

Transferring equation (3.20) to scalar equations in terms of the local stress components, and omitting the out of plane shears, we have

\[
\dot{\varepsilon}_{\text{m}} = \frac{T_{\text{m}}}{2G} - \frac{K - \frac{2}{3}G}{6KG} (\text{tr } \dot{\gamma}) \\
\dot{\varepsilon}_{\text{w}} = \left(\frac{1}{2G} + \frac{1}{2\xi G_s}\right) T_{\text{w}} \\
\dot{\varepsilon}_{\text{m}} = \frac{T_{\text{m}}}{2G} - \frac{K - \frac{2}{3}G}{6KG} (\text{tr } \dot{\gamma}) \\
\dot{\varepsilon}_{\text{u}} = \frac{T_{\text{u}}}{2G} - \frac{K - \frac{2}{3}G}{6KG} (\text{tr } \dot{\gamma})
\]

(3.21)
Solving for incremental stresses in terms of incremental strains produces

\[ \dot{T}_{xx} = \left( \frac{2G}{1 + \frac{G}{\delta G}} \right) \dot{\varepsilon}_{xx} \]

\[ \dot{T}_{nn} = 2G\varepsilon_{nn} + (K - \frac{2}{3}G) \text{tr} (\varepsilon) \]

\[ \dot{T}_{ss} = 2G\varepsilon_{ss} + (K - \frac{2}{3}G) \text{tr} (\varepsilon) \]

\[ \dot{T}_{tt} = 2G\varepsilon_{tt} + (K - \frac{2}{3}G) \text{tr} (\varepsilon) \]

(3.22)

Therefore for the elastic case, all components of incremental stress in the local coordinate system can be obtained directly from incremental strains. Next, it must be determined if plastic slip occurs. After the stress components have been obtained the slip function [Eq 3.18] must be tested. If \( P < 0 \), the strain rate was entirely elastic and the stress rates determined in equation (3.22) are correct. However, if \( P > 0 \), plastic slip has occurred.

In the case of plastic slip, determination of the incremental stresses can easily be made by assuming elastic and perfectly plastic behavior such that

\[ \dot{\varepsilon}_{ns} = \dot{\varepsilon}_{ns}^e + \dot{\varepsilon}_{ns}^p \]

(3.23)
where $\varepsilon_{ns}$ is the total strain increment, and $\varepsilon_{ns}^e$ and $\varepsilon_{ns}^p$ are the elastic and plastic components of incremental strain, and the shear stress increment is then defined as

$$
\tau_{ns} = \left( \frac{2G}{1 + \frac{G}{\delta G}} \right) \varepsilon_{ns}^p
$$

(3.24)

and the normal stresses are still defined as in equation (3.22) since they are not dependent on the incremental shear stress.

Using equations (3.23) and (3.24) and substituting into equation (3.18), the plastic strain, $\varepsilon_{ns}^P$ can be solved for directly by solving the slip function, $F$, equal to zero. After the plastic strain is determined the incremental shear stress can be determined by repeating equations (3.23) and (3.24).
SCRUBS Implementation

In order to activate the jointed media continuum model in the SCRUBS program, the first two material constants must have negative signs. Specifically, the negative of the bulk modulus, $K$, must be entered in place of Young's modulus, and the negative of the shear modulus, $G$, must be entered in place of Poisson's ratio. The negative on the first material constant is the flag which activates the formation of the elasticity matrix using $K$ and $G$ rather than $E$ and $\nu$ as described in Chapter 2.

Also for this case, the cohesion is stored in place of the yield stress, and the coefficient of friction is stored in place of the hardening modulus. Two new parameters, $G_s$ and $\delta$, are stored in the same material parameter array in positions already dimensioned for but not previously used.

The negative on the second material constant activates the jointed media material model. The incorporation of this model into SCRUBS is accomplished by the addition of a subroutine called JOINTM. This subroutine is accessed only if the second material constant is negative, and is called from within the RESIDUE subprogram. The purpose of JOINTM is to simply calculate the incremental stresses, given the
incremental strains. This process is usually accomplished in the LINEAR subroutine for the elastic case, and in a portion of RESIDUE for plastic deformations. The portions of these subprograms used to calculate the incremental stress vector are therefore simply skipped when the jointed media model is employed, and this task is performed in JOINTM.

The incremental stresses are calculated as described earlier in this Chapter. A complete FORTRAN listing of the JOINTM subroutine may be found in Appendix D.

EXAMPLES

To help illustrate the use of the jointed media capability, two small example problems will be presented. For comparative purposes, these examples will correspond closely to those presented in Chapter 2.

Simple Shear

This example corresponds only somewhat to the shifting plates example of Chapter 2. Being a continuum description, the jointed media model does not model the
joint descretely, but rather averages the effects of the joint throughout the jointed media continuum. The problem will, however, use the same geometry and loading conditions, which will now represent a jointed mass, fixed at its base and acted on by a horizontal load. Figure 13 shows the loading and the deflected shape of the problem analyzed.

Figure 13. Loading and deformed shape of simple shear example.

The bulk modulus was input as \(4 \times 10^7\) psi and the shear modulus as \(3 \times 10^7\) psi which would represent a common rock material. The fracture spacing was input as 2 inches, which simulates only one joint at midheight, and \(G_s\), the elastic modulus of slip, was varied. To insure that no plastic slip would occur, the cohesion was input extremely high, and the coefficient of friction was arbitrarily assigned a value of 1.0. Figure 14 shows that the lowering of \(G_s\) corresponds to lowering of \(G\) in the slip plane material.
Figure 14. Effect of varying $G_s$ of jointed material on deflection.

The examples do differ, however, in that the displacement effects only the joint in the example in Chapter 2, while the whole block is affected in this example.

It is also pointed out that for the boundary conditions imposed in this example, no plastic slip can be allowed to occur because rigid body motion would be initiated and no solution could be obtained.
Beam Example

The jointed media material model was also tested using the beam example used in Chapter 2. This example is a simply supported beam loaded with a concentrated load at midspan. The material constants were input the same as in the previous example, only using a fixed value of $3 \times 10^7$ psi/in for $G_s$. The cohesion was again set high so plastic slip would not occur. The mesh used and the deflected shape due to this loading are shown in Figure 15.

For comparison, the fracture spacing, $\delta$, was varied in this example to simulate use of varying numbers of slip planes as done in the Chapter 2 example. This problem was run using a very high value of $\delta$ to model no slip planes, and $\delta = 4$ inches, $2$ inches, and $1$ inch to model $1$, $2$, and $3$ slip planes respectively. TABLE 2 shows the effect of the varying of $\delta$ on the deflection.
at midspan and includes the deflections due to slip plane addition in Chapter 2 for comparison.

<table>
<thead>
<tr>
<th>NUMBER OF FRACTURES</th>
<th>DEFLECTION AT MIDSPAN</th>
<th>JOINTED MEDIA</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>SLIP PLANES</td>
<td>JOINTED MEDIA</td>
</tr>
<tr>
<td>0</td>
<td>.279 $\times 10^{-4}$</td>
<td>.140 $\times 10^{-4}$</td>
</tr>
<tr>
<td>1</td>
<td>.105 $\times 10^{-3}$</td>
<td>.201 $\times 10^{-4}$</td>
</tr>
<tr>
<td>2</td>
<td>.228 $\times 10^{-3}$</td>
<td>.242 $\times 10^{-4}$</td>
</tr>
<tr>
<td>3</td>
<td>.468 $\times 10^{-3}$</td>
<td>.419 $\times 10^{-4}$</td>
</tr>
</tbody>
</table>

It should be remembered that only the change in deflection can be compared because different material parameters were used in the two examples.

The plastic slip capability was also tested for this problem. Initially, the cohesion and coefficient of friction were set low enough to allow plastic slip; however, whenever plastic slip began, the solution became nonconvergent. It is believed that this is due to the fact that this plastic slip is distributed throughout the material, therefore no point in the continuum would be able to resist shear. This problem can be addressed by combining the two methods of incorporating joints. This is done by using the slip plane row addition to model each joint separately, but imputing the material model of this joint as a jointed media continuum having a low cohesion. In this way,
plastic slip occurs only in the plane of slip, and the solid material between slip planes resists the shear. Using this method, satisfactory results were obtained on the beam problem previously described, yielding results comparable with those in Chapter 2.
Due to the increased demand for coal in today's energy market, the use of underground mining methods to extract coal has increased considerably in recent years. This mining of the coal always leaves behind large cavities, and the prediction of the response of the overlying strata, particularly the subsidence at the surface, is of significant interest to mining engineers. Increased use of longwall [12] mining methods in which all of the coal in a seam is removed rather than leaving some of it as pillars to support the overburden, and current experimentation with underground coal gasification, are heightening the demand for the use of finite element analysis to help predict roof deflections and surface subsidence above the mined regions.

The SCRUBS finite element program is equipped with a special "rubble" logic which was added to model kinematic fall of elements above mined regions. In this method, if the state of stress on any element above a
mine cavity reaches some user specified failure criteria, that element falls. This method allows for element failure either part way or completely to the surface, and was incorporated to more accurately predict surface subsidence. [1] SCRUBS has been used in surface prediction analyses for both longwall mining applications [1] and underground coal gasification. [13]

The presence of joints and fractures in the material overlying mine cavities is often an important consideration in prediction of the behavior of such problems. By using the new capability added to SCRUBS in this work, the effects of these joints and cracks can be included in the analysis.

To investigate the effects of the jointed material modeling capabilities in SCRUBS, a mine cavity problem will be analyzed without fractures, and then with the fractures being modeled in several different ways. The results of the analyses will be demonstrated by showing the effects of the different crack models on the surface profile, as well as the roof of the cavity. Since the initiation of rubble formation is governed by the stresses in the elements directly above the cavity, a comparison of the differences in these stresses will also be included. Actual free fall of elements and
subsequent effects on surface subsidence will not be dealt with.

Centrifuge Experiment

The problem which will be analyzed corresponds to that of a centrifuge experiment performed at Sandia Laboratories. [14] A sketch of the test model is shown in Figure 16.

Figure 16. Experimental centrifuge model.
Materials

Two materials, an ashfall tuff and a silica flour, were used in constructing the model. The tuff is essentially a linear elastic material with a bulk modulus of $4.04 \text{ Gpa}$, a shear modulus of $3.42 \text{ Gpa}$, and a density of $1.13 \text{ Mg/m}^3$. The silica flour was used to simulate a soil layer above the tuff strata. This material had a density of $1.25 \text{ Mg/m}^3$, and was modeled as a very weak material ($K = 1.0 \text{ Gpa}$, $G = 1 \text{ Mpa}$).

Experimental Configuration

The model is $266.7 \text{ mm (10.5 in)}$ wide and has a mine drift that is $152.5 \text{ mm (6 in)}$ wide and $25.4 \text{ mm (1 in)}$ high. The overburden consisted of $44.4 \text{ mm (1.75 in)}$ of the silica flour on top of 47 layers of tuff. Each layer of tuff was approximately $3.18 \text{ mm (.125 in)}$ thick, making a total thickness of $149.5 \text{ mm (5.9 in)}$.

Finite Element Model

The finite element mesh used to model the problem is shown in Figure 17. The boundary conditions were imposed by fixing the bottom nodes in both directions, and fixing the side nodes in the $x$ direction only. The
left side represents a plane of symmetry. The crosshatched elements were "mined" using the element removing capability of the code.

Figure 17. Mine cavity roof deflection analysis mesh.

The problem is loaded by imposing a gravity load only. Since the material has such a low density, the gravitational factor is input as 3, which effectively triples the material weight. This also corresponds closely to the loading imposed on the sample when spun in the centrifuge.
For ease in modeling, only 10 layers of tuff separated by slip planes were defined in the model. While the results obtained will not duplicate the actual test, a good comparison of the slip plane capabilities can still be made. All other dimensions and parameters were input identically to those in the test model.

Joint Modeling

Five different methods were used to model the joints in the materials. First of all, the problem was solved by simply using a "reduced modulus" method. This method, which was used in a previous analysis of this problem, involves simply reducing Young's modulus of the cracked material by some assumed value. The problem was therefore run using a Young's modulus of 0.64 Gpa and a Poisson's ratio of 0.17. These values correspond to a reduction of Young's modulus by 20 percent. The problem with this model is that while reducing Young's modulus does weaken the material in shear, it also weakens it in compression, leading to errors in vertical displacements. Also, it is difficult to estimate this reduction to model the actual material.
The second method was also a "reduced modulus" method, however, this time K and G were input using a reduced shear modulus of \(0.174 \times 10^{10}\). While this method does model a general weakening in shear, it is still difficult to estimate G for a cracked medium.

The third joint model used is the low shear, slip plane model described in Chapter 2. For this model, the 10 slip planes were modeled separately. The low shear slip plane material was given a bulk modulus of 1 Gpa and a shear modulus of 0.01 Mpa.

The jointed media continuum description described in Chapter 3 was used for the fourth joint model. The same values of K and G were input as for the above case. The actual spacing of the joints used in method two was 19.7 mm and therefore this value was input as the fracture spacing, \(\delta\). Gs was input as 17.4 Gpa/m. The cohesion was input high in order to prevent plastic slip.

The fifth joint model was included to show the effects of plastic joint slippage. For this model a combination of the above methods was used. Discrete joints were used as in method 2, but the jointed media material description was used to model the slip material. The same material parameters were used as for the previous case, except that the cohesion was entered
as 10000 pa, and the coefficient of friction was entered as .45.

Figures 18, and 19 show the surface subsidence troughs and the mine cavity roof deflection profiles for each of the five methods compared with the unjointed model.
Figure 18. Surface profiles above mined region.

Figure 19. Mine cavity roof deflection profiles.

KEY:
A - NO JOINTS
B - REDUCED YOUNG'S MODULUS
C - REDUCED SHEAR MODULUS
D - JOINTED MEDIA
E - SLIP PLANES
F - SLIP PLANES WITH PLASTIC SLIP
It can be seen from these figures that each of these methods work well in modeling the weakening of this material due to the presence of joints. It should be noted that the results for the reduced modulus, slip plane, and jointed media methods cannot be compared directly. Since the formulation for each is different, it is difficult to create the equivalent material conditions using these different formulations. Similar results can be obtained for each of these cases by simply varying the material parameters input until similar results are obtained. However, while the reduced modulus methods produce good results, but they are not physically sound methods, and it is difficult to estimate the modulus reduction which would accurately model the actual material. The joint models are more useful in this way because only some knowledge of crack position and joint slip response is needed to estimate the parameters for use in these formulations. The figures do show that allowing plastic slip at the joints further weakens the bending stiffness producing greater deflections.

The effects of the slip model on the possible "rubbleization" of the material above the mine was also investigated. In the present formulation, the elements above the mine cavity will fall when the vertical stress in these elements exceeds some specified value. For all
cases run, the vertical stress in the elements directly above the mine cavity were nearly the same. This indicates that the failure criterion should possibly be changed, possibly to the flexural stress, in order to reflect the effects of material weakening due to joints.

Sandia Calculation Comparison

The centrifuge test model was also analyzed at Sandia using the JAC code to investigate the behavior of the layers immediately above the mine opening. Sliding interfaces were incorporated between each of the five layers shown in the greatly magnified distorted mesh plot shown in Figure 20a. This calculation required such a large amount of computer time that incorporation of additional layers would be prohibitive. This same problem was also modeled using the SCRUBS code utilizing separate slip planes between each of the first five layers on a similar mesh as shown in Figure 20b. The distorted shape produced in the SCRUBS calculation is similar to that of the JAC code, however, the run time of the SCRUBS calculation was very reasonable, thus demonstrating the efficiency of the slip plane model developed in this thesis.
Figure 20. Comparison of Sandia code "JAC" and SCRUBS.
CHAPTER 5
SUMMARY AND RECOMMENDATIONS

The ability to more accurately model and analyze geologic problems involving faults, joints, and cracked or fractured rock, was greatly enhanced through the additions made in this thesis. A versatile elastic-plastic, two-dimensional finite element code called SCRUBS was modified to include the joint model. Two capabilities, one which models separate, independent slip planes, and another which defines a jointed media material model, were added to facilitate more precise modeling of a wide variety of geomechanical problems. Single slip planes are recommended for faults and widely spaced joints or anywhere where the effects due to a specific joint are desired. The jointed media continuum description should be used when modeling rock mass with somewhat regular joint spacing. Thomas [2] indicates, however, that continuum theory is not applicable for all large jointed rock masses. Studies are currently underway to determine if continuum theory is applicable.
for several different types of rock mass configurations. If the representative elementary volume used is too large, then discrete joint models are necessary.

Although the addition of the new capability to the code is complete, further study of the effects of using this capability could be done. Possible areas of future work could include:

1. Studying the effects of faults on a variety of geomechanics problems using the slip plane addition to model the faults

2. Evaluating the effect of joints and fractures on problems where the rubble model available in SCRUNS is utilized

3. Investigating plastic yielding of the low shear slip plane material, particularly with utilization of the Drucker-Prager failure criteria

4. Performing experimental studies to determine single-joint slip responses in order to more accurately estimate the material constants which define the slip response

5. Performing of large-scale tests to verify analytical calculations.
BIBLIOGRAPHY
BIBLIOGRAPHY


APPENDIX A

PRESCRUBS USERS MANUAL
PRESCRUBS.BYU USER'S GUIDE

PRESCRUBS.BYU is an interactive user friendly preprocessor developed by the author to assist in rapid data input and modification. PRESCRUBS creates a data input file used by SCRUBS. This input file consists of: the solution controls, material properties, rubbleization properties, singular point definitions, boundary conditions, loading definitions, and printout controls.

PRESCRUBS has four features which makes it user-friendly: 1) fail-safe data input to prevent program abortion due to minor errors such as format errors, 2) data echo checks, which displays the set of data just entered to verify that it was input correctly, 3) modification of incorrect data, and 4) extensive help files to assist the user with program functions and options throughout the program.
This chapter explains the following basic functions of PRESCRUBS which are listed below. After explaining these options, the remainder of the chapter describes the input data which is used to run SCRUBS.

1. Creating a SCRUBS input file
2. Modifying or viewing a SCRUBS file
3. Reading a SCRUBS file
4. Writing a SCRUBS file
5. Printing a hard copy of the SCRUBS input data

Program Options

Option 1 (Enter SCRUBS data file)

Option 1 interactively guides the user through the data input necessary to run the program SCRUBS.FOR. The program is sub-divided into common data sets shown below. Data entry is structured so that the specification of certain flags and solution controls guides the user through only the necessary input for the particular solution selected. If the user desires help at any point, he may type "H" for help.

Solution controls
Material properties
Tabulated stress/strain data
Rubbleization data
Elemental mining data
Boundary conditions
Load description, point loads
Gravity an centrifugal, densities
Pressure loads
Increment, load factors
Impact nodes
Print-out controls

As the data is input it is checked to insure it is that valid. If the entry is valid, the user is prompted for following information. If the data is not valid, the user is prompted for the required data until it is correctly input.
The dimension of the arrays in the program require some practical limits. As values are entered they are checked against the program's current maximum dimensions. If the value entered is less than the maximum, execution continues. If the value exceeds this maximum an error message is printed, stating that this maximum value has been exceeded, and then the current maximum is printed. If a greater value than that of the current maximum is desired the program must be altered, to alter consult the the section on changing dimensions.

After data is entered, the information is displayed on the screen of the terminal. The user is prompted if changes are desired. If modifications are desired, a menu is displayed and the user can then enter the option to be corrected. If no further changes are desired the user may enter the last menu option (END MODIFICATIONS) or may simply type a carriage return, and program execution transfers to the next data set.

For example, after entry of the material properties the data is viewed and the user is prompted if changes are desired. If changes are desired, the user types Y and the menu shown below is displayed.

\textit{<SELECT OPTION TO BE MODIFIED>}
1. YOUNG'S MODULUS
2. POISSON'S RATIO
3. YIELD STRESS
4. HARDENING MODULUS
5. YIELD ANGLE
6. THICKNESS OF MATERIAL
7. RE-ENTER DATA
8. END MODIFICATIONS

If Poisson's ratio of material number 2 needed to be modified the user simply types 2 when prompted for the option to be selected. The user then enters the correct information corresponding to the prompts given. Upon re-entry of the modified data the complete data set is again viewed. If no further changes are desired the user may type 8 or simply type a carriage return \textit{<CR>}, and program execution continues. If further modifications are desired, the above process is repeated.

If the data set is too large to be shown on the screen, the user may select the (RE-SHOW) option and the data will be again displayed. Using \textit{<CTRL> S} the user may stop data display; using \textit{<CTRL> Q} data display is continued.
Option 2 (View or modify SCRUBS data)

This option allows the user to quickly and conveniently modify a data set that has just been entered or a set which was read from a disc file.

With the selection of Option 2, the main modification menu listed below is displayed and the user may choose the following options: 1. View or modify all data sets, or 2. View or modify a single data set. With this the corresponding data is then displayed on the terminal screen and the user is prompted whether changes are to be made. If no changes are desired the user types the return key, which transfers execution to the next related data block if the general modification option was selected or back to the main modification menu, if the single data option was selected.

If changes are desired another menu is displayed from which the user may modify the desired data. The data set is then re-displayed on the screen. If further modifications are desired the user may simply repeat this process. When all changes have been made the user types return and execution continues to the next data set or back to view or modify menu.

1. GENERAL MODIFICATION
2. SOLUTION CONTROLS
3. MATERIAL PROPERTIES
4. TABULATED STRESS/STRAIN DATA
5. RUBBLEIZATION DATA
6. ELEMENTAL MINING DATA
7. BOUNDARY CONDITIONS
8. LOAD DESCRIPTION, POINT LOADS
9. GRAVITY AND CENTRIFUGAL, DENSITIES
10. PRESSURE LOADS
11. INCREMENT, LOAD FACTORS
12. IMPACT NODES
13. PRINT-OUT CONTROLS
14. END MODIFICATIONS

Option 3 (Read from a disc file)

This option allows the user to read an existing set of data which is stored on a disc file. The file specified is read in the identical format used in SCRUBS.FOR.
The user may specify the data file to be read. The file name determines where the file is located and which file is to be read. If the correct file name is not entered or the file does not exist the user is again prompted for the file name until correctly entered.

Data files created by this pre-processor are named SCRUBS.DAT, hence the file name entered is simply SCRUBS.DAT, unless the file has been renamed.

If the file has been renamed or is in another area, the corresponding file name must be entered. Listed below is the maximum possible file name specifications.

\[
\text{DISC:} [\text{USER .SUBDIR]}\text{FILENAME}.\text{EXT};\text{VERSION}
\]

- DISC - disc on which users account is located
- USER - user's number account
- SUBDIRECTORY - user's subdirectory
- FILENAME - name of file to be read or created
- EXTENSION - type of file
  - DAT = default file
  - FOR = fortran file
  - OBJ = machine code file
  - EXE = executable file
  - DAT = data file
- VERSION NUMBER - current version of file

For example, DRC1:[204026.FINITE]SCRUBS.DAT would read the file named SCRUBS.DAT located in subdirectory FINITE of user 204026 on disc 1.

Option 4 (Writes to a disc file)

This option allows the user to write the current data set to a disc file. The data is written on the file in a format compatible with that necessary to run SCRUBS.

Note: The user may wish to insure the existence of the file created by checking his directory, after program termination.
Option 5 (Prints a hard copy of data set)

Selecting this option sends a hard copy of the current data appropriately labeled to the user's default line printer.

Option 6 (Ends program execution)

This option terminates program execution.

NOTE: All menus are designed to end current operations by selecting the last menu option, but for convenience the carriage return may also be used.

All "yes" or "no" questions will default to "no" with a carriage return.

If data set is too large to fit onto the screen, <CTRL> S may be used to stop the display and <CTRL> Q may be used to continue display.

If at any point the user desires help, type "H" and a help message will be printed.
SOLUTION CONTROLS

This section of SCRUBS will input controls which specify how the problem will be analyzed. The solution controls in this section are used as flags throughout the data input process and later during the execution of SCRUBS. Care should be taken in specifying the correct controls and flags corresponding to the problem being considered.

<ENTER TITLE OF PROBLEM>
This is the title or description of current problem (80 characters or less).

<ENTER NUMBER OF PROBLEMS TO BE SOLVED>
This is the number of problems to be considered per run (usually one).

<ENTER NUMBER OF LOAD CASES PER PROBLEM>
This is the number of different loading combinations to be considered per problem.

<ENTER NUMBER OF DOF PER NODE>
DOF (Degrees of Freedom) is the number of coordinate directions in which nodes may translate.

<ENTER NUMBER OF DIFFERENT MATERIALS>
Input the number of materials of which the body is composed. Current maximum is 10 materials.

<ENTER ELEMENT TYPE NUMBER: 1=LINEAR, 2=QUADRATIC, 3=CUBIC>
SCRUBS is capable of handling the three types of elements shown in Figure 1. Quadratic and cubic elements have mid-side nodes which allow more complex modes of deformation, but computation costs are slightly higher. At the present time QMESH.BYU, the mesh generator, will not process the cubic element.
FIGURE 1. Types of Elements

The number of Gauss points refers to the order of Gaussian quadrature defined for an element. A higher order of quadrature increases the number of integration points, which can give a higher degree of accuracy. Increasing the number of integration points increases the computational cost.

1st Order  2nd Order  3rd Order

Figure 2. Gaussian Quadrature

<ENTER SOLUTION ALGORITHM NUMBER>

0=ELASTICITY ONLY
1=CONSTANT STIFFNESS
2=TWO STEP PROCESS
3=TANGENT STIFFNESS

The solution algorithm determines which type of stress/strain relationship is to be considered, linear or non-linear. The first option analyzes elastic solutions only and care should be taken to ensure that the stresses remain elastic. The last three options treat materials in the plastic region. The tangent stiffness is the best of the three.
<ENTER STRESS/STRAIN TYPE NUMBER>
  0=PLANE STRAIN
  1=PLANE STRESS
  2=AXISYMMETRIC PROBLEM

The stress/strain type allows the user to model different geometric conditions. Plane strain is used for thick shapes. Plane stress is used for thin shapes, such as plates. Axisymmetrical problems model geometrical shapes which are symmetrical about the Z axis.

<ENTER YIELD CONDITION PARAMETER NUMBER>
  1=VON MISES
  2=TRESCA
  3=DRUCKER-PRAGER
  4=BELTRAMI

This parameter specifies the yield theory used. Von Mises and Tresca are generally used with metallic materials and Drucker-Prager and Beltrami are used for soils. See section 2.1 on material modeling.

<ENTER THE STIFFNESS CONTROL NUMBER>
  0=NO.ELASTICALLY COUPLED NODES
  1=INPUT STIFFNESS COEFFICIENTS

The stiffness control allows the input of the stiffness matrix, this option is rarely used (use 0).

<ENTER SLIP PLANE FLAG, 0=NO SLIP PLANES, 1=SLIP PLANES>

The user here identifies whether or not any slip planes are to be specified in the problem.

<ENTER RUBBLE FLAG NUMBER, 0=NO RUBBLE, 1=RUBBLE>

At this point the user may specify if a material or materials are to be allowed to rubbleshoot.

<ENTER BANDWIDTH>

This refers to the size of the stiffness matrix and is not used unless a stiffness control of 1 is specified.

<ENTER NUMBER OF COORDINATES PER NODE,(DEFAULT=2)

The number of coordinates is the dimension of the space considered.
<ENTER NUMBER OF GAUSS POINTS>
This is usually the same as the Gaussian quadrature entered earlier. This prompt refers to the number of gauss points for nodal force residual calculation and stress storage.
MATERIAL PROPERTIES

This section defines properties which model a given material. Material properties are input for the total number of materials in the body. To reduce needless interactive prompting the user is not be prompted for the yield angle or the material thickness unless the corresponding solution controls were previously entered.

<ENTER YOUNG'S MODULUS OF ELASTICITY FOR MATERIAL I >
Young's modulus is the slope of the stress/strain curve in the elastic region, for material I. A negative value in this position activates the formulation of the elasticity matrix using $K$ and $G$ rather than $E$ and Poisson's ratio. In this case the bulk modulus $K$, is entered as a negative value here and the shear modulus $G$, is entered in the place of Poisson's ratio.

<ENTER POISSON'S RATIO OF MATERIAL I >
Poisson's ratio is the ratio of lateral to longitudinal strains, of material I.

<ENTER YIELD STRESS OF MATERIAL I >
This is the magnitude of stress at which the material begins to yield and deform plastically. If the Drucker-Prager option is selected the cohesion, $c$, is entered here.

<ENTER HARDENING MODULUS OF MATERIAL I>
At this point the user may specify how the plastic stress/strain curve is to be defined. This stress/strain curve defines the curve used by SCRUBS to interpolate the magnitudes of stress and strain for the plastic solutions. The hardness modulus is the slope of the stress/strain curve in the plastic region. Note: this is true stress, true strain rather than engineering stress strain. If the modulus is entered in as a positive value, the plastic stress/strain is defined by the hardening modulus. If it is entered in as negative, the user will be prompted for tabulated stress/strain points defining the plastic stress/strain curve. The two options for defining the plastic region are illustrated in Figure 3. If the Drucker-Prager option is selected, the cohesion, $c$, is input as a function of plastic strain, rather than yield stress.
Defined by the hardening modulus
Defined by tabulated stress/strain points

FIGURE 3. Plastic Stress/Strain Curve

<ENTER CONICAL YIELD SURFACE ANGLE OF MATERIAL I >
This is the angle of friction and is entered if the Drucker-Prager yield condition is chosen.

<ENTER THICKNESS OF MATERIAL I >
Enter the material thickness if plane stress conditions are considered.
Slip Plane Material

When the slip plane flag is set the user will automatically be prompted for the material parameters for the row of thin elements which will be added along the slip plane.

<ENTER BULK MODULUS OF SLIP PLANE MATERIAL>
Here the bulk modulus for the slip plane material is entered. This number will be automatically negated and stored in place of Young's Modulus.

<ENTER SHEAR MODULUS FOR SLIP PLANE MATERIAL>
The shear modulus for the slip plane material is usually input as a relatively low number to model a material with little shear resistance. This number is stored in place of Poisson's ratio.
For the slip plane material the yield stress is set at 1.0. This is done simply to insure that the slip plane material remains in the elastic range.
For a plane stress formulation, the user is also prompted for the thickness of the slip plane material.
Jointed Media Material Description.

A material may be defined as a jointed media continuum by entering the negative of the shear modulus in place of Poisson's ratio. When Poisson's ratio is entered as a negative number, the user is prompted for these parameters as follows:

<ENTER COHESION OF MATERIAL I>
Here the cohesion of the fractures is entered.

<ENTER COEF. OF FRICTION OF MATERIAL I>
This is the coefficient of friction of the fractures.

<ENTER JOINTED MEDIA FRACTURE ANGLE OF MATERIAL I>
The angle the fractures make with respect to coordinate axes must be entered here.

<ENTER FAILURE STRESS OF FRACTURES OF MATERIAL I>
The maximum tensile stress allowed normal to the fractures is entered. For rubble problems, this is take to be the failure stress entered in the rubble section, and this prompt is not given.

<ENTER FRACTURE SPACING FOR MATERIAL I>
The approximate distance between joints in the material is entered here.

<ENTER MODULUS OF SLIP (GS) FOR MATERIAL I>
As is a modulus of slip and represents the slope of the line describing the displacement to stress elastic curve of a single joint.
Tabulated yield stress as a function of plastic strain is to be entered for a given material if the corresponding hardening modulus for that material was set negative. These stress/strain points define the plastic stress/strain curve point by point rather than using a single slope hardening modulus, (see hardening modulus in the Material Properties Section). Sample true stress/strain curves are given in Figure 4.

**FIGURE 4. True Stress Strain Curves**
This section defines the rubble parameters and element death (or mining) sequence. The rubble bulking parameter and the failure stress level for each material are entered for the rubble description. The specific elements that are mined are also defined.

**Enter Bulking Parameter of Material I**

The bulking parameter is the ratio of material expansion, the initial volume over the new volume, always less than one.

**Enter Failure Stress of Material I**

The failure stress is the magnitude of stress at which the material fails and rubbleization occurs.

**Enter Minimum Z Coordinate, of Which Elements May Not Pass (E Format)**

The minimum Z coordinate specifies the coordinate through which the rubbleized elements may not pass.

**Enter Total Number of Increments**

The total number of increments specifies the number of successive increments for which a given number of elements are to be removed.

**Enter Number of Elements to Be Mined on Increment I**

This defines the number of elements which are to be removed, or mined, on increment I.

**Enter N Element Numbers to Be Mined on Increment I**

The N specific elements which are to be removed on increment I are now input.
BOUNDARY CONDITIONS

This section defines the boundary conditions and provides the option of specifying a singular quarter point crack tip element. The user inputs the boundary flag assigned in QMESH along with the desired boundary condition. PRESCRUBS assigns the desired boundary condition to each nodal point that has that boundary flag.

<DO YOU WANT TO DEFINE A SINGULAR POINT (Y OR N)?>

If a singular point is to be defined for use in a crack-tip analysis enter 'Y' for yes. Note: a crack-tip analysis is valid only for the quadratic element.

<ENTER NODE NUMBER OF SINGULAR POINT>

Enter the node number of the point of the crack-tip. Defining a singular point will automatically translate the midside nodes of the surrounding elements to their 'quarter point' positions.

<ENTER FILE NAME>, TYPE H FOR HELP

Enter the name of the QMESH file containing the finite element mesh and boundary flags of the body.

DO YOU TO CONSIDER BOUNDARY ANGLES? (Y or N)

This option is used if a boundary of a given body is inclined with respect to the coordinate axis. By typing 'Y' the angle between the axis and boundary can be defined.

DO YOU WANT TO READ A BOUNDARY FLAG? (Y or N)

This prompt allows the user to interactively process boundary conditions. Answering "yes" to the prompt starts the input loop; answering "no" ends the boundary processing.

<ENTER BOUNDARY FLAG NUMBER AS SET IN QMESH>

This is the boundary flag set in QMESH identifying a given boundary.
This boundary code defines a given set of displacement constraints which act on the nodal points corresponding to the flag set in QMESH.

This is the specified displacement in the R (i.e. X) direction.

This is the specified displacement in the Z (i.e. Y) direction.

This is the thickness of the new row of elements that will be added defining the low shear slip plane.

This defines the angle of the slip plane relative to the coordinate axes.

Enter the boundary angle described previously. If no boundary angles where specified this prompt will not be given.

FIGURE 5. Boundary Angle
<ENTER NUMBER OF SUMMED REACTION NODES>
This is the total number of reaction nodes which are to be summed.

<ENTER SUMMED REACTION NODE NUMBER>
Enter the reaction node numbers.
LOADING CONTROLS

This section specifies the manner in which the body is to be loaded. The user may specify independent loadings, loadings proportional to a load factor array, or loads equal on all increments. The user may also specify the possible load combinations which are: concentrated loads, pressure or shear tractions, gravitational loads, or centrifugal forces.

<ENTER TITLE FOR LOADING CASE I>
This is the title or description of the current loading case (80 characters or less).

<ENTER INCREMENT CONTROL FOR PRESCRIBED LOADS AND DISPLACEMENTS>
O=EQUAL INCREMENTS
1=INCREMENTS PROPORTIONAL TO FACTOR ARRAY
2=INDEPENDENT SET OF LOADS FOR EACH INCREMENT

The increment control allows the user to specify how loads or displacements are to be handled over a given time increment. Displacements and loads can be treated as constant, proportional to the elements of the factor array, or as different loadings for each increment.

<DO YOU WANT TO CONSIDER POINT LOADINGS? (Y or N)>
This allows the application of concentrated loads onto nodal points.

<DO YOU WANT TO CONSIDER GRAVITY LOADS? (Y or N)>
This allows the processing of forces due to gravitational or magnetic fields.

<DO YOU WANT TO CONSIDER PRESSURE LOADS? (Y or N)>
This allows surface pressures and shear tractions.

<DO YOU WANT TO CONSIDER CENTRIFUGAL FORCES? (Y or N)>
This considers forces caused by centripetal acceleration.
POINT LOADING DATA

This section enters concentrated load data. The total number of loaded nodes is entered, then each individual node with the corresponding load in the X and Y direction are entered.

<ENTER NUMBER OF NODES WHERE POINT LOADS ARE APPLIED>
Enter the total number of nodal points in the body which are to be loaded.

<ENTER NODE NUMBER AND FORCES IN X,Y DIRECTION>
Enter three numbers: the nodal point to be loaded, the magnitude of the concentrated load in the X direction, and the magnitude of the concentrated load in the Y direction.

FIGURE 6. Point Load in -Z Direction
GRAVITATIONAL, CENTRIFUGAL LOADINGS

This section enters the gravitational constant and the angular velocity and material density; if either gravitational or centrifugal loading was previously specified.

<IF GRAVITY AXIS IS TILTED ENTER GRAVITY ANGLE>
If the gravitational axis is not along the negative Z axis enter the value of the angle between the negative Z axis and the gravity axis. The angle entered is positive clockwise.

<ENTER CONSTANT OF ACCELERATION DUE TO GRAVITY>
Enter the magnitude of the gravitational constant or the constant of the magnetic field.

<ENTER ANGULAR VELOCITY IN RADIANS/UNIT TIME>
Enter the angular velocity of the body as it revolves about the Y axis.

<ENTER DENSITY FOR MATERIAL I>
For the material number prompted, enter the material density. Enter densities for all materials.
PRESSURE LOADINGS

This section allows the input of pressure loadings or tractions on a given element surface. The total number of elements on which pressure or shear tractions are applied is entered, then each element along the loaded surface is entered with its corresponding node number and the pressure and shear tractions at the nodes.

<ENTER NUMBER OF LINE ELEMENTS ON WHICH PRESSURES ARE APPLIED>

Enter the total number of elements on which pressure loads are applied.

<ENTER I SPECIFIED NODE NUMBERS ON ELEMENT PRESSURE SURFACE>

Enter I nodal points which lie on the edge of the element being loaded. The number of nodes varies depending on the type of element being considered. Linear elements have two loaded nodes, quadratic elements have three, and cubic elements have four.

<ARE SAME LOADS APPLIED ON ALL NODES OF THE LINE ELEMENT? (Y or N)>

If the normal or tangential pressures are constant across the loaded surface only those pressures need to be entered rather than the individual pressures at each nodal point. If a linear element is being used the individual pressure loads must be applied for each node.

<ENTER CONSTANT NORMAL PRESSURE>

This is the magnitude of a uniformly distributed load applied across the loaded surface.

<ENTER CONSTANT TANGENTIAL PRESSURE>

This is the magnitude of a uniformly distributed shear traction applied across the loaded surface.
<ENTER NORMAL PRESSURE OF NODE I>

Enter the magnitude normal pressure applied on the nodes previously entered. Note these pressures must be entered in the same order as were the corresponding nodal points.

<ENTER TANGENTIAL PRESSURE OF NODE I>

Enter the magnitude of the shear traction in the same manner as the normal pressures for individual nodal points.
LOAD FACTORS AND INCREMENT DATA

This section specifies increment and load factor data. The load factors are used to increase or decrease the magnitude of the current loadings over a given time increment. At this point the user may specify the type of output data which is to be printed. Extensive output is available or a reduced amount of output may be specified according to the user's desire.

<ENTER NUMBER OF LOAD INCREMENTS>
This is the number of incremental steps in which the loading will be applied.

<ENTER LOAD FACTOR FOR INCREMENT I>
This is a factor applied to the specified loadings or displacements to define the magnitude of the load to be added for increment I. The total load at any step is, of course, the sum of all the increments to that point.

The SCRUBS.LIS file will contain a listing of the solution for each loading increment.

<ENTER INITIAL AND FINAL INCREMENT WRITE-OUT INDICATOR>
0=NO OUTPUT
1=DISPLACEMENTS AT NODAL POINTS
2=REATIONS AT CONSTRAINED NODES AS WELL
3=STRESSES AT GAUSS POINTS AS WELL
4=RESIDOAL FORCES AT NODES AS WELL

The write-out indicator is a three digit number specifying the type of output data for the initial and final increment. The first of the three digits indicates the type of output for the initial increment, the second digit is zero, and the third digit indicates the type of output for the final increment. The type of output is specified by choosing one of the five conditions listed above. For example, the number 100 will cause the displacements at the nodal points to be printed for the initial increment and no output will be printed for the final increment. For most cases simply use 003.

<ENTER MAXIMUM NUMBER OF ITERATIONS>
The maximum number of iterations is the number of iterations allowed for convergence to occur. If convergence has not occurred calculations will be terminated.
The convergence factor is the value used to determine whether the solution is within a certain degree of accuracy (i.e., convergence has been obtained). SCRUBS checks convergence on the ratio of the sum of the norm of force residuals divided by the sum of the norm of applied forces.
IMPACT BOUNDARY NODES

This section enters impact boundary nodal points.

<ENTER NUMBER OF IMPACT BOUNDARY NODES>
This is the total number of boundary nodes subjected to impact.

<ENTER IMPACT NODE NUMBER>
This is the boundary node number subjected to impact.
PRINT-OUT CONTROLS

This section allows the user to control the printed output data. Output for all elements or for a few selected elements may be defined.

<ENTER NUMBER OF ELEMENTS TO BE PRINTED>
This is the total number of elements for which output data is to be printed.

<ENTER NODAL OUTPUT CONTROL>
  0 = ALL NODES PRINTED
  1 = NO NODES PRINTED
This control specifies whether data for all or no nodes are to output.

<ENTER ELEMENT TO BE PRINTED>
Enter the number of the element which is to be printed.
APPENDIX B

SCRUBS USERS MANUAL
SCRUBS is a FORTRAN computer program designed to compute the failure, collapse, and resulting subsidence of geologic materials. The uniqueness of this program is its ability to model rubble formation and collapse in a continuum, as opposed to a discrete, sense. Both pre and post failure aspects of particular problems are treated. SCRUBS is a nonlinear finite element program which is capable of determining the deformation and state of stress in plane and axi-symmetric bodies. Pre and post failed material properties may be elastic-perfectly plastic, or elastic work hardening. Linear, quadratic or cubic isoparametric elements are used to represent the region and its boundary. Four different yield conditions can be imposed on the materials: von Mises, Tresca, Drucker-Prager, or Beltrami. Three different algorithms for solving the nonlinear discretized equations are available. They are: 1) a constant stiffness initial stress method, 2) a two step process where the stiffness matrix is updated on the second iteration of an otherwise initial stress process and 3) a regular tangent stiffness method. A check on force residuals is used to evaluate convergence for any load increment.

Three specific files are read and three other specific files are created by SCRUBS. The three files that are read have the following names and functions.

QMESH9.DAT = Mesh information as created by QMESH.BYU
SCRUBS.DAT = SCRUBS input data
SCRUBS.RST = Restart file

The three files that are created have the following names and functions.

SCRUBS.MOV = Displacement, stress and strain data at each load step
RUBBLE.MOV = Element rubbleization plot file
SCRUBS.RST = Restart file
SCRUBS.LIS = List file in readable format

The contents of these seven files are now explained in detail.
This is the standard QMESR.BYU renumbered file that contains the two dimensional finite element meshing information. It consists of five records written in blocked binary form. The content of these records is as follows.

Record 1. (8 words) A comment, packed 4 characters per word. This comment is from the COMMEN card in the QMESH input.

Record 2. (4 words)
- KKK, the number of elements in the mesh
- NNN, the number of nodes in the mesh
- NFF, the number of words in the boundary flag table
- MAXDIF, the maximum difference of node numbers for any element in the renumbered mesh

Record 3. (2 x NNN words) the lists of nodes, in renumbered order; that is
- \((X(N), Y(N), N=1, NNN)\),
- \((R(N), Z(N), N=1, NNN)\)

Record 4. (5 x KKK words) The list of elements:
- \((N1(K), N2(K), N3(K), N4(K), MAT(K), K=1, KKK)\)
where \(N1(K)\) to \(N4(K)\) are the node numbers for the \(K\)-th element in counterclockwise order, and \(MAT(K)\) is the material number of the \(K\)-th element.

Record 5. (NFF words) the list of boundary flags and nodes,
- \((IFLAG(I), I=1, NFF)\). Flags will be negated to distinguish them from nodes, and the corresponding node or list of nodes will follow each flag.
If \(NFF=0\), this record will not be written.

End-of-File mark.
SCRUBS.DAT

This file contains the data necessary to define the problem to be analyzed. The groups of lines of this file and the specified format of each line follows.

GROUP 1  INTERNAL MESH GENERATION OR RESTART SPECIFICATION

Line 1. (4I5) Geometry parameters
   1 -  5 Number of elements (NEL) **
   6 - 10 Number of nodes (NODES)
   11 - 15 Number of load cards (NUMPC)

** NOTE if NEL = 0, omit remaining lines of the group and read total mesh from the input file QMESH9.DAT. If NEL = -1, the problem is to be restarted and the remaining lines of this group are omitted.

Line 2. (6I5) N,IX array
   1 -  5 Element number (N)
   6 - 10 Ith nodal point (IX array)
   11 - 15 Jth nodal point (IX array)
   16 - 20 Kth nodal point (IX array)
   21 - 25 Lth nodal point (IX array)
   26 - 30 Material number (MAT, IX array)

In general every element must be defined; but with the semi-automatic mesh generation feature, a minimum of one element per row need be input. For example, if element 10 is read with values I=12, J=13, K=24, L=23, and MAT=1, and the next element is read is element 15 with values I=23, J=24, K=35, L=34, and MAT=1, then element 11 would be assigned values 13, 14, 25, 24, and 1.

Line 3. (I5,F5.0,F10.0)
   1 -  5 Nodal point number (N)
   6 - 10 Boundary condition code (CODE)
   11 - 20 Radial coordinate (R)
   21 - 30 Axial coordinate (Z)

In general, every nodal point must be defined, but since the program has a semi-automatic mesh generation feature, a minimum of two nodal points per row need be input and the intervening points will be assigned coordinates based on a linear interpolation procedure. For example, if nodal point 1 is the first point in a row with coordinates (2.5, 5.4), and nodal point 11 is the next point defined with coordinates (12.5, 10.4), then nodal point 2 will be located at (3.5, 5.9), etc.
GROUP 2 SOLUTION CONTROLS

Line 1. Problem identification
1 - 5 Total number of problems to be solved in one run (NPROB)

Line 2. Title (20A4)
1 - 72 Title of problem.

Line 3. Control data (16I5)
1 - 5 Total number of nodes, NP (not more than 1000)
6 - 10 Total number of elements, NE (not more than 1000)
11 - 15 Total number of restrained boundary points, NB
16 - 20 Total number of load cases/problem, NLD
21 - 25 Number of d.o.f. per node, NDF
26 - 30 Number of different materials, NMAT (not more than 10)
31 - 35 Element type 1=linear, 2=parabolic, 3=cubic, NSFR
36 - 40 Number of gauss points for stiffness calc. NGAUS
41 - 45 Solution algorithm, NALGO
  0 = elasticity only
  1 = constant stiffness
  2 = two step process
  3 = tangent stiffness
46 - 50 Stress/strain type NPP
  0 = Plane strain
  1 = Plane stress
  2 = Axisymmetric problem
51 - 55 Yield condition parameter, NYIELD
  1 = Mises
  2 = Tresca
  3 = Drucker-Prager
  4 = Beltrami
56 - 60 Input stiffness control, NT
  0 = Number of elastically coupled nodes
  1 = input stiffness coefficients
61 - 65 Flag for Rubble calculation, NL
  0 = No rubble
  1 = rubble
66 - 70 Bandwidth (leave blank unless NT = 1)
71 - 75 Number of coordinates per node NCORD (default 2)
76 - 80 Number of gauss points for nodal force residual calculation and stress storage (MGAUS) result even for elastic solution NGAUS defaults for NGAUS.

******************************************************************************
GROUP 3 - MATERIAL PROPERTIES

Line 1. Material data (I10, 7F10.2) NMAT Lines, limited to NMAT=10
(Note, if NL=1, card NMAT of this section must give
unconsolidated rubble constants)
1 - 10 Material property number N
11 - 20 Young's modulus ORT(N,1)*
21 - 30 Poisson's ratio ORT(N,2)**
31 - 40 Yield stress ORT(N,4)
41 - 50 Hardening modulus ORT(N,5)
51 - 60 Conical yield surface angle ORT(N,6)
61 - 70 Thickness (leave as zero if NPP = 1)

* A negative value in this position activates the low shear
material description. In this case, the bulk modulus, K,
is the absolute value of ORT(N,1), and the shear modulus, g,is the value of ORT(N,2)

** A negative value in this position activates the jointed
media material description. For this case, one more line is
read, for that material only, defined as follows: (2G12.5)
1 - 12 Failure stress of fractures ORT(N,7)
13 - 24 Fracture spacing ORT(N,8)
25 - 36 Slip modulus of joint ORT(N,10)

Line 2. Tabulated plastic stress-strain data
(Only for materials with ORT(N,4) negative) (I10,F10.3)
1 - 10 Number of tabulated strain points, NTAB
11 - 20 Strain increment in percent strain, TABSTN(N)
If ORT(N,4)>0 these values default to 2 and 100
respectively.

Line 3. String of stress values, NTAB in number, one line required for
each material (15P8.0)
1 - 8 Stress value for point 1
9 - 16 Stress value for point 2
17 - 24 Stress value for point 3
25 - 32 Stress value for point 4

etc.

Line 4. String of strain values, NTAB in number, one line required for
each material (15P8.0)
1 - 8 Strain value for point 1
9 - 16 Strain value for point 2
17 - 24 Strain value for point 3

etc.
GROUP 4  RUBBLEIZATION DATA  (Omit this group if NL=0)

Line 1. Rubble material data, one line required for each material
   (I10,2P10.3)
   1 - 10 Material number, N
   11 - 20 Bulking parameter, ORT(N,7)
   21 - 30 Failure stress ORT(N,8)

Line 2. Minimum boundary coordinate (E10.3)
   1 - 10 Value of minimum Z coordinate for which element nodal
   points cannot pass through

Line 3. Material failure flag (I5)
   1 - 10 Failure flag, NRFF
      If this flag is non zero, a subroutine, READF is
      called. This subroutine defines a list of elements
      that can be removed (i.e. mined) on a particular
      increment. NRFF is the total number of increments
      allowed (10 max).

Line 4. Element mining data, omit if NRFF=0. (16I5)
   1 - 5 Number of elements to be mined on increment 1, NME(1)
   6 - 10 Number of elements to be mined on increment 2, NME(2)
   11 - 15 Number of elements to be mined on increment 3, NME(3)
   16 - 20 Number of elements to be mined on increment 4, NME(4)
       :     
       :     
       :     
       etc.

Note: Currently 16 is the maximum number of elements
that can be mined on any given increment.

Line 5. Specified elements to be mined, one line required for each
       increment, NRFF total, omit if NRFF=0. (16I5)
   1 - 5 Element to be mined on this increment, MINES(INC,1)
   6 - 10 Element to be mined on this increment, MINES(INC,2)
   11 - 15 Element to be mined on this increment, MINES(INC,3)
   16 - 20 Element to be mined on this increment, MINES(INC,4)
       :     
       :     
       :     
       etc.
GROUP 5 BOUNDARY CONDITIONS

Line 1. Ancillary control data (1615)
   1 - 5 Not used - leave blank
   6 - 10 Not used - leave blank
   11 - 15 Number of boundary points NNBB
   16 - 20 Number of reactions to be summed MPR
   21 - 80 Indexing vector, automatic generation of NOPE array
        IB(I), I=1,12

Line 2. Boundary conditions (215,4F10.3) NNBE cards in ascending order
       of boundary nodes.
       1 - 5 Boundary node number NBC(I)
       6 - 10 u condition, 0=free, 1=fixe d
       11 - 20 v condition, 0=free, 1=fixe d
       21 - 30 Prescribed u displacement US(I,1)
       31 - 40 Prescribed v displacement US(I,2)
       41 - 50 Boundary angle (if inclined) ANG(I)
       in degrees of X' axes from X.
       Positive when counterclockwise.
       Lines 3 and 4 are repeated in NSP blocks, each block representing a new
       slip plane. If NSP = 0, omit lines 3 and 4.

Line 3. Slip plane control data (215,G10.3)
       1 - 5 Number of nodes on slip plane boundary NSLP(N)
       6 - 10 Angle flag of slip plane IANG(N)
       11 - 20 Thickness of slip plane STHIK(N)

Line 4. Slip plane node list (1615) NSLP(N)/16 cards
       1 - 5 Node number ISLP(I,1), I = 1,NSLP(N)
       6 - 15 Node number
       etc.

Line 5. Summed reaction nodes (1615) (If MPR>0, otherwise omit)
       1 - 5 Node number NOPR(I), I = 1, MPR
       6 - 10 Node number
       etc.
GROUP 6 LOADING DEFINITION

Line 1. Ancillary load control data (2I5)
- 5 Initial stress counterISTS (not used)
  6 - 10 Increment control [Note LDTYPE applies to both prescribed
loads and displacements.)
  0 ... equal increments
  1 ... increments in proportion to total
       according to the fac array
       \[ x \] = PAC [x ]
       inc inc total
  2 ... independent set of loads for each
       increment

Line 2. Loading title (20A4)

Line 3. Loading control (8I5)
- 5 NSTRS (Not used)
  6 - 10 Number of nodes where point loads are applied, NRE
  11 - 15 Gravity load flag, if nonzero read lines 5 and 6.
  16 - 20 Pressure load flag, if nonzero read lines 7, 8 and 9.
  21 - 25 Temperature load flag, currently not available.
  26 - 30 Centrifugal force flag, if nonzero read lines 5 and 6.

Line 4. External Point load data (I10,3F10.3) NRE lines
- 10 Node number
  11 - 20 Force in the X direction
  21 - 30 Force in the Y direction

Line 5. Gravity and centrifugal force data (3F10.3) two or three lines
if NRC + NRG > 0. Otherwise go to line 7.
- 10 Angle of Y or Z axes from gravity axes, clockwise
  positive. THEDA
  11 - 20 Number of g's applied for this system
  (Only when NRC = 1) GRAV
  21 - 30 Angular velocity in radians/unit time
  (only when NRC = 1) ANGVEL

Line 6. Density (8F10.3)
- 10 Density for material 1 DENS(1)
  11 - 20 Density for material 2 DENS(2)
  etc. (Density units are weight/volume)
Line 7. Pressure element control, if no pressure loads (i.e. NRPRS=0) leave out and go to line 10. (815)
1 - 5 Number of line elements on which pressures are applied
LNE

Note each line 8 and 9 should be in sequence i.e., 8,1
9,1
8,2
9,2
etc.

Line 8. Element nodes with applied pressure, one line/element (415)
1 - 5 node 1
0-------------0
6 - 10 Node 2
0-------0------0
11-15 Node 3
0------0------0
16-20 Node 4
0-----0-----0

NOPL(I),I=1,NSPR+1

Node numbers on element pressure surfaces numbered in counterclockwise direction.

Line 9. Pressure data (8F10.3)
(a) same loads on each node of line element
1 - 10 Normal pressure (Pn)

11 - 20 Tangential pressure (Pt)

21 - 30 Zero

(b) Different loads at each node of line element Pn
1 - 10 Pn 1
11 - 20 Pt 1
21 - 30 1
31 - 40 1

etc.

Return to 8 for next line element.

Note (a) does not apply for linear elements
Line 10. Load increment data (I5)
1 - 5 Number of load increments NINC

Line 11. Load factor data (16F5.3)
1 - 5 Multiplier for increment 1 FAC(1)
6 - 10 Multiplier for increment 2 FAC(2)

etc.

Line 12. Output control data (16I5) NOUT(I), I=1,NIC
1 - 5 s 0 t  s = 1st iteration write out indicator
t = final iteration write out indicator

all printout is given up to and including s or t

s or t = 0 no output
= 1 displacements at nodal points
= 2 reactions at constrained nodes as well
= 3 stresses at gauss points as well
= 4 residual forces at nodes as well

6 - 10 NOUT(2)
11 - 15 NOUT(3)

etc.

Line 13. Iteration and convergence data (I10,F10.3)
1 - 10 Maximum number of iterations NIT
11 - 20 Convergence factor in percent CONPAC

checks on 100 x (Sum of |force residuals|)
divided by (Sum of |applied forces|)
GROUP 7 REDUCED PRINTOUT CONTROLS

Line 1. Print control (2I5)
1 - 5 Number of elements to be printed. If negative, all elements printed.
6 - 10 Nodal point output control, if zero, no nodes printed, if 1, all nodes printed.

Line 2. List of elements to be printed (16I5) (only if NELP > 0)
1 - 5 element number to be output
6 - 10 element number to be output
11 - 15 element number to be output
::
::
:: etc.
This file is written in blocked binary form as follows:

Record 1. (22 words) the first twenty words are the title of the problem and the last two words are NE, the total number of elements, NP, the total number of nodal points, that is,

\[ \text{TITILE, NE, NP} \]

Record 2. (2xNP+5xNE words) the first two lists are the nodal point coordinates, the next is the element connectivity, and the last is the element type.

\[ (\text{CORD}(I,1), I=1, \text{NP}), (\text{CORD}(I,2), I=1, \text{NP}), \\
((\text{NOP}(4*(K-1)+M), K=1, \text{NE}), M=1, 4), \\
(\text{IMAT}(I), I=1, \text{NE}) \]

The following records are repeated for each load increment.

Record 1'. (1 word) Increment number, TINC.

Record 2'. (3x2xNP words) Nodal point displacements, repeated 3 times.

\[ (\text{TDIS}(1,I), \text{TDIS}(2,I), I=1, \text{NP}), \\
(\text{TDIS}(1,I), \text{TDIS}(2,I), I=1, \text{NP}), \\
(\text{TDIS}(1,I), \text{TDIS}(2,I), I=1, \text{NP}) \]

Record 3'. (4xNE words) Average element stresses

\[ ((\text{SPLT}(I,J), J=1, \text{NE}), I=1, 4) \]

Record 4'. (4xNE words) Average element strains

\[ ((\text{SPLT}(I,J), J=1, \text{NE}), I=1, 4) \]
RUBBLE.MOV

The records of this file are written to draw rubbleized groups of subelements to effectively display a failed region.

Record 1. (2 words, El5.5, I5) Increment number, TT, and total number of parts, ICNT.

Record 2. (4 words, 4I5) Number of "parts", NONE; number of nodal points, NUMNP; number of unfailed elements, NP2; number of entries in connectivity array, NCON.

Record 3. (2 words, 2I5) Beginning element number, NP1; Ending element number, NP2.

Record 4. (3xNUMNP words, 6El2.5) Nodal point coordinates (CORD(I,1), CORD(I,2), 0.0), I=1, NUMNP)

Record 5. (4xNUMEL WORDS, 16I5) Connectivity array (IXAR(I), I=1, NCON)

The above records provide geometry data for the original mesh. The following records are written, for each and every failed or rubble element, to describe the rubbleization as a "rubble" region.

Record 1'. (4 words, 4I5) The variables NONE, NJ, NPT AND NCON. These standard descriptors for the group of rubble subelements as used to describe a rubble region (see below). NONE is the number of "parts" for this group (i.e. 1), NJ is the number of nodes (i.e. 12), NPT is the number of elements (i.e. 5), and NCON is the number of entries in the connectivity (i.e. 20).

Record 2'. (2 words, 2I5) The variables NP1 and NP2 which are the smallest and largest element numbers associated with the subelement group.

Record 3'. (36 words, 6El2.5) the coordinates of the subelement group. (RN(J), ZN(J), 0.0), J=1, 12

Record 4'. (20 words, 16I5) the connectivity of the subelement group (IXN(J), J=1, 20)
SCRUBS.RST

This file is the restart file that is used to both restart from and write subsequent restart files. A file is automatically written after each converged increment. To restart a run, the following lines of SCRUBS.DAT, along with the proper SCRUBS.RST file, are all that are required.

1. Group 1, line 1.
2. Group 3, all lines

Note: Group 6, line 10 (NINC) must be changed to the number of remaining load increments.
This file contains input and output data formatted into a readable form. The input data is first written in its entirety, and includes a nodal mapping for the node numbers changed due to the slip plane addition.

After the input data is written, convergence information is output for each iteration. This information includes run times at various points in the program, and information on current loads, residuals, displacements, and plastic work. Convergence is reached when the normal of the residual sum ratio reaches the convergence percentage specified by the user. The flag NCHECK will be zero when convergence is attained.

The last portion of this file contains the output information. Output includes displacements and reactions for all boundary nodes, nodal displacements for all nodes if specified, and all stresses at each gauss point of any elements specified.

SCRUBS also have several checks for errors or incomplete solution which halt execution before the solution is obtained. For each of these an error message is output to the SCRUBS.LIS file. The following is a list of each of these stops, including a description and possible cause of the error.

NO CONVERGENCE ON INCREMENT I
Solution could not be reached in the number of iterations specified. This is usually due to too much plastic flow and can often be corrected by reducing the loads.

NEGATIVE DET STOPPED AT ELEMENT NO = I
X1,Y1= x , y
X2,Y2= x , y
X3,Y3= x , y
X4,Y4= x , y
This indicates a element is improperly deformed as shown
\[
\begin{array}{ccc}
1 & 3 & 1 \\
\hline
| & | \\
1 & | \\
\hline
| & |
\end{array}
\begin{array}{ccc}
2 & 4 & 2 \\
\hline
| & | \\
| & | \\
\hline
|
\end{array}
\]
This indicates a element is improperly deformed as shown

\[
\begin{array}{ccc}
1 & 3 & 1 \\
\hline
| & | \\
| & |
\end{array}
\begin{array}{ccc}
2 & 4 & 2 \\
\hline
| & | \\
| & |
\end{array}
\]

Usually caused by too large of deformations.

PROGRAM HALTED IN SET STIFFNESS SPACE EXCEEDED
Bandwidth is too high and stiffness dimensioning is exceeded, bandwidth must be decreased.
PROGRAM HALTED IN SOLVE
NEGATIVE OR ZERO DIAGONAL STIFFNESS
Stiffness matrix is not positive definite, error in material parameters or problem has rigid body modes.
ERROR-- FRACTURE SCPACING (DELTA) MUST BE POSITIVE & NONZERO
Fracture spacing has been input improperly

ERROR-- HALTED IN SLIPP, ERROR IN CONNECTIVITY
When slip plane is being added, improper slip node numbering leads to this error. Caused by improper mesh generation or adding of a slip plane along a free edge.

ERROR-- NODE I NOT FOUND IN CONNECTIVITY ARRAY
Program cannot find new node number of a node in the nodal mapping. Caused by error in mesh generation.

************************************************************************
************************************************************************
SUBROUTINE SLIPP

IMPLICIT DOUBLE PRECISION (A-H,O-Z)
COMMON/CONTR/TITLE(20), NP, NE, NB, NDF, NCN, NSFR, NSZF, LV, NPP, IC5, NCORD
1. NSN, NS12, NL, NBUF, NT, ND, NNP, IELD, NLID, NMAT, IT, NIT, INC, NINC, NSTORE
2. MPA, LTYPE, FACTOR, OMEGA, CONFC, NIELD, IST5, NALC0, PWORK, TFWORK, KNE
COMMON/BOUN/NBC(200), NFIX(200), U(400), ANG(200), TCR(400), US(200, 2)
COMMON/SCR/DIS(2,1000), TDIS(2,1000), DISN(2,1000), CORD(1000, 2)
1. NDF(4000), IMAT(1000), TSTS(5,4000), EPSFN(5,4000), DUMMY(1000)
COMMON/SLP/DCOR(1000,2)
DIMENSION MAP(1000,2)
INTEGER SLJP(500), CORNER(500), EL
NSP=JABS(NNP)

c save old information
NEOLD=NE
NPOLD=NP
DO 3 N=1, NP
DCOR(N,1)=CORD(N,1)
DCOR(N,2)=CORD(N,2)
3 WRITE(6,1)
1 FORMAT(/,10(' '), SLIP PANE BOUNDARY DATA ',10(' '),)
WRITE(6,2) NSP
2 FORMAT(/, ' NUMBER OF SLIP PLANES = ',12)

loop on number of slip planes
DO 1000 INS=1,NSP
read in slip plane data
READ(5,4) NSLIP, IANG, DELTA
4 FORMAT(2I5,G10.3)
K=(NSLIP-1)/16.+1
J=1
DO 10 I=1,K
JJ=J+15
IF(JJ.GE.NSLIP)JJ=NSLIP
READ(5,5) (SLIPIN(J,J),J=J, JJ)
5 FORMAT(16I5)
10 IF(Delta L.E.0.Delta A=(DABS(CORD(1,1)-CORD(2,1))+DABS(CORD(1,2)-
+CORD(2,2)))/50

***** change slip node array to new node numbers
IF(INSP.LE.1) GOTO 15
DO 13 N=1, NSLIP
13 CALL NEWNOD(SLIP(N,1))
15 CONTINUE
L=4*NSFR
NECN=NCN+NE

loop on number of slip plane nodes
DO 160 I=1, NSLIP
NADD = 1
IX = SLIP(1, 1)
SLIP(1, 2) = 0
CORDN = CORD(IX, IANG) + DELTA

determine if node is a corner node
CALL CORNN(NADD, NSFR, NP, SLIP(1, 2), IX)

add NADD to all numbers in connectivity greater than IX

DO 60 J = 1, NECN
   IX = NDF(J) - IX
   IF (IX) 60, 30, 50
   EL = (J + L - 1) / L
   X = 0.0
   MMM = L * (EL - 1)
   DO 40 M = 1, L, NSFR
      MM = MMM + M
      IF (X .LT. CORDN) GOTO 60
      NDP(J) = NDP(J) + NADD
   40 CONTINUE

update remaining slip plane node numbers

DO 70 N = 1, NSLIP
   IF (SLIP(N, 1) .GT. SLIP(1, 1)) SLIP(N, 1) = SLIP(N, 1) + NADD

update coordinate array numbers (add NADD to those < IX)

DO 90 J = NP, IX, -1
   IJ = J + NADD
   CORD(IJ, 1) = CORD(J, 1)
   CORD(IJ, 2) = CORD(J, 2)

assign coordinate to new node created

CORD(IJ, IANG) = CORDN
NP = NP + NADD
NUMDUM = 0
JJ = NB

update boundary arrays, giving new nodes same boundary conditions
as corresponding slip plane nodes

DO 150 JJ = 1, JJ
   J = J + NUMDUM
   IXX = NBC(J) - IX
   IF (IX) 150, 100, 140
100
   DO 150 IJ = 1, NADD
      NE = NE + 1
      IJ = IJ + I
      DO 120 N = NB, IJ, -1
         NBC(N) = NBC(N - 1)
         NFIX(N) = NFIX(N - 1)
      120
      DO 110 N = 1, NDF
         US(N, NN) = US((N - 1), NN)
110
   ANG(N) = ANG(N - 1)
130
   NBC(IJ) = NBC(IJ) + 1
ADD ROW OF SLIP ELEMENTS TO CONNECTIVITY

set up array of corner nodes

```
NCORN=0
DO 200 I = 1, NSLIP
IF(SLIP(I,2).EQ.0)GOTO 200
NCORN=NCORN+1
CORNER(NCORN)=SLIP(I,1)
200 CONTINUE
```  

find corner node in connectivity array

```
J=0
210 J=J+1
IF(J.GT.NENT)GOTO 300
IF(NOP(J).NE.CORNER(I))GOTO 210
```  

determine element number of element containing node

```
EL=(J+L-1)/L
```  

find other corner node contained in same element

```
MMM=L*(EL-1)
DO 220 M=1, L, NSFR
MM=MMM+M
DO 220 N=1, NCORN
IF(N.EQ.1)GOTO 220
IF(NOP(MM).EQ.CORNER(N))K=N
220 CONTINUE
230 CONTINUE
```  

print error if no such node exists

```
IF(K.GT.0)GOTO 240
WRITE(6,235)
235 FORMAT('ERROR--HALTED IN SLIPP, ERROR IN CONNECTIVITY')
STOP
```  

make new element

```
240 IF(K.GE.KK)GOTO 210
CALL MAKEEL(MMM, CORNER(I), CORNER(K), IANG)
```  

check to see how many elements are made

```
IQUIT=IQUIT+1
IF(IQUIT.GE.NCORN)GOTO 400
KK=1
```
IF (I.EQ. 1) KKK = K
J = K

branch to next element
GOTO 205

300 I = 1
KK = KKK
GOTO 205

***** write slip plane data *****

CONTINUE
WRITE (6, 500) IINS, DELTA, IANC
500 FORMAT (/,' SLIP PLANE ', 12,
+ /,' THICKNESS=', .10, 3,
+ /,' ANGLE FLAG=', .12,
+ /,' NODE LIST:' )
KNUM = (NSLIP - 1) / 16 + 1
KSTART = 1
DO 800 ITT = 1, KNUM
KEND = KSTART + 15
IF (KEND .GT. NSLIP) KEND = NSLIP
WRITE (6, 600) (SLIP (J, 1), J = KSTART, KEND)
600 FORMAT (16I5)
800 KSTART = KSTART + 16
1000 CONTINUE

***** write mapping of old to new nodes into scrubs list *****

J = 0
DO 2000 I = 1, NPOLD
N = I
CALL NEWNOD (N)
IF (N.EQ. 1) GOTO 2000
J = J + 1
MAP (J, 1) = I
MAP (J, 2) = N
2000 CONTINUE
WRITE (6, 3000)
3000 FORMAT (/,' ***** NODE NUMBERS WERE CHANGED DUE TO SLIP PLANE
+ADDITION *****', /,' NODAL MAPPING OF NODES CHANGED: ',
+ '4X, 4(' 'ORIGINAL NEW ' '), 5X, 4(' 'NODE# NODE# ' '))
K = J / 4
KK = K * 4 - J
K1 = 0
IF (KK .LE. -1) K1 = 1
K2 = K1
IF (KK .LE. -2) K2 = K2 + 1
K3 = K2
IF (KK .LE. -3) K3 = K3 + 1
DO 5000 M = 1, K
WRITE (6, 4000) MAP (M, 1), MAP (M, 2), MAP (M + K + K1, 1), MAP (M + K + K1, 2),
+ MAP (M + K + K2, 1), MAP (M + K + K2, 2), MAP (M + K + K3, 1), MAP (M + K + K3, 2)
4000 FORMAT (13X, 4(13, 6X, 13, 7X))
5000 CONTINUE
WRITE (6, 4000) (MAP (N, 1), MAP (N, 2), N = K + 1, K3 * (K + 1), K + 1)
WRITE (6, 6000)
6000 FORMAT (/,' 'SLIP PLANE ELEMENTS CREATED: ', 7X, 'ELEMENT #'
+, 6X, 'CONNECTIVITY',/)
ISTART=L*NEOLD+1
DO 8000 N=NEOLD+1,NE
IEND=ISTART+L-1
WRITE(6,7000) N,(NOP(I),I=ISTART,IEND)
7000 FORMAT(10X,13,6X,12I5)
8000 ISTART=IEND+1

numbrr new midside nodrs

N=NEOLD+1
IF(NSFR.GT.1)CALL NODEXY(N)
RETURN
END

SUBROUTINE MAKEEL(MMM, I, K, IANC)
IMPLICIT DOUBLE PRECISION (A-H, O-Z)
COMMON/CONTR/TITLE(20), NF, NE, NB, NDF, NCN, NSFR, NE2F, LV, NFP, ICS, NCORD
1. NSK, NSIZ, NL, NBUF, NT, ND, NNP, ILD, NLD, NMAT, IT, NIT, INC, NINC, NSTORE
2. MPR, LDTYP, FACTOR, OMEGA, CONFA, NYIELD, ISTS, NALGO, PWORK, TPWORK, KNE
COMMON/SCR/DIS(2,1000), TDIS(2,1000), DISX(2,1000), CORD(1000,2)
1. NOP(4000), IMAT(1000), ISTS(5,4000), EPSTN(5,4000), DUMMY(1000)

this subroutine adds a new element to the connectivity containing
corner nodes I and K.

ISTART=I
IF(IANG.EQ.2)GOTO 100
IF(CORD(I,2).LT.CORD(K,2))JSTART=K
GOTO 110
100 IF(CORD(I,1).GT.CORD(K,1))JSTART=K
110 CONTINUE
L=NSFR*4
DO 120 M=1,L,NSFR

MM=MM+M
IF(NOP(MM),NE,ISTART)GOTO 120
JSTART=MM
IFLAG=M
120 CONTINUE
NEW=NE+L+1
NOP(NEW)=NOP(JSTART)
DO 140 N=1,NSFR

NEW=NEW+1
IF(IFLAG.EQ.1)JSTART=JSTART+L
JSTART=JSTART-1
140 NOP(NEW)=NOP(JSTART)
DO 150 N=1,NSFR
NOP(NEW+1)=NOP(NEW)+1
150 NEW=NEW+1
IX=NSFR-1
DO 160 N=1,IX

NEW=NEW+N
NOP(NEW+N)=NOP(NEW-NSFR-N)+1
NEW=NEW+1
IX=IX-1
160 
DO 170 N=NSFR,1,-1
NEW=NEW+1
170 NOP(NEW)=JSTART+N
NE=NE+1
IMAT(NE)=NMAT-NL
RETURN
END
SUBROUTINE CORRN(NADD, NSFR, NP, IFLAG, IX)
IMPLICIT DOUBLE PRECISION (A-H, O-Z)
COMMON/SBR/DIS(2,1000), TDIS(2,1000), DISN(2,1000), CORD(1000,2)
1. NDF(4000), IMAT(1000), TSTE(5,4000), EPSN(5,4000), DUMMY(1000)
REAL CORD

this subroutine determines if node IX is a corner node

J=0
100 J=J+1
IF (J GT 4000) THEN
WRITE(6,200) IX
200 FORMAT('ERROR -- NODE ', 13, ' NOT FOUND IN CONNECTIVITY ARRAY')
STOP
ELSE
IF(NDF(J) NE IX) GOTO 100
CORD=FLOAT(J-1)/FLOAT(NSFR)
IF(CORD NE 0.0) RETURN
NADD=NSFR
IF it is a corner node, IFLAG = 1
IFLAG=1
RETURN
ENDIF
END

SUBROUTINE NEWNODE(N)
IMPLICIT DOUBLE PRECISION (A-H, O-Z)
COMMON/CONT/ TITLE(20), NP, NE, NB, NDF, NCN, NSFR, NSZF, LV, NPP, ICS, NORD
1. NSK, NSIZ, NL, NBUF, NT, NO, NNP, ILD, NLD, NMAT, IT, NIT, INC, NINC, NSTORE
2. MPF, LDTYPE, FACTOR, OMEGA, CONFC, NYIELD, ISTS, NALGO, PWORK, TPWORK, KNE
COMMON/SBR/DIS(2,1000), TDIS(2,1000), DISN(2,1000), CORD(1000,2)
1. NDF(4000), IMAT(1000), TSTE(5,4000), EPSN(5,4000), DUMMY(1000)
COMMON/SLP/...

this subroutine finds the new node number of node N

K=0
100 IF(DCORD(N,1).EQ.CORD(1,1).AND. DCORD(N,2).EQ.CORD(1,2)) K=1
N=K
IF(K NE 0) RETURN
WRITE(6,200) N
200 FORMAT('***** ERROR IN MAPPING: NO NEW NODE FOUND FOR OLD NODE
+13.*****')
STOP
END
APPENDIX D

JOINTED MEDIA CODE LISTING
SUBROUTINE JOINTM(L, STRN, STRS, MOSH, YIELD, PSTRN)
IMPLICIT DOUBLE PRECISION (A-H, O-Z)
DOUBLE PRECISION K, MU, LAMB
COMMON/CONTR/TITLE(20), NP, NE, NB, NDF, NCSF, NSIF, L, NPP, ICE, NCORD
1. NSK, NS2, NL, NBUF, NT, ND, NDF, ILD, NL, NMAT, IT, NIT, INC, NINC, NSTORE,
2. MPK, LDTY, FACTOR, OMEGA, CONFACT, NYIELD, ISTS, NAPOD, PWORK, TPWORK
COMMON/GEP/FAC(40), NSD(40), OUT(10), NDFR(30), IOLARY(1000),
COMMON/SCR/DIS(2,1000), TDIS(2,1000), DIS(2,1000), CORD(1000,2)
1. ND(4000), IMAT(1000), TSTS(5,4000), EPIST(5,4000)
2. SIG(4), PSI(4), DSIG(4), EP(4), NG(16), NGSTP(16)
DIMENSION STRN(4), STRS(4), T(4), TT(4), E(4)

Assign material parameters

K=-ORT(L,1)
G=-ORT(L,2)
C0=ORT(L,3)
MU=ORT(L,4)
TH=ORT(L,5)
FAIL=ORT(L,7)
DEL=ORT(L,9)
GS=ORT(L,10)

check for delta and GS equal to zero

IFGS EG . 0.
IFDEL GT . 0. GOTO 5
WRITE16 . 61
FORMAT(1. ERROR-- FRACTURE SPACING (DELTA) MUST BE
& POSITIVE & NONZERO')
STOP

rotate incremental strain vector to local system

5 CONTINUE
STRN(3)=STRN(3)/2
CALL ROTATE(STRN,E,TH)

get total stress vector and rotate

DO 6 I=1,4
STRS(1)=TSTS(1,MOSH)
6 CONTINUE
CALL ROTATE(STRS,TT,TH)

solve for incremental stresses

TRTDDOT=(K-2.*G/3.)*(E(1)+E(2)+E(4))
T(1)=2.*G+E(1)+TRTDDOT
T(2)=2.*G+E(2)+TRTDDOT
T2TOT=TSTS(2,MOSH)+T(2)

test for tensile failure of crack

IF(T2TOT GT FAIL) THEN
WRITE(6,'(6.2)') 'FAILURE STRESS= ',T2TOT
TSTS(2,MOSH)=FAIL
T(2)=0.0
ENDIF
T(3)=2.*G+T(3)/(1.+G/DEL/GS)
T(4)=2.*G+E(4)+TRTDDOT
check for plastic slip

\[ F = \text{DABS}(T(3) + T(2)) \times MU \times (TT(2) + T(2)) - CD \]

IF \( F \leq 0 \) GOTO 20
\[ XX = (2 \times C / (1 + G/DEL/CS)) \]
\[ Y = XX \times E(3) + TT(3) \]

solve for plastic strain

IF \( Y \leq 0 \) \( \text{LAMB} = E(3) + (CO - MU \times (TT(2) + T(2)) + TT(3)) / XX \)
IF \( Y > 0 \) \( \text{LAMB} = E(3) - (CO - MU \times (TT(2) + T(2)) - TT(3)) / XX \)

determine shear stress for plastic condition

CONTINUE
\[ \text{ELAS} = E(3) - \text{LAMB} \]
\[ T(3) = 2 \times G + \text{ELAS} / (1 + G/DEL/CS) \]
\[ \text{EPSTN}(A, MSH) = \text{EPSTN}(A, MSH) + \text{LAMB} \]

CONTINUE
\[ \text{YIELD} = \text{DABS}(T(2) + T(2)) \times MU \]

rotate incremental stress matrix to global system

CALL ROTATE(T, STRS, TH)
\[ \text{STRN}(3) = \text{STRN}(3) \times 2 \]
RETURN

SUBROUTINE ROTATE(A, B, THETA)
IMPLICIT DOUBLE PRECISION (A-H, O-Z)
DIMENSION A(4), B(4)

rotate 2D stress or strain vector by angle THETA

C = DCOS(THETA)
S = DSIN(THETA)

B(1) = A(1) \times C + A(2) \times S - 2 \times A(3) \times C + S
B(2) = A(1) \times S - S + A(2) \times C + 2 \times A(3) \times C + S
B(3) = A(1) \times C \times A(2) \times C + S + A(3) \times C + A(3) \times S + S
B(4) = A(4)
RETURN
END
TWO DIMENSIONAL COMPUTER MODELING OF JOINTS 
AND FRACTURES IN CONTINUA

Royd R. Nelson
Department of Civil Engineering
M.S. Degree, December 1983

ABSTRACT

This thesis presents a numerical formulation which incorporates the effects of joints, cracks, and fractures into a mathematical mode used in an existing finite element code. Two separate crack definitions were incorporated, one involving a discrete crack model, and the other a continuum approach defining a new material with regularly spaced joints. Several examples are also presented demonstrating the use of this capability.

COMMITTEE APPROVAL:

Steven E. Benzley
Committee Chairman

S. Otani Durrant
Committee Member

Henry W. Christiansen
Department Chairman