

IC 9322

BUREAU OF MINES
INFORMATION CIRCULAR/1992

MULSIM/NL Application and Practitioner's Manual

By R. Karl Zipf, Jr.



UNITED STATES DEPARTMENT OF THE INTERIOR

Information Circular 9322

MULSIM/NL Application and Practitioner's Manual

By R. Karl Zipf, Jr.

UNITED STATES DEPARTMENT OF THE INTERIOR
Manuel Lujan, Jr., Secretary

BUREAU OF MINES
T S Ary, Director

Library of Congress Cataloging in Publication Data:

Zipf, R. Karl.

MULSIM/NL application and practitioner's manual / by R. Karl Zipf, Jr.

p. cm. — (Information circular; 9322)

Includes bibliographical references (p. 45).

Supt. of Docs. no.: I 28.27:9322.

1. MULSIM/NL. 2. Ground control (Mining)—Computer simulation. I. Title.
- II. Series: Information circular (United States. Bureau of Mines); 9322.

TN295.U4 [TN288] 622 s—dc20 [622'.28'0113] 91-47134 CIP

CONTENTS

	<i>Page</i>
Abstract	1
Introduction	2
Objectives	2
MULSIM/NL overview	2
Scope of manual	2
MULSIM/NL capabilities	2
Nonlinear material properties	4
Multiple mining steps	4
Energy release calculations	4
MULSIM/NL modeling	6
Idealized mine plan	6
Coarse-mesh generation	6
Fine-mesh generation	6
Input file structure for MULSIM/NL	8
Input file sections	8
Units	10
Global coordinate system	11
Control parameter definition	12
Record type 1—Single record—format (20A4)	12
Record type 2—Single record—format (F8.2, E12.6, I8)	12
Record type 3—Single record—format (I8)	12
Record type 4—NMATS records—format (10E8.0)	12
Linear elastic for coal	13
Linear elastic for gob	13
Strain softening for coal	13
Elastic plastic for coal	13
Bilinear hardening for gob	13
Strain hardening for gob	14
Record type 5—Single record—format (6(F6.0, F6.4))	15
Record type 6—Single record—format (F8.2, 2I8, 16X, 4I8)	15
Record type 7—NSEAM records—format (4F8.1)	15
Record type 8—Single record—format (9F8.5)	16
Record type 9—Single record—format (2F8.2, I8, 24X, 2I8)	16
Input summary	17
Input file generation with preprocessor MULPRE/NL	17
Running MULSIM/NL	18
MULSIM/NL files	18
MULSIM/NL execution command sequences	18
MULSIM/NL running times	19
Output files from MULSIM/NL	20
Printout file	20
Fine- and coarse-mesh output data files	20
Output file examination with postprocessor MULPLT/NL	20
MULSIM/NL practical examples	21
Longwall mining example	21
Multiple-seam mining example	22
Random room-and-pillar mining example	32
Summary	45
References	45
Appendix.—Special instructions for personal computer version of MULSIM/NL	46

ILLUSTRATIONS

	<i>Page</i>
1. Typical mine layout and boundary element method model	3
2. Stress-strain models for MULSIM/NL	4
3. Strain energy relations for various linear and nonlinear backfill or gob behaviors	5
4. Strain energy relationships for linear and nonlinear material models in MULSIM/NL	6
5. Coarse-mesh generation in basic mine plan	7
6. Fine-mesh generation in basic mine plan	7
7. MULSIM/NL input file structure for single-seam, single-step problem	8
8. MULSIM/NL input file for sample problem shown in figure 1	9
9. MULSIM/NL input file structure for single-seam, multiple-step problem	10
10. MULSIM/NL input file structure for multiple-seam, single-step problem	10
11. MULSIM/NL input file structure for multiple-seam, multiple-step problem	10
12. Options for global coordinate system location	11
13. Six different stress-strain models available in MULSIM/NL	12
14. Typical direction cosine matrices for orienting local coordinate system relative to global coordinate system	16
15. Overrelaxation factor versus computer time and number of iterations per step	17
16. MULSIM/NL execution command sequence for Hewlett Packard engineering workstation using UNIX operating system	18
17. MULSIM/NL execution command sequence for VAX minicomputer using VMS operating system	19
18. Execution time on SUN III workstation versus problem size	19
19. Control section of input file for longwall example	22
20. Step 1 fine-mesh section of input file for longwall example	23
21. Step 1 coarse-mesh section of input file for longwall example	24
22. Step 2 fine-mesh section of input file for longwall example	25
23. Step 2 coarse-mesh section of input file for longwall example	26
24. Longwall example—normal stresses and normal displacements around headgate during mining step 2	27
25. Longwall example—normal stresses along cross section just ahead of face during mining step 2	28
26. Longwall example—normal displacements along cross section just behind face during mining step 2	29
27. Control section of input file for multiple-seam mining example	30
28. Seam 1 (upper seam) fine-mesh section of input file for multiple-seam mining example	31
29. Seam 2 (lower seam) fine-mesh section of input file for multiple-seam mining example	33
30. Seam 1 (upper seam) coarse-mesh section of input file for multiple-seam mining example	34
31. Seam 2 (lower seam) coarse-mesh section of input file for multiple-seam mining example	34
32. Multiple-seam mining example—normal stresses	35
33. Multiple-seam mining example—normal displacements	36
34. Control section of input file for random room-and-pillar mining example	37
35. Step 1—Fine-mesh section of input file for random room-and-pillar mining example	38
36. Step 1—Coarse-mesh section of input file for random room-and-pillar mining example	39
37. Step 2—Fine-mesh section of input file for random room-and-pillar mining example	40
38. Step 2—Coarse-mesh section of input file for random room-and-pillar mining example	41
39. Step 3—Fine-mesh section of input file for random room-and-pillar mining example	42
40. Step 3—Coarse-mesh section of input file for random room-and-pillar mining example	43
41. Random room-and-pillar mining example—normal stresses and normal displacements during mining step 2	44

TABLES

1. MULSIM/NL input and output units for three typical consistent unit systems	11
2. Material model parameters and material property array EPROP structure	14
3. MULSIM/NL files	18
4. First record variables of coarse- and fine-mesh output data files	20
5. Variables in coarse- and fine-mesh output data file records	21

UNIT OF MEASURE ABBREVIATIONS USED IN THIS REPORT

ft	foot	lb·in/in ³	pound inch per cubic inch
h	hour	m	meter
in	inch	Mb	megabyte
kb	kilobyte	min	minute
lb	pound	MJ	megajoule
lb/ft ²	pound per square foot	MJ/m ²	megajoule per square meter
lb·ft	pound foot	MJ/m ³	megajoule per cubic meter
lb·ft/ft ²	pound foot per square foot	MPa	megapascal
lb·ft/ft ³	pound foot per cubic foot	MPa/m	megapascal per meter
lb/in ²	pound per square inch	psi	pound per square inch
lb·in	pound inch	psi/ft	pound per square inch per foot
lb·in/in ²	pound inch per square inch	psi/in	pound per square inch per inch

Disclaimer of Liability

The U.S. Bureau of Mines expressly declares that there are no warranties expressed or implied that apply to the software contained (or described) herein. By acceptance and use of said software, which is conveyed to the user without consideration by the Bureau of Mines, the user hereof expressly waives any and all claims for damage and/or suits including special, consequential, or other similar damages arising out of or in any way connected with the use of the software contained (or described) herein.

MULSIM/NL APPLICATION AND PRACTITIONER'S MANUAL

By R. Karl Zipt, Jr.¹

ABSTRACT

MULSIM/NL (multiple seams, nonlinear) is a new U.S. Bureau of Mines boundary-element-method (BEM) program for calculating stresses and displacements (i.e., convergence) in coal mines or thin, tabular metalliferous veins. This manual gives detailed operating instructions for MULSIM/NL and illustrates its use with several practical examples. While this manual concentrates on the practical aspects of actually running and using MULSIM/NL, another companion report titled "MULSIM/NL—Theoretical and Programmer's Manual" provides mathematical and programming details to those engineers and programmers who need to fully understand the FORTRAN program or desire to alter and enhance it.

MULSIM/NL analyzes one to four parallel seams that have any orientation with respect to the Earth's surface. Three main features distinguish MULSIM/NL from its predecessors: (1) nonlinear material models, (2) multiple mining steps, and (3) comprehensive energy release and strain energy computations. MULSIM/NL has six material models for the in-seam material including (1) linear elastic for coal, (2) strain softening, (3) elastic plastic, (4) bilinear hardening, (5) strain hardening, and (6) linear elastic for gob. The multiple mining step capability enables the user to simulate a changing mine geometry. Finally, MULSIM/NL performs comprehensive energy release rate calculations.

¹Structural engineer, Denver Research Center, U.S. Bureau of Mines, Denver, CO.

INTRODUCTION

OBJECTIVES

The program described in this U.S. Bureau of Mines report is part of the MULSIM/NL package, which features the actual BEM program described herein, as well as a preprocessor program called MULPRE/NL and a plotting postprocessor program called MULPLT/NL. The preprocessor helps the user generate the requisite input file for the main program MULSIM/NL, whereas the postprocessor assists the user in examining the calculated stresses and displacements in a very rapid graphical manner.

Documentation and instructions for the MULSIM/NL package are divided into two related reports with each providing essential elements toward an understanding of the whole. *The objective* of this report "MULSIM/NL—Application and Practitioner's Manual" is to provide users with detailed operating instructions for MULSIM/NL along with several practical examples to better illustrate the capabilities of the program. A related report "MULSIM/NL—Theoretical and Programmer's Manual" provides certain mathematical and programming details to those engineers and programmers who need to fully understand the FORTRAN program or desire to alter and enhance it.

MULSIM/NL OVERVIEW

MULSIM/NL calculates stresses and displacements throughout a coarse- and fine-mesh modeling grid used to approximate an actual mining geometry. It can analyze one to four parallel seams having any orientation with respect to the Earth's surface. Topographic or free surface effects are neglected. MULSIM/NL uses a coarse-mesh grid (up to 50 by 50) with an embedded fine-mesh grid (up to 150 by 150) for greater computational detail in

important regions. The material comprising each coarse-mesh block or fine-mesh element can follow any one of six linear and nonlinear stress-strain relationships. MULSIM/NL also performs comprehensive energy release rate (ERR) calculations for the model based on Salamon's (1)² theoretical work. In addition, MULSIM/NL features a multiple mining step capability. This feature allows the user to simulate the various temporal stages of mine development and examine stress and displacement changes as the mine advances. These three new capabilities, namely nonlinear material models, energy calculations, and multiple mining steps, set MULSIM/NL apart from its predecessors, MULSIM/BM, developed by Beckett and Madrid (2), and the original MULSIM created by Sinha (3).

SCOPE OF MANUAL

This manual provides detailed operating instructions for MULSIM/NL beginning with the basics of the modeling process and followed by a full discussion of the input file giving the meaning, allowable range, and format for each variable in that file. Basic instructions for operating the preprocessor MULPRE/NL are also included. This preprocessor helps the user create a MULSIM/NL input file in a user-friendly, graphical manner. The next discussion focuses on actually running MULSIM/NL by giving the required command sequences for various computer systems. Subsequent discussions describe the output files, which include a print file, a coarse-mesh data file, and a fine-mesh data file. The postprocessor MULPLT/NL, described next, aids the user in examining these voluminous data files in a very simple manner using very fast graphics routines. Finally, the manual culminates with several complete examples, including sample input files and postprocessor plots.

MULSIM/NL CAPABILITIES

MULSIM/NL is a BEM program that calculates three-dimensional stresses and displacements caused by mining tabular deposits like coal seams. The BEM model used for these computations can have one to four parallel seams at any orientation in the earth. Likewise, the virgin stress field can also have any orientation. MULSIM/NL assumes that these seams are far below the Earth's surface, hence the program does not include the effects of complex topography.

MULSIM/NL uses a coarse- and fine-mesh modeling grid to approximate the actual in-seam coal mine geometry. Figure 1 shows a typical coal mine geometry and

illustrates its connection to a MULSIM/NL grid. In this analysis problem, retreat mining methods will extract the central pillar in this layout. The coarse-mesh blocks must cover a sufficiently large section of the actual mine layout, whereas fine-mesh elements will cover a central region of interest where greater computational detail is desired. MULSIM/NL allows up to a 50 by 50 coarse-block array and up to a 150 by 150 fine-mesh array.

²Italic numbers in parentheses refer to items in the list of references preceding the appendix at the end of this report.

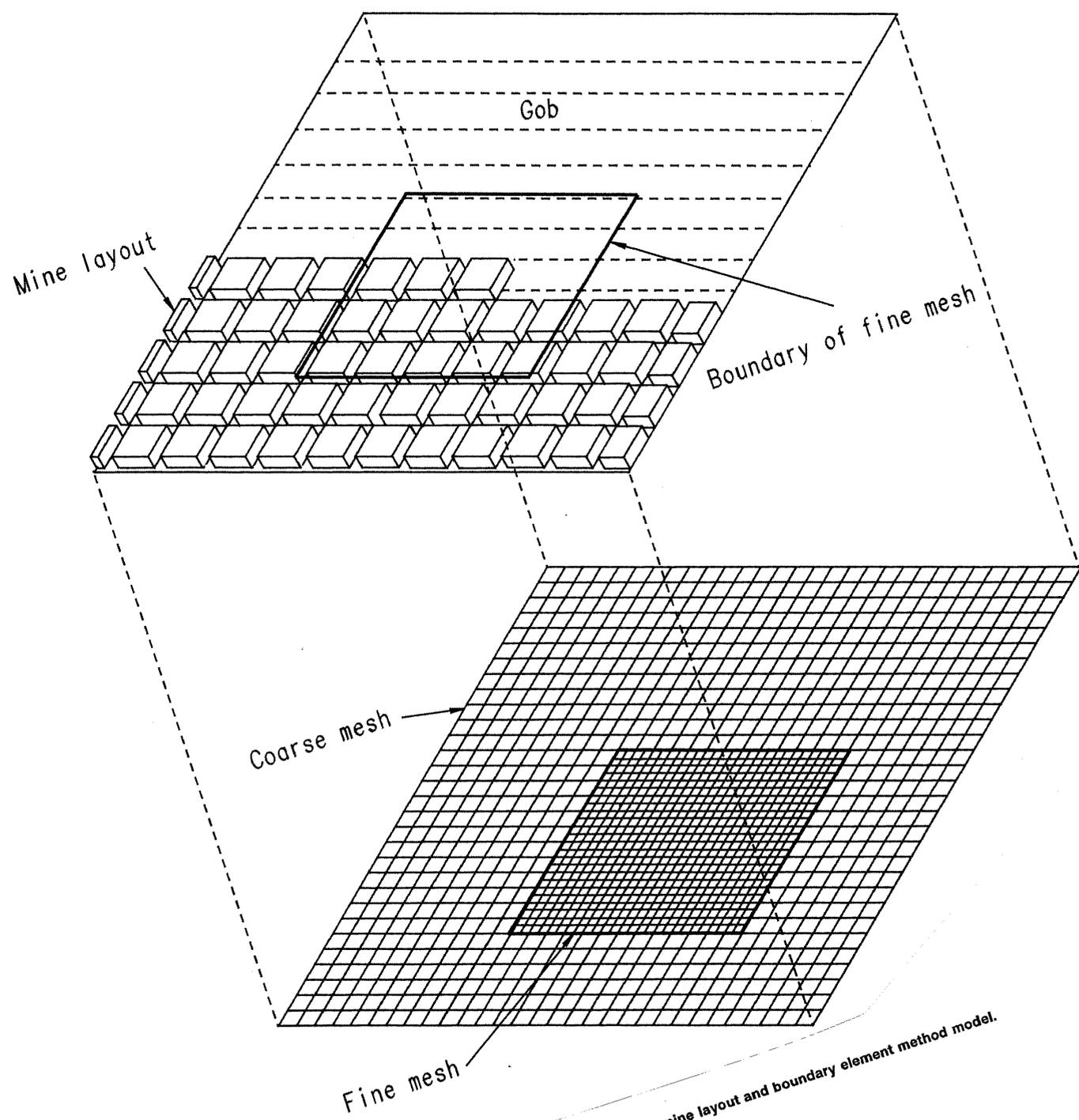


Figure 1.—Typical mine layout and boundary element method model.

NONLINEAR MATERIAL PROPERTIES

As with most BEM programs, a linear elastic rock mass surrounds the seams; however, MULSIM/NL now permits various nonlinear material models for the in-seam block and element materials. Prior versions of MULSIM (2-3) permitted linear stress-strain relations only for in-seam materials, such as coal or gob. As shown in figure 2, MULSIM/NL now has six material models from which to choose including (1) linear elastic for coal, (2) strain softening, (3) elastic plastic, (4) bilinear hardening, (5) strain hardening, and (6) linear elastic for gob. The first three are intended for the unmined in-seam coal material, while the latter are for the broken gob material left in the wake of full extraction mining. Model 2 (strain softening) after Crouch and Fairhurst (4) approximates the yielding behavior of small-pillars or large-pillar perimeters, while model 3 (elastic plastic) approximates a pseudoductile behavior in pillar cores (5). Model 5 (strain hardening) allows the gob material to increase in stiffness as it consolidates under increasing load. Model 4 (bilinear hardening) permits a certain amount of deformation to occur prior to introducing stiffness. Models 1 and 6 are basic linear elastic models for coal and gob, respectively.

MULTIPLE MINING STEPS

MULSIM/NL features multiple mining steps to simulate various stages of mine development. This feature enables the user to examine stress and displacement *changes* as the mine development advances or retreats. Such changes more readily compare with field measurement data that tend to measure stress and displacement (convergence) changes as opposed to *total* or absolute stresses and displacements.

ENERGY RELEASE CALCULATIONS

Most important, MULSIM/NL contains an energy subroutine that uses the calculated stresses and displacements at each block and element to evaluate detailed energy changes for each mining step. This subroutine computes various ERR quantities for the entire model, as well as various strain energy values for each element. Cook (6) developed the original ERR concept, and research found that ERR correlated well with the incidence of devastating rock bursts in deep underground South African gold mines. By analogy, it is hypothesized that the ERR will also correlate with the incidence of coal mine bumps.

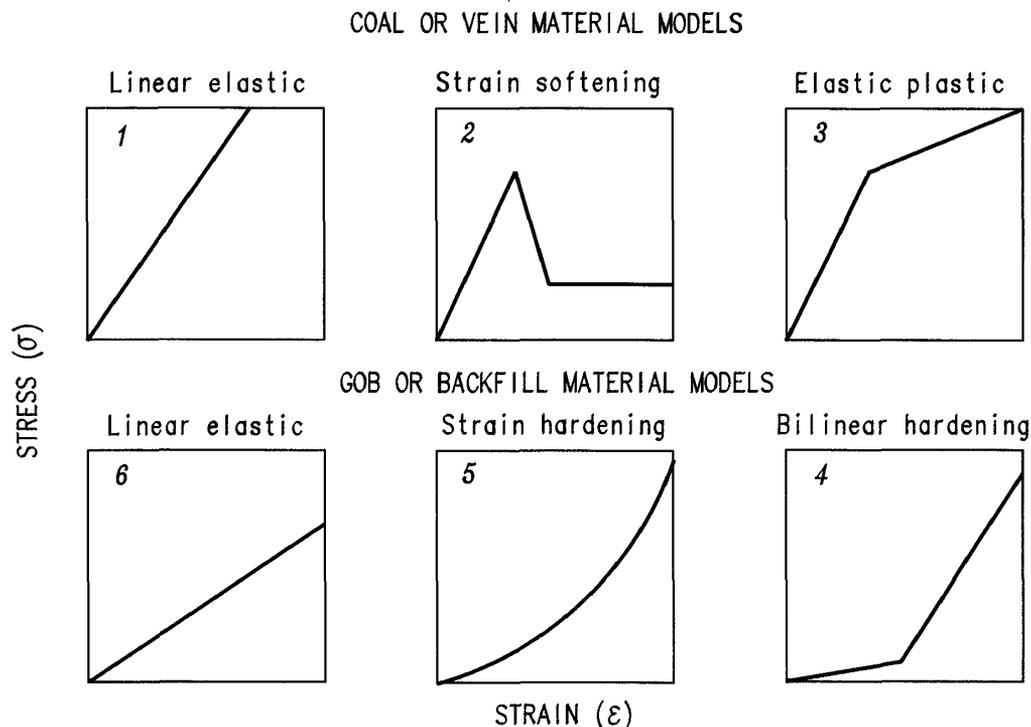


Figure 2.—Stress-strain models for MULSIM/NL.

Salamon (1) clarified the original ERR concept and derived the following relationship implemented in MULSIM/NL:

$$W_R = \frac{1}{2} \int_{S_M} T_I u_I ds + \frac{1}{2} \int_{S_M} T_I \Delta u ds + \frac{1}{2} \int_{S_{GH}} (1 - \alpha) \Delta R \Delta u ds, \quad (1)$$

where W_R = total energy release during current step,

S_M = area mined during current step,

S_{GH} = total backfill or gob area,

T_I = total stress in coal during prior step,

ΔR = stress change in backfill during current step,

u_I = total displacement during prior step,

Δu = displacement change during current step,

and α = nonlinearity factor.

In this relation, Δ refers to the change in a field quantity between mining steps. The first term represents the strain energy release from the mined-out material.

The second term, called the linear kinetic energy release, is the change in gravitational potential over the area mined during this step. The third term, called the nonlinear kinetic energy release, is an additional energy release (or sink) arising in the backfill or gob area. This term contains a so-called nonlinearity factor α . Figure 3 illustrates the meaning of α and shows how it accounts for the degree of nonlinearity. Together, the second and third terms of equation 1 form the kinetic energy release. The strain energy release, kinetic energy release, and total energy release terms are computed and written in the MULSIM/NL output files for all elements within the areas S_M and S_{GH} . Summing up the elemental energy releases (i.e., carrying out the integrations prescribed by equation 1) gives the total energy releases for the model during the current mining step.

The ERR calculations within MULSIM/NL depart slightly from the original prescription of Salamon (1). MULSIM/NL evaluates the strain energy release (i.e., the first term of equation 1) as a "recoverable strain energy" as shown in figure 4. These calculations assume that the unloading modulus from the state (T, u) equals the initial loading modulus. In addition to evaluating recoverable strain energy as the strain energy release over the current mined area S_M , MULSIM/NL also evaluates recoverable, dissipated, and total strain energy for each unmined seam element with the same scheme as shown in figure 4. The models shown in figure 4 apply to the unmined seam material, either coal or rock. The output data files from MULSIM/NL include these strain energies along with the stresses and displacements.

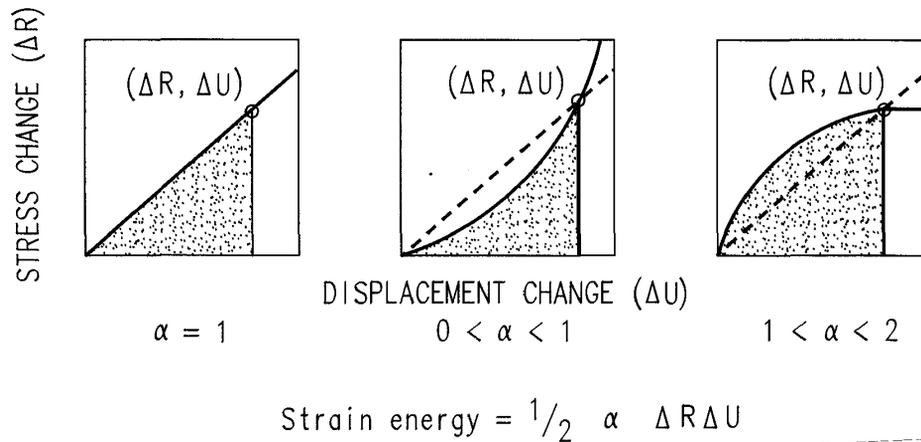


Figure 3.—Strain energy relations for various linear and nonlinear backfill or gob behaviors.

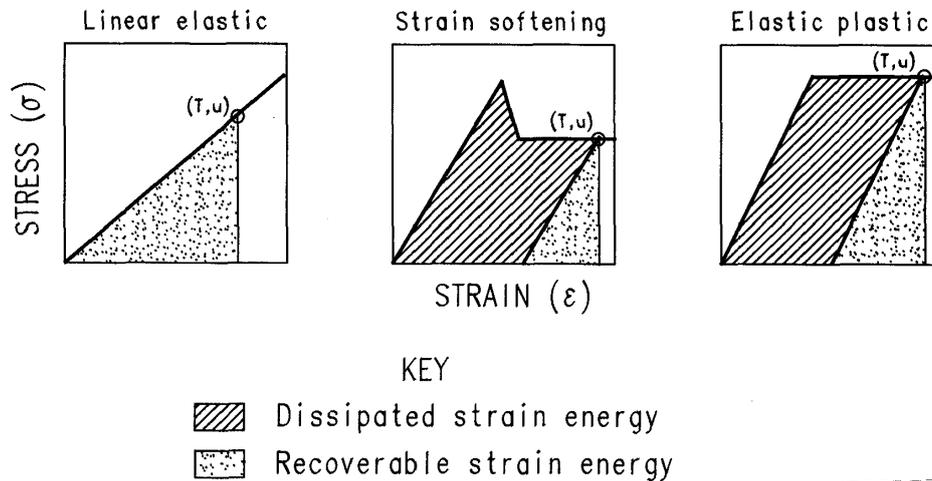


Figure 4.—Strain energy relationships for linear and nonlinear material models in MULSIM/NL. (T,u) is stress-strain state in material.

MULSIM/NL MODELING

Knowing something about basic BEM model generation will help the novice user understand the input file structure for MULSIM/NL. BEM modeling starts with an idealized in-seam mine plan, then overlays a grid work of elements, and finally assigns material properties to those elements. This array of material property assignments delivers the basic model geometry to the BEM program. Coupling this geometry with material property data, in situ stress data, and other program controls then forms a basic input file ready for solution by MULSIM/NL. The following figures will illustrate this modeling process for the new user.

IDEALIZED MINE PLAN

Modeling begins with an idealized mine plan such as the example shown in figure 5 (top). This plan shows part of a retreating room-and-pillar panel. Pillars are 9.1 by 10.7 m with 6.1-m opening widths. The upper area is mined out and filled with gob, whereas the lower area is approximately 50% extracted. Mining is about to extract the point pillar in the area of interest.

COARSE-MESH GENERATION

The first important decision for constructing a MULSIM/NL model concerns the fine-mesh element width, which is always one-fifth the coarse-mesh block width. Typical coal mine models might fix the element width at a quarter, third, half, or else equal to the entry width. This example uses a 1.52-m element size, which is one-quarter of the entry width. Accordingly, the block

width is 7.62 m. (The term block used herein refers to a modeling block and not to a coal pillar or coal block.) Figure 5 (middle) shows the basic mine plan again, but now a 7.62-m coarse-mesh block grid is superimposed. That grid consists of an array 12 blocks long in the x direction and 11 blocks wide in the y direction. In addition, a fine-mesh boundary is defined around the area of interest. Coarse-mesh blocks that will be divided into fine-mesh elements extend from four to nine in the x direction and from four to eight in the y direction. (Note, the coordinate axes follows normal conventions by starting at the lower left corner.)

Last, as shown in figure 5 (bottom), each coarse-mesh block is assigned a letter code representing a particular set of in-seam material properties. Material B represents linear elastic coal blocks that are 50% extracted. Material E is for the gob left in the wake of full extraction mining. (Note, actual input of properties for materials B and E occurs later.) The material label 1 signifies an open element. Such elements represent entries, crosscuts, and any other open areas within the seam. In this model, an open area exists between the gob and the partially mined areas. Finally, the material label O (zero) represents coarse-mesh blocks within the fine-mesh boundary.

FINE-MESH GENERATION

With the coarse-mesh grid complete, modeling then proceeds to the fine-mesh covering the area of interest. Figure 6 (top) shows a closeup of the mine plan and the six by five array of coarse-mesh blocks within the

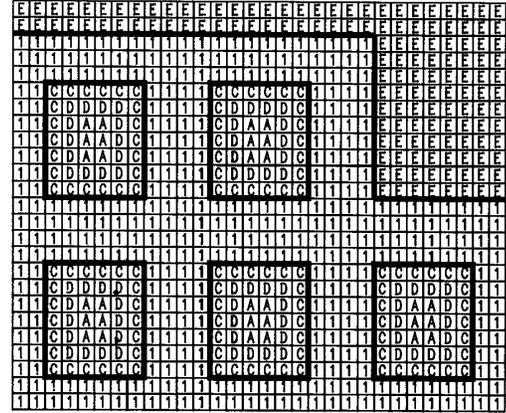
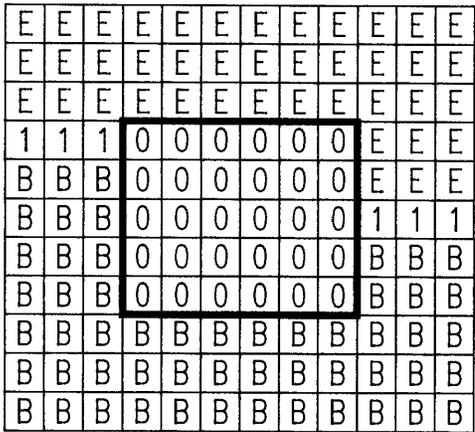
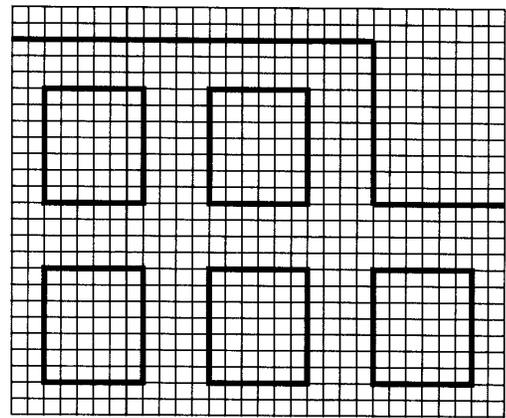
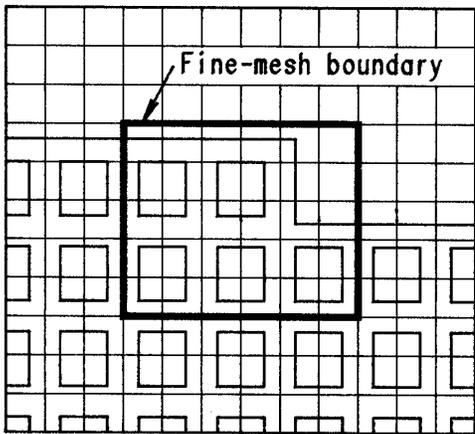
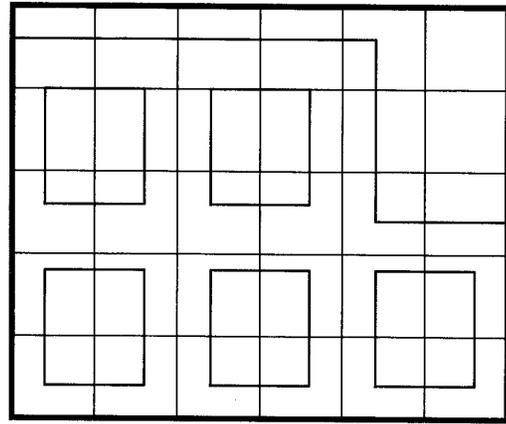
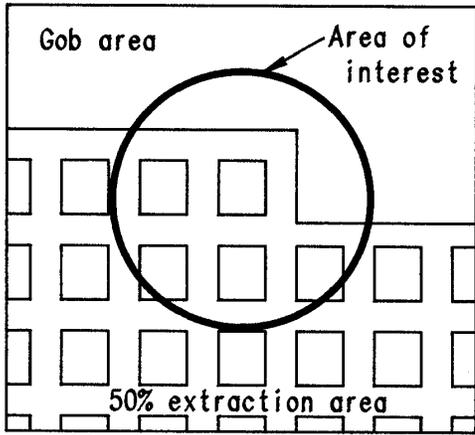


Figure 5.—Coarse-mesh generation in basic mine plan. Top, Idealized plan for retreat room-and-pillar mining; middle, superimposed coarse-mesh block grid; bottom, material property assignments for coarse-mesh grid.

Figure 6.—Fine-mesh generation in basic mine plan. Top, Closeup of plan and coarse-mesh blocks within fine-mesh boundary; middle, superimposed fine-mesh grid within fine-mesh boundary; bottom, material property assignments for fine-mesh grid.

fine-mesh boundary. Figure 6 (middle) shows how each coarse-mesh block is subdivided into a five by five array of fine-mesh elements. Finally, as shown in figure 6 (bottom), material properties are assigned with a letter code to each fine-mesh element. As in the coarse-mesh, the label 1 means an open element, and E represents gob. Material A is for linear elastic coal, while materials C and D are for coal materials with a low- and high-yield strength, respectively. Notice how the model pillars are constructed with low-yield-strength material in the outer elements and linear elastic elements in the core.

An important choice in setting up a MULSIM/NL model is the size of the coarse- and fine-mesh grid areas. Unfortunately, each problem differs significantly, and no definite guidelines exist. The fine-mesh should cover the area of interest so that MULSIM/NL provides stress and displacement calculations with sufficient detail and accuracy. However, the coarse-mesh serves a slightly different purpose than the fine mesh. In BEM programs like MULSIM/NL, the in-seam material outside the coarse-mesh area is completely rigid (i.e., it has infinite stiffness). Therefore, stress and displacement calculations in blocks or elements near the external boundary of the model have very low accuracy due to the effect of the infinitely stiff external in-seam material. The coarse-mesh blocks act as a buffer between the infinitely stiff model boundary and

the fine-mesh area where computational accuracy is desired. Good modeling practice with MULSIM/NL requires at least three rows (and preferably ten rows) of coarse-mesh blocks beyond the fine-mesh area to keep the infinitely stiff external boundary at bay.

To summarize, the basic MULSIM/NL modeling process requires the following steps:

- Start with an idealized mine plan.
- Choose an element and/or block size.
- Overlay a coarse-mesh grid.
- Assign material properties to coarse-mesh blocks.
- Subdivide coarse-mesh blocks within fine-mesh boundary into elements.
- Assign material properties to fine-mesh elements.

The example problem uses five different in-seam materials (A through E) whose properties will require definition in an actual MULSIM/NL input file. These discussions have illustrated the relationship between an actual mine plan and the coarse- and fine-mesh boundary-element models used to approximate it. Knowing something about these grids and their creation should make the following detailed discussions of the actual MULSIM/NL input file more understandable.

INPUT FILE STRUCTURE FOR MULSIM/NL

INPUT FILE SECTIONS

As shown in figure 7, a basic MULSIM/NL input file has three parts: a control section, the fine-mesh geometry, and the coarse-mesh geometry. Figure 8 shows an actual input file for the sample problem discussed in the prior section that illustrates this basic three-part structure again. The control section defines all the basic variables for MULSIM/NL, such as problem size, material properties, in situ stress fields, and seam orientation. The fine-mesh section defines geometry and assigns properties to a sub-array of fine-mesh elements within the coarse-mesh blocks. The coarse-mesh section defines the mine geometry and assigns material properties to an array of coarse-mesh blocks that comprise the overall mine model.

Prior discussions covered the basic modeling process with MULSIM/NL and the creation of the coarse- and fine-mesh data arrays. Again, figures 5 and 6 show this process. The coarse-mesh array shown in figure 5 (bottom) becomes the coarse-mesh section of the input file shown in figure 8. Similarly, the fine-mesh array shown in figure 6 (bottom) becomes the fine-mesh section of the input file.

MULSIM/NL also has multiple mining step and multiple-seam capabilities. Utilization of these options changes the input file structure as shown in figures 9, 10, and 11. Variables within the control section will change in ways discussed later. Figure 9 shows the input file structure for a single-seam, multiple mining step problem. Notice how this input file structure has changed from the single-seam, single-step input file shown in figure 7. The fine- and coarse-mesh geometry sections are repeated for all subsequent mining steps. Each mining step changes the fine- and/or coarse-mesh geometry array from the prior

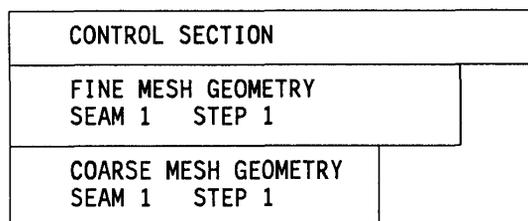


Figure 7.—MULSIM/NL input file structure for single-seam, single-step problem.

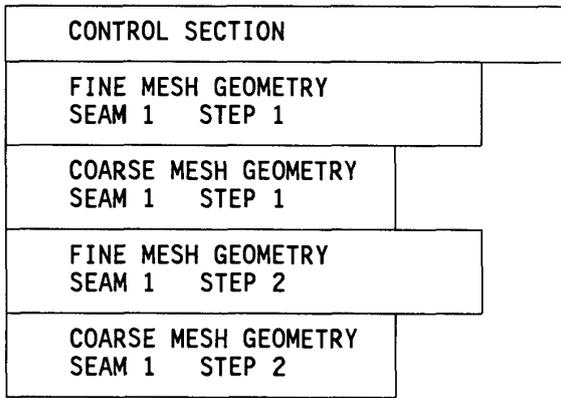


Figure 9.—MULSIM/NL input file structure for single-seam, multiple-step problem.

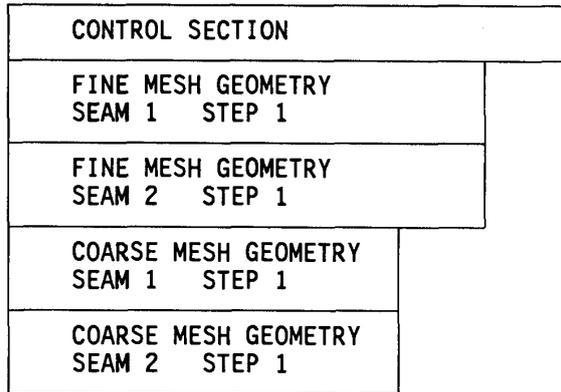


Figure 10.—MULSIM/NL input file structure for multiple-seam, single-step problem.

step. By way of example with the mining geometry shown in figure 8, a second mining step might split the point pillar; a third step could recover the right wing, and finally, a fourth step would recover the left wing.

Figure 10 shows the input file structure for a multiple-seam, single-step problem. Variables within the control section undergoes minor changes as discussed later. Notice how this input file structure changes from the single-seam, single-step structure of figure 7, and also note how it differs from the single-seam, multiple-step structure of figure 9. In a multiple-seam problem, additional seams

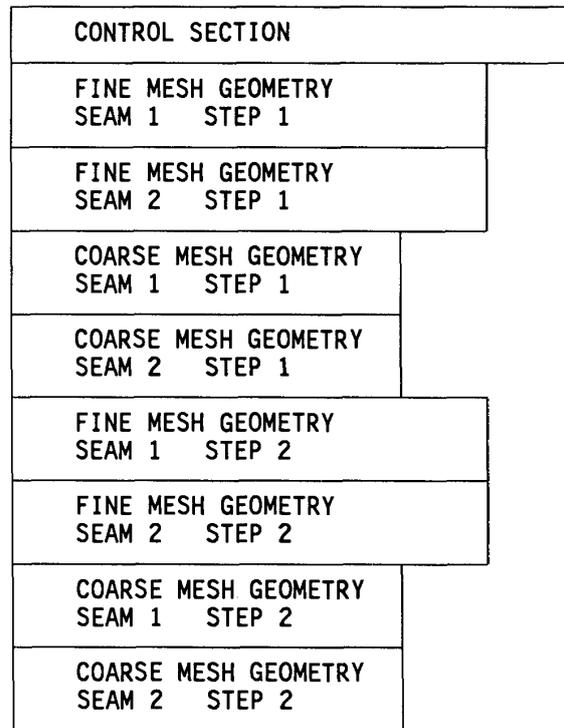


Figure 11.—MULSIM/NL input file structure for multiple-seam, multiple-step problem.

might lie above and/or below the seam. First, the fine-mesh geometry is defined for each seam, followed by the coarse-mesh geometry for each seam.

Finally, figure 11 shows the input file structure for a multiple-seam, multiple-step problem. Again, compare how this input file structure changes from the prior structures shown in figures 7, 9, and 10. The multiple-seam fine- and coarse-mesh geometry sections are repeated for each additional mining step.

UNITS

Before moving into discussions on the control parameter section of the input file, the important subject of units requires a few words. MULSIM/NL will work with any *consistent* system of units. Table 1 gives examples of three unit systems for input and the resultant units for the output.

Table 1.—MULSIM/NL input and output units for three typical consistent unit systems

Variable	Megapascal-meter system	Pound-inch system	Pound-foot system
Input:			
Length	m	in	ft
Stress	MPa	lb/in ²	lb/ft ²
Modulus	MPa	lb/in ²	lb/ft ²
Output:			
Displacement	m	in	ft
Stress	MPa	lb/in ²	lb/ft ²
Energy	MJ	lb-in	lb-ft
Energy density	MJ/m ³	lb-in/in ³	lb-ft/ft ³
Energy release rate ..	MJ/m ²	lb-in/in ²	lb-ft/ft ²

GLOBAL COORDINATE SYSTEM

MULSIM/NL uses a global coordinate system from which the local origin of each seam is defined. The local origin for each seam always lies in the lower left corner of the coarse-mesh geometry for that seam. Both the local and the global coordinate systems are right-handed. The Z axis of the global coordinate system is always

perpendicular to the earth's surface, while the global X and Y axes are parallel to that surface. The orientation of the global coordinate system is therefore fixed with respect to the earth. However, the local coordinate system can have any orientation with respect to the global system, i.e., the seams can have any orientation relative to the Earth's surface.

In the control section of the input file, the user must define the local origin of each seam relative to a global coordinate system origin. Implicitly, the user must choose a location for the global coordinate system. As shown in figure 12, two logical alternatives exist for this choice: (1) the actual ground surface, or (2) the lowermost seam. As discussed later, this choice affects the way primitive or premining stresses are input. MULSIM/NL does not permit rotation of the seams or their local coordinate systems with respect to each other. This restriction means that the local x axes and local y axes shown in figure 12 must remain parallel. MULSIM/NL does permit lateral translation of the seams and their local coordinate systems with respect to each other. Therefore, the local z axes are not required to be collinear; however, in most practical applications, the local z axes are collinear as shown in figure 12.

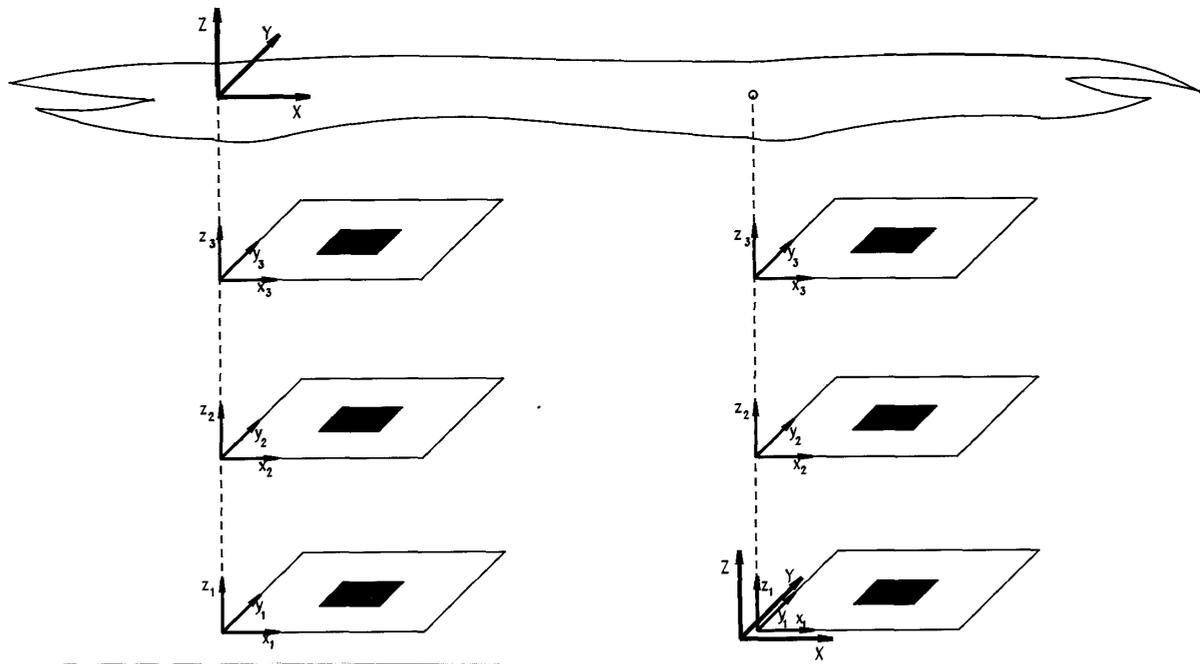


Figure 12.—Options for global coordinate system location. Left, Global coordinate system located at surface for multiple-seam problem; right, global coordinate system located at lowermost seam.

CONTROL PARAMETER DEFINITION

Prior discussions gave the general three-part structure of the input file and showed how to create the coarse- and fine-mesh sections of that file. Knowing something about the overall input file structure and how it relates to the mine geometry under consideration will facilitate descriptions of the very important control parameters. Nine different record types comprise the control section, and many record types can contain multiple records. The following discussions will describe each record type and each variable within that record type. Where necessary, appropriate ranges are provided for those variables. These discussions use the control section of the sample problem input file from figure 8 as an example.

Record Type 1—Single Record—Format (20A4)

TITLE - an arbitrary 80 character title.

The first record gives a descriptive title to the problem.

Record Type 2—Single Record—Format (F8.2, E12.6, 18)

ν - Poisson's ratio

E - Young's modulus

NSEAM - Number of seams.

The second record gives the Poisson's ratio and Young's modulus for the rock mass surrounding the seam. As in most BEM programs, the rock mass is linear elastic. In this example, ν equals 0.25, which is the common value, and E equals 20,000 MPa (3,000,000 psi), which is moderately high for most rock, but typical for massive sandstones. The number of seams, NSEAM, is 1 for this single-seam example. MULSIM/NL can analyze up to four seams.

Record Type 3—Single Record—Format (18)

NMATS - Number of materials.

The third record specifies the number of in-seam materials. At least one in-seam material is required, and up to 26 materials are allowed (A through Z). This example uses five materials (A through E) in the fine- and coarse-mesh geometry as shown in figure 8.

Record Type 4—NMATS Records—Format (10E8.0)

These records define the in-seam material properties beginning with those for material A, then material B, etc., up to NMATS materials. Figure 13 shows the stress-strain models for MULSIM/NL that include (1) linear elastic for coal, (2) strain softening, (3) elastic plastic, (4) bilinear hardening, (5) strain hardening, and (6) linear elastic for gob. The first three models are for the unmined in-seam coal material, while the latter are for the broken gob

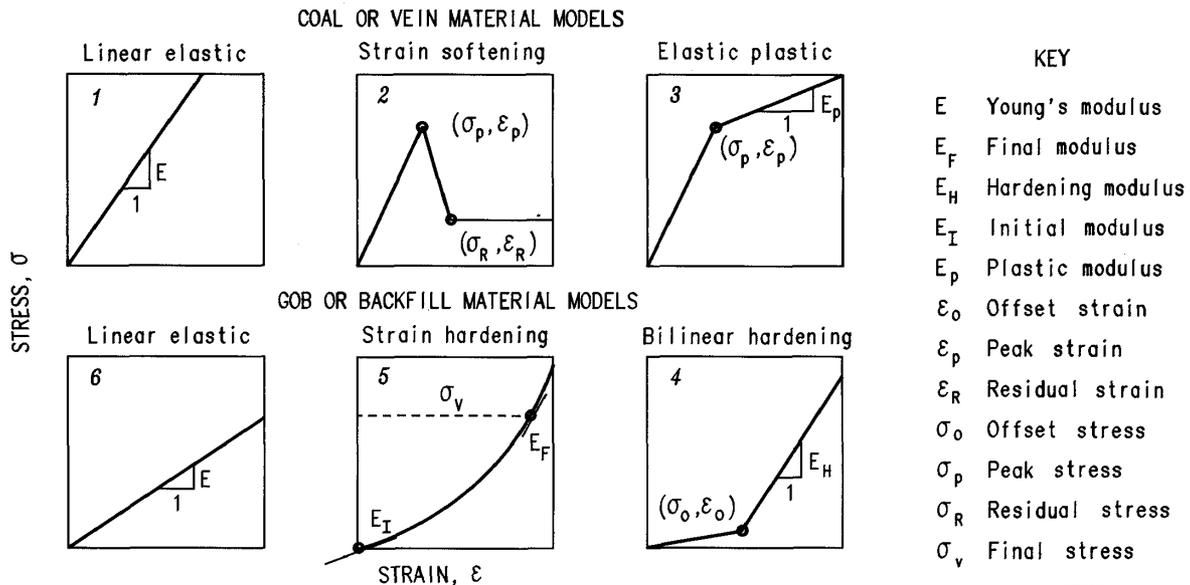


Figure 13.—Six different stress-strain models available in MULSIM/NL.

material left in the wake of full extraction mining. As will be discussed later, the distinction between coal material models and gob material models becomes crucial in the subsequent energy calculations.

Linear Elastic for Coal (Model 1)

This basic model requires little discussion since it is the basis for MULSIM/NL's predecessors MULSIM (3) and MULSIM/BM (2). The required parameters for the model are E (Young's modulus) and G (shear modulus), which are related by

$$G = \frac{E}{2(1 + \nu)}. \quad (2)$$

In the BEM calculations, Young's modulus E relates the normal stress to the normal displacement (closure) across an element, whereas G relates the associated shear stresses to the corresponding shear displacements (rides). As discussed in a related document, the three boundary conditions applied to each element behave *independently* of one another.

With the linear elastic model, MULSIM/NL works internally with a stiffness K given by

$$K = \frac{E}{t}, \quad (3)$$

where t is the seam thickness.

Linear Elastic for Gob (Model 6)

This model is the counterpart to model 1 and is intended for the gob material left in the wake of mining. Required parameters are again E and G plus a "gob height factor," n. The factor n is the ratio between the height of the zone of broken rotated gob fragments and the unmined seam thickness. The factor n typically ranges from 2 to 6 and averages about 4. This material model and BEM program assume that the rock mass remains linear elastic beyond a range of "n" seam thicknesses. With linear elastic gob elements, MULSIM/NL operates with a stiffness given by

$$K = \frac{E}{nt}. \quad (4)$$

In effect, the factor n accounts for the larger effective seam thickness present in extracted areas now filled with gob material. (Another way to consider n is as a modulus reduction factor as was done by Beckett and Madrid (2).)

Strain Softening for Coal (Model 2)

This model, after Crouch and Fairhurst (4), approximates the complete stress-strain curve observed during laboratory strength tests on coal conducted under true displacement control. In principle, it also describes the yielding behavior of moderately sized pillars or the perimeters of large pillars. Field observations by Wang (7) and Iannacchione (8) support a strain-softening model for full-scale pillars in many mines.

Required input parameters for this stress-strain model are a peak stress and peak strain plus a residual stress and residual strain. As shown in figure 13, these two points define the strain-softening model. In addition, the model requires a Poisson's ratio, ν . For the normal components of stress and displacement, the strain-softening model works with the peak and residual stress-strain points. However, for the two shear components, MULSIM/NL scales the specified peak and residual stresses and strains by a factor $1/[2(1 + \nu)]$. In the linear elastic portion of the strain-softening model, MULSIM/NL relates normal stress and displacement with an elastic modulus computed as $E = \sigma_p / \epsilon_p$, where σ_p is peak stress and ϵ_p is peak strain. The shear stresses and displacements also satisfy a similar relation in the initial linear portion. Shear modulus is computed from elastic modulus using equation 2.

One restriction on the strain-softening model is that residual strain must exceed peak strain, and peak stress must exceed residual stress. For strains greater than residual, stress remains constant at the residual level.

Elastic Plastic for Coal (Model 3)

This stress-strain model, closely akin to the strain-softening model, approximates a pseudoductile behavior in pillar cores (5). Required input for the model is again a peak stress and peak strain, the modulus of the postyield portion (i.e., its slope) and a Poisson's ratio, ν . As in the prior model, the factor $1/[2(1 + \nu)]$ scales the amplitude of the normal stress-strain curve to obtain the shear stress-strain relations. Therefore, E and G satisfy equation 2 in the initial linear portion.

Bilinear Hardening for Gob (Model 4)

This stress-strain model, analogous to the elastic-plastic model in certain respects, permits a certain amount of deformation to occur prior to introducing significant element stiffness. Input requirements for the model are stress and strain at the inflection point, modulus in the hardening region beyond the inflection point, and Poisson's ratio, ν . Again, the stress-strain relation for the normal direction of the element is scaled by $1/2[(1 + \nu)]$ to obtain the stress-strain relation for the tangential directions of the element.

This gob model also requires the gob height factor n . This factor increases the effective seam thickness for the element and reduces its stiffness in the BEM program workings.

Strain Hardening for Gob (Model 5)

This model allows the gob material to increase in stiffness as it consolidates under increasing load. As derived in the theoretical and programmer's manual, the stress-strain relationship is

$$\sigma = \frac{E_I}{n} \left[\frac{n \sigma_v}{E_F - E_I} \right] \left[\exp \left[\frac{E_F - E_I}{n \sigma_v} \right] \left[\frac{D}{t} \right] - 1 \right], \quad (5)$$

where E_I = initial modulus,

E_F = final modulus,

σ_v = virgin vertical stress,

D = seam closure,

t = seam thickness,

and n = gob height factor.

The parameter $(E_F - E_I)/(n \sigma_v)$ controls the degree of nonlinearity in the model. As shown in figure 13, input parameters are E_I , E_F , σ_v , n , and ν . As with the prior

models, scaling the above stress-strain relation for the normal direction of the element by a factor of $1/[2(1 + \nu)]$ provides appropriate stress-strain relations for the tangential directions of the element.

Table 2 summarizes the basic input parameters for these six material models and the input format required. For each material, MULSIM/NL reads one record containing the appropriate parameters in the proper format for that material. The value of the first parameter directs the program to the proper stress-strain model for each material defined. For the six models currently permitted, the assignments are 1—linear elastic coal, 2—strain-softening coal, 3—elastic-plastic coal, 4—bilinear gob, 5—strain-hardening gob, and 6—linear elastic gob.

The control parameter example shown in figure 8 uses five materials. The first, material A, is linear elastic coal with a Young's modulus of 500 MPa and a shear modulus of 200 MPa. Material B is also linear elastic coal, but the moduli have half the values as material A. In effect, elements and blocks composed of material B behave as if they were made of material A at 50% extraction. Materials C and D are elastic-plastic coal, with yield stresses at 10 and 20 MPa, and yield strains at 0.020 and 0.040, respectively. Once they yield, the stress cannot increase further. Poisson's ratio is 0.25 for both materials. The initial modulus is 500 MPa for both C and D (10.0/0.020 or 20.0/0.040). Finally, material E is linear elastic gob with a Young's modulus and shear modulus of 50 and 20 MPa, respectively. The gob height factor is 4, which implies that the gob thickness is four times the coal seam thickness defined later.

Table 2.—Material model parameters and material property array EPROP structure

Model	1	2	3	4	5	6
1 . . .	Linear elastic coal-(1).	Young's modulus (E).	Shear modulus (G).	NAP	NAP	NAP.
2 . . .	Strain-softening coal-(2).	Peak stress . .	Peak strain . .	Residual stress . . .	Residual strain.	Poisson's ratio (ν).
3 . . .	Elastic plastic coal-(3).	. . do. do.	Plastic modulus . .	Poisson's ratio (ν).	NAP.
4 . . .	Bilinear hardening gob-(4).	Offset stress . .	Offset strain . .	Hardening modulus.	. . do.	Gob height factor (n).
5 . . .	Strain hardening gob-(5).	Initial modulus (E_I).	Final modulus (E_F).	Final stress (σ_v) . .	Gob height factor (n).	Poisson's ratio (ν).
6 . . .	Linear elastic gob-(6).	Young's modulus (E).	Shear modulus (G).	Gob height factor (n).	NAP	NAP.

NAP Not applicable.

NOTE.—For these 6 material models, there are no parameters 7 through 10.

**Record Type 5—Single Record—Format
(6(F6.0, F6.4))**

- A₁₁ - normal stress component σ_{xx} at global origin
 B₁₁ - normal stress gradient $\Delta\sigma_{xx}$ from global origin
 A₁₂ - shear stress component τ_{xy} at global origin
 B₁₂ - shear stress gradient $\Delta\tau_{xy}$ from global origin
 A₁₃ - shear stress component τ_{xz} at global origin
 B₁₃ - shear stress gradient $\Delta\tau_{xz}$ from global origin
 A₂₂ - normal stress component σ_{yy} at global origin
 B₂₂ - normal stress gradient $\Delta\sigma_{yy}$ from global origin
 A₂₃ - shear stress component τ_{yz} at global origin
 B₂₃ - shear stress gradient $\Delta\tau_{yz}$ from global origin
 A₃₃ - normal stress component σ_{zz} at global origin
 B₃₃ - normal stress gradient $\Delta\sigma_{zz}$ from global origin.

This record specifies primitive or premining stresses to MULSIM/ NL. The quantities A₁₁, B₁₁, etc., listed above form symmetric tensors that look like

$$[A] = \begin{bmatrix} A_{11} & A_{12} & A_{13} \\ A_{12} & A_{22} & A_{23} \\ A_{13} & A_{23} & A_{33} \end{bmatrix} = \begin{bmatrix} \sigma_{xx} & \tau_{xy} & \tau_{xz} \\ \tau_{xy} & \sigma_{yy} & \tau_{yz} \\ \tau_{xz} & \tau_{yz} & \sigma_{zz} \end{bmatrix} \quad (6)$$

$$[B] = \begin{bmatrix} B_{11} & B_{12} & B_{13} \\ B_{12} & B_{22} & B_{23} \\ B_{13} & B_{23} & B_{33} \end{bmatrix} = \begin{bmatrix} \Delta\sigma_{xx} & \Delta\tau_{xy} & \Delta\tau_{xz} \\ \Delta\tau_{xy} & \Delta\sigma_{yy} & \Delta\tau_{yz} \\ \Delta\tau_{xz} & \Delta\tau_{yz} & \Delta\sigma_{zz} \end{bmatrix}. \quad (7)$$

[A] contains the primitive stress components at the *global* origin, whereas [B] has the stress gradient components as a function of *vertical* distance from the global origin ΔZ . Thus, the primitive stress tensor [P] for any element in the MULSIM/NL model is easily computed within the program as

$$[P] = [A] + [B]\Delta Z. \quad (8)$$

For the input file shown in figure 8, the global origin is located at seam level similar to figure 12 (right) rather than on the surface. The σ_{zz} (A₃₃) stress component equals 10 MPa, and all other stress components and gradients are zero. Therefore, the primitive stress tensor P for each element in the model has the P_{zz} component only, which equals 10 MPa.

Sign conventions for the stress and stress gradient components can cause problems. MULSIM/NL uses positive for compression; therefore, the stress components A at the global origin are usually all positive. In the global coordinate system, ΔZ is *negative* for increasing depths; therefore, the stress gradient components B are also *negative* in most cases. Negative stress gradients and negative ΔZ will give positive primitive stresses P that increase with depth.

**Record Type 6—Single Record—Format
(F8.2, 2I8, 16X, 4I8)**

- BW - coarse-mesh block width
 NBXI - number of coarse-mesh blocks in x direction
 NBET - number of coarse-mesh blocks in y direction
 IFXS - fine-mesh starting block along x axis
 IFXE - fine-mesh ending block along x axis
 IFYS - fine-mesh starting block along y axis
 IFYE - fine-mesh ending block along y axis.

The sixth record defines the coarse-mesh size (NBXI by NBET) and gives the location of the fine-mesh area within the coarse-mesh. For the input file example shown in figure 8, blocks of 7.62 m width comprise the coarse-mesh, which extends for 12 blocks in the x direction and 11 blocks in the y. The fine-mesh runs from blocks four to nine along the x axis and from blocks four to eight along the y. IFXS equals 4; IFXE equals 9; IFYS equals 4; and IFYE equals 8. A five by five array of fine-mesh elements subdivides each block within this region. Therefore, the fine mesh in the example, which covers six by five blocks, divides into a 30- by 25-element array.

Record Type 7—NSEAM Records—Format (4F8.1)

X_o, Y_o, Z_o - global coordinates of the local coordinate system for the seam

THIKNS - average seam thickness.

These records specify the location in global coordinates of the local origin of each seam, as well as the average seam thickness. This example (fig. 8) assumes a global coordinate system at the seam level much like figure 12 (right) so the global and local coordinate systems coincide. Therefore, X_o , Y_o , and Z_o are zero. The seam thickness in the example is 1.5 m.

The implicit choice of global coordinate system origin affects not only the local coordinate system definition in this record but also the primitive stress definition in record 5. Placing the global coordinate system origin at seam level, as in this example (fig. 8) and in figure 12 (right), requires nonzero terms in the global origin stress component tensor A in equation 8. Generally, the stress gradient tensor B will contain nonzero terms for problems entailing multiple or inclined seams. The example considered in figure 8 is a single-horizontal seam. In this special case, all terms in the stress gradient tensor B are zero. Placing the global origin on the surface as shown in figure 12 (left) example requires that all terms in the stress component tensor A equal zero. Therefore, the stress gradient tensor B must have nonzero terms to have nonzero primitive stresses at seam level.

For the special case of horizontal seams (either single or multiple), all global X_o 's must equal one another and similarly for the Y_o 's. MULSIM/NL does not permit relative rotation or lateral translation of the local coordinate system of the seam. The local x and y axes must remain parallel, and the local z axes should remain collinear. Normally, as shown in the example input file (fig. 8) and also in figure 12, all X_o and Y_o equal zero. For a single-inclined seam, X_o and Y_o can remain zero. Only in the special case of inclined, multiple seams does one encounter nonzero and nonequal X_o and Y_o . In this case, the analyst must use trigonometry to calculate global coordinates (X_o , Y_o , Z_o) for each seam such that the local z axes remain collinear.

Record Type 8—Single Record—Format (9F8.5)

EN(1,1), EN(2,1), EN(3,1) - direction cosines of the local x axis with respect to the global axes X, Y, and Z.

EN(1,2), EN(2,2), EN(3,2) - direction cosines of the local y axis with respect to the global axes X, Y, and Z.

EN(1,3), EN(2,3), EN(3,3) - direction cosines of the local z axis with respect to the global axes X, Y, and Z.

The record specifies the orientation of the local coordinate system axes with respect to the global axes. Figure 14 provides several typical examples of different local coordinate system orientations with respect to the global system along with the approximate direction cosine values.

Record Type 9—Single Record—Format (2F8.2, 18, 24X, 218)

ORF - overrelaxation factor

SIGACC - stress convergence criterion

ITMAX - maximum number of iterations per mining step

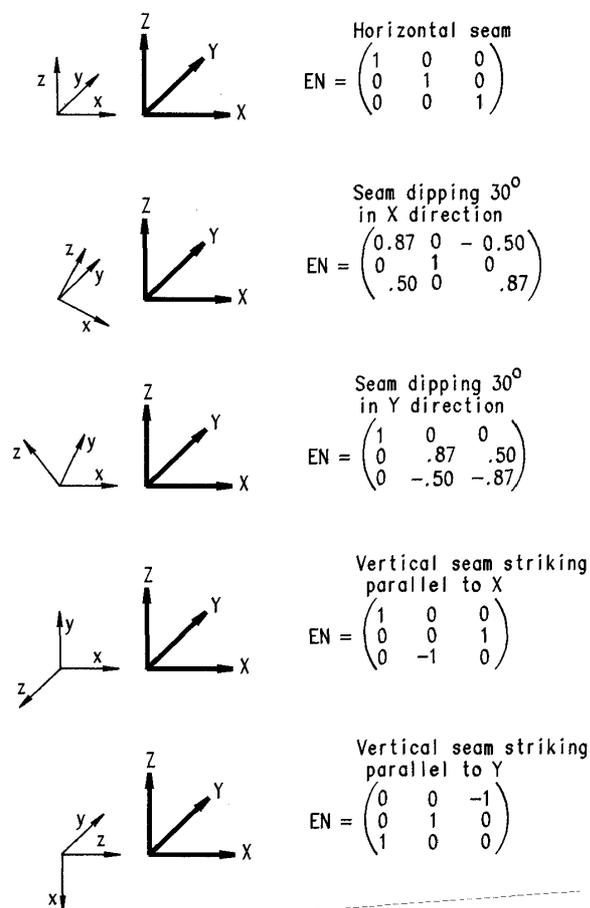


Figure 14.—Typical direction cosine (EN) matrices for orienting local coordinate system relative to global coordinate system.

- NSTEP - mining step number for the first step
- MXSTEP - number of new mining steps in this problem.

This record specifies the critical MULSIM/NL control parameters. The first variable, ORF, is an overrelaxation factor. MULSIM/NL solves the nonlinear system of boundary-element equations with a Gauss-Seidel iteration procedure that uses the overrelaxation factor to accelerate convergence of the stress-displacement solution. (See Dahlquist and Bjork (9) for further explanation of a Gauss-Seidel iteration procedure.) ORF can range from 1.00 to about 1.50. Figure 15 shows the effect of ORF on computation time and number of iterations to solve a typical MULSIM/NL problem. Both these measures of equation-solving efficiency decrease as ORF increases to about 1.45; however, beyond 1.45, the computation time and number of iterations increases dramatically. No method exists to tell what the optimum ORF is for any particular problem. Optimum values for ORF usually range from 1.25 to 1.45. This example uses 1.35.

SIGACC is a stress convergence criterion used during the equation-solving process by the nonlinear material models. MULSIM/NL also has a built-in displacement convergence criterion computed as the seam thickness divided by 1,000. When all elements satisfy both the stress and displacement convergence criteria, then the iterative equation solver stops. The calculated stresses and displacements did not change significantly from the previous iteration, and they lie sufficiently close to the prescribed nonlinear stress-strain relationships. This example specifies SIGACC as 0.10 MPa (15 psi), which is quite stringent by most practical engineering requirements.

ITMAX places a limit on the number of iterations that the MULSIM/NL equation solver can use in any one step to calculate elemental stresses and displacements. Simple problems that only use MULSIM/NL's linear capabilities converge in 15 to 30 iterations. Using the nonlinear material models may double the number of iterations required for convergence. Large, well-behaved, nonlinear problems rarely require more than 100 iterations to

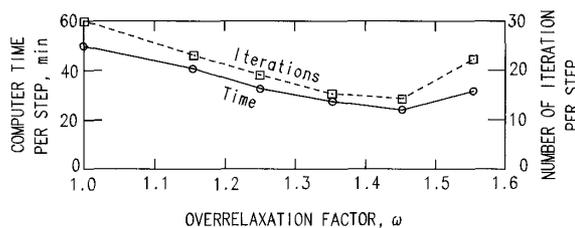


Figure 15.—Overrelaxation factor versus computer time and number of iterations per step.

converge. This example specifies a maximum of 120 iterations per mining step.

The variables NSTEP and MXSTEP control the multiple mining step capabilities in MULSIM/NL. NSTEP gives the number for the first mining step and MXSTEP specifies the number of new mining steps in the current problem. This single-step example starts at mining step 1 (NSTEP) and continues for just 1 step (MXSTEP). The MULSIM/NL energy calculations performed on the basis of equation 1 require stress and displacement *changes* as input. Therefore, to obtain these energy calculations, the user must specify at least *two* mining steps. Energy calculations begin at the second mining step and continue thereafter.

INPUT SUMMARY

A basic MULSIM/NL input file contains three parts: a control section, the fine-mesh geometry section, and the coarse-mesh geometry section. The control section specifies all the basic parameters for the boundary-element model, such as problem size, material properties, in situ stress fields, and seam orientation. The fine-mesh and coarse-mesh geometry sections define the mine geometry and material distribution for the model. As shown in the sample input file (fig. 8), these sections approximate the actual mine geometry and actual material distribution in the real seam or vein. As shown in figures 9, 10, and 11, the fine- and coarse-mesh geometry sections are repeated in a specific pattern for multiple seam and/or multiple mining step problems.

INPUT FILE GENERATION WITH PREPROCESSOR MULPRE/NL

A preprocessor called MULPRE/NL is available to assist the user generate a MULSIM/NL input file. The preprocessor uses various menus and a question and answer format to create the control section of the input file. Next, MULPRE/NL helps the user build the fine- and coarse-mesh models with friendly computer graphics. The MULPRE/NL graphics contain two important functions for speeding the model building. The first function allows the user to change the material property code from one value to another over a user-defined area. The second function lets the user copy a set of material property codes from one user-defined area to another. Other functions also exist, such as a zoom capability, but the change and copy functions perform the bulk of the model-generation effort. Subsequent Bureau documents will provide more complete instructions on the operation and program structure of the preprocessor MULPRE/NL. New MULSIM/NL users are advised to use the preprocessor for initial model generation since accurate model generation is greatly facilitated.

RUNNING MULSIM/NL

The command sequence to actually run MULSIM/NL depends heavily on the host computer and its operating system. Subsequent discussions will provide two example execution command sequences - one for a Hewlett Packard (HP)³ engineering workstation using the UNIX operating system and another for a VAX minicomputer using the VMS operating system. In principle, these examples will apply to other machines and other operating systems. Prior to presenting these examples though, the files used by MULSIM/NL require discussion. During its execution, MULSIM/NL creates and uses files for input, output, and temporary storage. These discussions will give the purpose, format, and general contents of those files. Finally, a chart is presented of actual MULSIM/NL execution time versus the problem size. This chart provides the user with a simple means to approximate the execution time for new problems.

MULSIM/NL FILES

MULSIM/NL uses many files during execution. Table 3 lists these files and gives their general content. The FORTRAN unit name is somewhat generic. The actual FORTRAN unit name will vary with different computer systems. Most of the files are binary and unreadable by the user. Only the input, output, and fine- and coarse-mesh output data files are formatted American Standard Code for Information Interchange (ASCII) files and hence readable by the user.

³Reference to specific products does not imply endorsement by the U.S. Bureau of Mines.

Table 3.—MULSIM/NL files

FORTRAN unit name	Contents	Save status
FOR05.DAT ..	Input file	Save - Input file.
FOR06.DAT ..	Out file (printout)	Save - print-out file.
FOR11.DAT (12, 13, 14).	Coarse-mesh element stress and dis- placement data for seam 1 (2, 3, 4).	Temporary-delete.
FOR21.DAT (22, 23, 24).	Fine-mesh block stress and displacement data for seam 1 (2, 3, 4).	Do.
FOR31.DAT (32, 33, 34).	Coarse-mesh element stress, displacement and energy data for seam 1 (2, 3, 4).	Save - coarse-mesh data files.
FOR41.DAT (42, 43, 44).	Fine-mesh block stress, displacement and energy data for seam 1 (2, 3, 4).	Save - fine-mesh data files.

MULSIM/NL uses FORTRAN unit 05 for input and unit 06 for printed output. The user must rename and save these files for subsequent use and inspection. After MULSIM/NL converges on a sufficiently accurate stress and displacement solution, it writes the fine- and coarse-mesh stress and displacement data for *each* seam to files in ASCII format. FORTRAN unit 21 (22, 23, 24) stores the fine-mesh element stress and displacement data for seam 1 (2, 3, 4). Similarly, FORTRAN unit 11 (12, 13, 14) stores the coarse-mesh block stress and displacement data for seam 1 (2, 3, 4). Again, the user can opt to delete all these files after MULSIM/NL finishes.

MULSIM/NL uses the stresses and displacements on the above files to calculate various energy and energy change quantities for each mining step. Upon completing the energy calculation, MULSIM/NL writes the fine- and coarse-mesh stress, displacement, and energy data to final output files in ASCII format. FORTRAN unit 41 (42, 43, 44) contains the fine-mesh data for seam 1 (2, 3, 4), and FORTRAN unit 31 (32, 33, 34) contains the coarse-mesh data for seam 1 (2, 3, 4). The user must rename and save these important output data files for subsequent use and study. To facilitate graphical display of the output data, MULSIM/NL writes the fine-mesh element data in order. Starting at the element with the lowest x and y coordinate, data records are written by increasing x coordinate first, then by increasing y coordinate. The coarse-mesh data also follow the same format of increasing x coordinate first, followed by increasing y coordinate.

MULSIM/NL EXECUTION COMMAND SEQUENCES

As already stated, the actual commands needed to execute MULSIM/NL depend heavily on the host computer and its operating system. Figure 16 shows an example execution command sequence for a HP engineering workstation using a UNIX operating system. In this case, these UNIX commands form a file named "multirun." Submitting this file with "multirun &" causes the following

```
# Csh Program: multirun
cd /home/zipf/gpanels
mulnl <gpan2.inp >gpan2.prt
mv fort.41 gpan2.f1
mv fort.42 gpan2.f2
mv fort.31 gpan2.c1
mv fort.32 gpan2.c2
rm fort.*
```

Figure 16.—MULSIM/NL execution command sequence for Hewlett Packard engineering workstation using UNIX operating system.

to occur. First, the computer changes directory (cd) to /home/zipf/gpanels. This directory must contain an executable version of MULSIM/NL, in this case called "mulnl," and an input file, in this case called "gpan2.inp." Next, the computer runs "mulnl" on input file "gpan2.inp," which is equivalent to FORTRAN unit 05 and sends the printed output to "gpan2.prt," which is equivalent to FORTRAN unit 06. This example is a multiple-seam problem. FORTRAN units 41 and 42 contain the fine-mesh output data, and 31 and 32 contain the coarse-mesh output data that must be saved for future use. The move command (mv) renames each of these FORTRAN files with an appropriate new name. Finally, the remove command (rm) deletes all other FORTRAN files that this example chooses not to save including 11, 12, 21, and 22. Upon completion of this command sequence and MULSIM/NL, the directory "gpanels" will contain five new output data files including "gpan2.prt," "gpan2.f1," "gpan2.f2," "gpan2.c1," and "gpan2.c2," as well as the input file "gpan2.inp" and the executable file "mulnl."

Figure 17 shows another execution command sequence for a VAX minicomputer using the VMS operating system. These commands happen to reside in a file called "SHARP.COM," which upon submission to the computer causes the following actions to occur. First, the default directory is set to [ZIPF.SHARP]. That directory contains an input file called "SHARP.INP," which is a single-seam, multiple-step problem. Next, certain necessary FORTRAN unit files are set equivalent to other files such as "005" and the input file "SHARP.INP," "006" and the printed output file "SHARP.PRT," "041" and the fine-mesh data file "SHARP.FMD," and last, "031" and the coarse-mesh data file "SHARP.CMD." The following command will run MULSIM/NL, which in this case is the executable file called "MULNL" that resides in a separate directory called [ZIPF.MULSIM]. Finally, after MULSIM/NL stops, the delete command removes all other FORTRAN files including 11 and 21.

Again, the actual command sequence needed to run MULSIM/NL depends on the available computer and its operating system. These examples, coupled with the prior explanation of MULSIM/NL's many operating files, should provide users with sufficient background information to tailor an execution command sequence for their available hardware.

MULSIM/NL RUNNING TIMES

Execution times for MULSIM/NL will depend on the particular computer used and the size and complexity of the problem. Figure 18 shows an approximate relationship between solution time and problem size. This plot originates from observed execution times for a large variety of MULSIM/NL problems. These times only apply to a SUN III engineering workstation; however, these

```

$ SET DEFAULT [ZIPF.SHARP]
$ DEFINE FOR005 SHARP.INP
$ DEFINE FOR006 SHARP.PRT
$ DEFINE FOR041 SHARP.FMD
$ DEFINE FOR031 SHARP.CMD
$ RUN [ZIPF.MULSIM]MULNL
$ DEL FOR*.*;*

```

Figure 17.—MULSIM/NL execution command sequence for VAX minicomputer using VMS operating system.

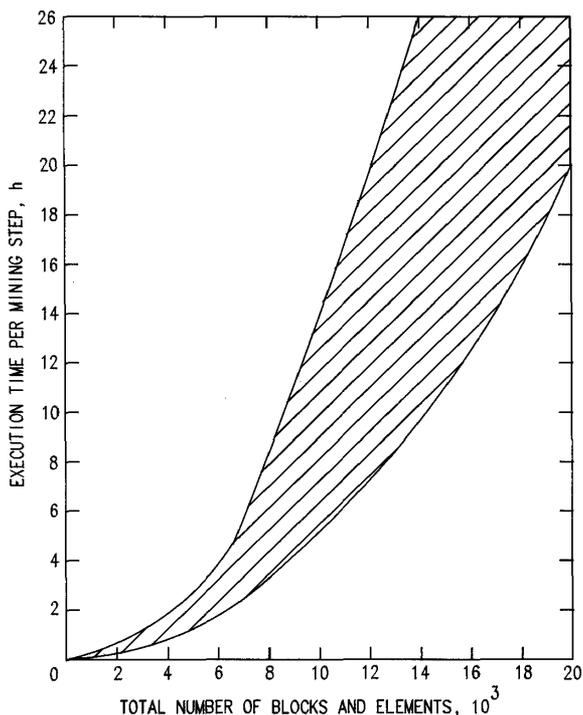


Figure 18.—Execution time on SUN III workstation versus problem size. Shaded area covers approximate possible range of computing time.

times can be adjusted for other computer systems if the relative performance of that system to a SUN III is known approximately. Otherwise, it is extremely easy to generate another, given any particular system. Knowing the size of a problem in terms of the total number of elements and blocks in that problem, figure 18 provides an estimate of the execution time per mining step. The lower bound time estimate is for simple, linear elastic problems, whereas the upper bound time estimate is for highly complex, nonlinear elastic problems. Again, the user must regard these time estimates as very approximate, but generally conservative.

OUTPUT FILES FROM MULSIM/NL

As discussed in the prior section, MULSIM/NL produces three distinct output file types after a successful problem run: a printed output file, a coarse-mesh data file, and a fine-mesh data file. The fine- and coarse-mesh data files contain the calculated stresses, displacements, and energies for each element and block and for each mining step in the problem. However, each seam has its own fine- and coarse-mesh data files. During execution of MULSIM/NL, the user should label these output files with appropriate file name extensions, such as PRT for the print file, CMD for the coarse-mesh data file, and FMD for the fine-mesh data file.

PRINTOUT FILE

MULSIM/NL writes a printout file that basically reflects all the input data and problem control variables. First, the printout shows the rock mass material properties, the in-seam material properties, the primitive stress field, basic model control and model data, location, and orientation of each seam, and finally the program control parameters. Next, the fine- and coarse-mesh mine models are printed for each seam and for the first mining step. These models show the mine geometry and material specifications. After MULSIM/NL finds a stress and displacement solution for the first mining step, it prints out certain vital statistics on the equation-solving process such as the number of iterations required for convergence and the approximate solution accuracy. For each additional mining step, MULSIM/NL will printout the new mine model and its solution statistics. Last and most important, the MULSIM/NL print file provides the energy calculations for the model. These calculations include various total energy release quantities and the energy release per unit area mined (i.e., the energy release rate). As mentioned earlier, the energy calculations require stress and displacement *changes* as input; therefore, the problem must have at least two mining steps. MULSIM/NL energy release calculations begin at the *second* mining step and continue thereafter.

FINE- AND COARSE-MESH OUTPUT DATA FILES

The voluminous output data files from MULSIM/NL contain calculated stresses, displacements, and energies. Both the coarse- and fine-mesh files have identical structure so that they are indistinguishable from one another

for subsequent postprocessing purposes. The first record of these ASCII files specifies five important parameters that define the rest of the structure of the file. Table 4 shows these variables and gives their meaning. The next NXY records give the block and/or element output data for the first mining step followed by additional groups of NXY records for subsequent mining steps. Thus, the total size of any output data file is $(MXSTEP * NXY + 1)$ records.

Table 4.—First record variables of coarse- and fine-mesh output data files

Variable	Format	Definition
MXSTEP ..	I10	Number of mining steps (≥ 1).
NXY	I10	Total number of coarse-mesh blocks or fine-mesh elements in model ($NXY = NX * NY$).
W	F10.2	Block or element width.
NX	I10	Number of blocks or elements in X direction.
NY	I10	Number of blocks or elements in Y direction.

Each data record within an output file contains 15 variables displayed in FORTRAN format (2F7.1, I3, 1X, 12E9.3). Table 5 lists these output variables and provides their meaning. For a more complete discussion of these variables and in particular the energy quantities, see the companion volume "MULSIM/NL - Theoretical and Programmer's Manual." In essence, the output files store X, Y, Z triplets ready for graphical display.

OUTPUT FILE EXAMINATION WITH POSTPROCESSOR MULPLT/NL

When studying various engineering problems with a numerical analysis program like MULSIM/NL, the prudent user will typically complete many different runs that systematically vary the problem geometry and material properties over an expected range. Such parametric studies will generate many MULSIM/NL output data files. Examining the numerous output files requires a simple and *fast* graphical postprocessor. Ideally, such a postprocessor also has a high degree of transportability between computer platforms. The Bureau developed postprocessor called MULPLT/NL provides a simple, very fast graphical means to examine MULSIM/NL output files.

Table 5.—Variables in coarse- and fine-mesh output data file records

Variable	Format	Definition
X	F7.1	Global X coordinate of element or block centroid.
Y	F7.1	Global Y coordinate of element or block centroid.
MATCOD	13(1X)	Material code for element or block.
DX	E9.3	X displacement component—ride 1.
DY	E9.3	Y displacement component—ride 2.
DZ	E9.3	Z displacement component—normal closure.
SX	E9.3	X stress component—shear stress 1.
SY	E9.3	Y stress component—shear stress 2.
SZ	E9.3	Z stress component—normal stress.
TSE	E9.3	Total strain energy into element or block.
RSE	E9.3	Recoverable strain energy from element or block.
DSE	E9.3	Dissipated strain energy in element or block (DSE = TSE - RSE).
KER	E9.3	Kinetic energy release for element or block.
SER	E9.3	Strain energy release for element or block.
TER	E9.3	Total energy release for element or block (TER = KER + SER).

MULPLT/NL is a menu-driven, graphics program that can display pseudo-three-dimensional plots or two-dimensional cross sections of the stresses, displacements, and energies contained in the MULSIM/NL output files. With MULPLT/NL, the user interactively specifies the data file name and mining step number to examine. The user then selects a plot type (three dimensional or two

dimensional) and chooses a quantity from the output file to display - either stress, displacement, or energy. MULPLT/NL can also plot *changes* in these quantities between any two specified mining steps. This capability enables direct comparisons between numerical models, which tend to compute total or absolute stresses and displacements, and the field programs, which tend to measure their changes. MULPLT/NL can calculate and display stress and displacement changes between mining steps, thus allowing the user to compare numerical results directly (with due caution) to field measurements of stress and displacement changes.

MULPLT/NL also permits the user to adjust the scale on all plot types so that comparisons are not distorted. In addition, the user can save the raw data comprising any particular graph created by MULPLT/NL as an ASCII file. Afterwards, a wide variety of commercially available software packages, such as a computer-aided-drafting program, can import these files and create presentation-quality graphical displays from these subsets of the MULSIM/NL output data.

Aside from producing easy-to-read, graphical displays of MULSIM/NL output data, the real advantage of MULPLT/NL lies in its speed and simplicity. Experience seems to show that for every presentation quality graph created from MULSIM/NL data, at least 100 similar plots are examined quickly with MULPLT/NL during the course of numerical experiments. Thus, the simple post-processor MULPLT/NL serves a vital function in a well-executed numerical modeling study.

MULSIM/NL PRACTICAL EXAMPLES

Earlier sections of this manual presented the general capabilities of MULSIM/NL and gave an overview of the modeling process with this three-dimensional BEM program. A later section discussed the input file structure and all the input variables in great detail. Another section covered actual execution of MULSIM/NL. The last section described the output from the program and discussed the postprocessor for graphically viewing the output data.

This section provides three detailed examples of successful MULSIM/NL analyses and includes the actual input files and sample graphical outputs produced by the postprocessor MULPLT/NL. Experience shows that actual examples often provide the best means to illustrate the use of a program. However, while these examples stem from current Bureau projects using MULSIM/NL, they do not represent a complete analysis and solution to the problems depicted. MULSIM/NL analyses are but a small part of the overall engineering design process, and

these examples merely seek to illustrate the "how to" aspect of those analyses.

LONGWALL MINING EXAMPLE

This first example is of a unique longwall panel that extracts one of the headgate pillars as the panel advances. Figure 19 shows the input file control section for this two-step problem. This example illustrates use of the program with the pound per square inch and inches unit system. All stress and modulus values have units of pounds per square inch and all lengths have units of inches. Eight in-seam materials are defined as follows: A is linear elastic coal at 0% extraction; B is linear elastic gob; C is also linear elastic coal, but at 45% extraction, and D is elastic-plastic coal yielding at 500 psi. Materials E, F, G, and H represent linear elastic coal at 20%, 40%, 60%, and 80% extraction; however, they remain unused in this model.

```

LONGWALL EXAMPLE
0.1500000.900000E+06      1
      8
1.00.500E+06.217E+06.000E+00.000E+00.000E+00
6.00.500E+04.179E+04.100E+01.000E+00.000E+00
1.00.275E+06.119E+06.000E+00.000E+00.000E+00
3.00.500E+03.250E-02.000E+00.300E+00.000E+00
1.00.400E+06.174E+06.000E+00.000E+00.000E+00
1.00.300E+06.130E+06.000E+00.000E+00.000E+00
1.00.200E+06.870E+05.000E+00.000E+00.000E+00
1.00.100E+06.430E+05.000E+00.000E+00.000E+00
0.0.0458      0.0.0000      0.0.0000      0.0.0458      0.0.0000      0.0.0917
300.00      50      32      30      45      9      24
0.0      0.0 -6000.0      72.0
1.00000 0.00000 0.00000 0.00000 1.00000 0.00000 0.00000 0.00000 1.00000
1.35      50.00      100      1      2

```

Figure 19.—Control section of input file for longwall example.

Some other noteworthy values include the block width at 300 in (which implies an element width of 60 in), a seam thickness of 72 in and a seam depth of -6,000 in (-500 ft). The seam has a horizontal orientation. Stresses increase at a rate of 0.0917 psi/in of depth (1.1 psi/ft) vertically and .0458 psi/in (0.55 psi/ft) horizontally.

Figure 20 shows the fine-mesh for mining step 1. Solid coal exists on either side of the headgate pillars. This step serves as the basis from which stress changes are computed as mining extracts the diagonal pillar. Elastic-plastic coal, which yields at 500 psi, forms the skin of each pillar while all other coal remains linear elastic. Figure 21 shows the coarse-mesh for this step. The fine-mesh encompasses the headgate area somewhat right and center within the coarse-mesh. Solid coal exists on either side of the headgate, while gob (material B) is on one side of the tailgate. Material C (linear elastic coal at 45% extraction) comprises the headgate and tailgate areas.

Figure 22 shows the fine-mesh area for mining step 2, which has extracted the upper left portion of the model and replaced it with gob (material B). Figure 23 shows the coarse-mesh for this step. Note how the longwall panel has advanced and where gob (material B) has replaced linear elastic coal (material A).

Running this two-step example consumed about 3 h on a SUN Engineering workstation. Next, the postprocessor MULPLT/NL was used to produce the following plots from the fine-mesh output file. Figure 24 (top) shows normal stresses in pound per square inch around the

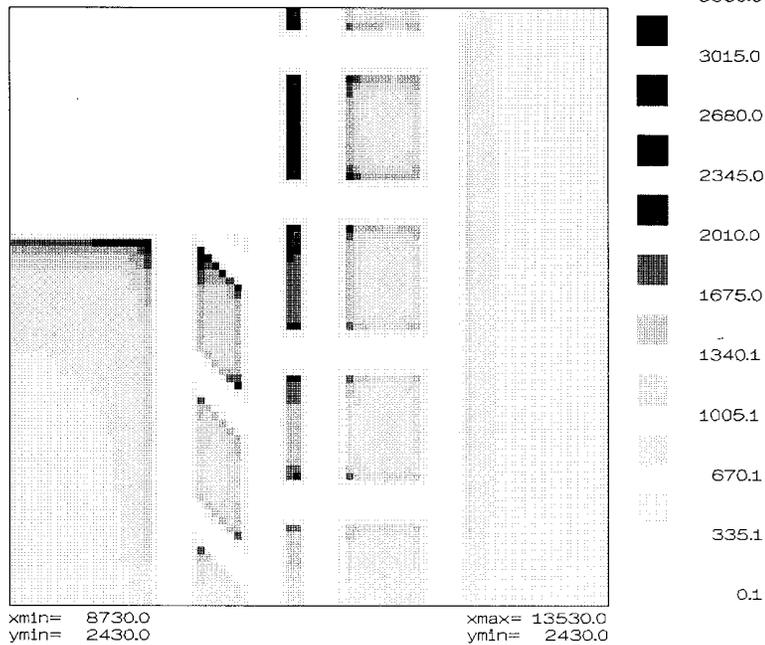
headgate area, while figure 24 (bottom) shows displacements in inches. These are total stresses and displacements and not changes, although the latter can be requested. Figure 25 is a profile of normal stresses at a cross section just ahead of the face, while figure 26 shows displacements along a section just behind the face. Again, interpretation of these calculations are neither attempted nor implied in this example.

MULTIPLE-SEAM MINING EXAMPLE

The second example demonstrates an interaction analysis between room-and-pillar mining of an upper seam and longwall development in a lower seam. Figure 27 shows the input file control section for this example. Only one step is considered in what should become a multiple step problem. This example illustrates use of the program with the megapascal and meter unit system. Three in-seam materials are defined as follows: A is linear elastic coal at 0% extraction; B is linear elastic coal at 50% extraction; and C is linear elastic gob. Vertical stress increases at a rate of 0.025 MPa/m, while all other stress components remain zero. The block width is 75 m implying an element width of 15 m. Seam thicknesses are 2 m for the upper seam and 3 m for the lower. The seams are horizontal and lie at depths of -560 and -600 m.

Figure 28 shows the fine-mesh for the upper seam where partial extraction has occurred. Areas of no extraction (A) are interlaced with areas of partial extraction (B).

USBM MULPLT OF longwall.fmd
 PILLAR RECOVERY PANEL
 TOTAL NORMAL STRESS-Z DIRECTION MINING STEP 2
 xmin= 8730.0 xmax= 13530.0
 ymax= 7230.0 ymax= 7230.0



USBM MULPLT OF longwall.fmd
 PILLAR RECOVERY PANEL
 TOTAL CLOSURE-Z DIRECTION MINING STEP 2
 xmin= 8730.0 xmax= 13530.0
 ymax= 7230.0 ymax= 7230.0

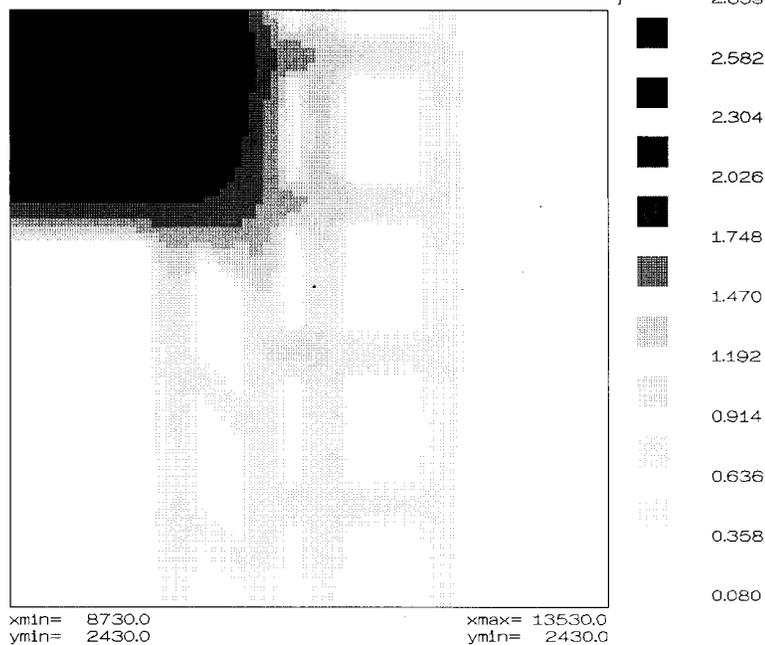


Figure 24.—Longwall example—normal stresses (top) and normal displacements (bottom) around headgate during mining step 2. Stresses are in pounds per square inch, and displacements are in inches. Postprocessor program MULPLT/NL generates these computer drawn plots.

USBM MULPLT OF longwall.fmd
PILLAR RECOVERY PANEL
TOTAL NORMAL STRESS-Z DIRECTION MINING STEP 2
SECTION PARALLEL TO X - ROW 45

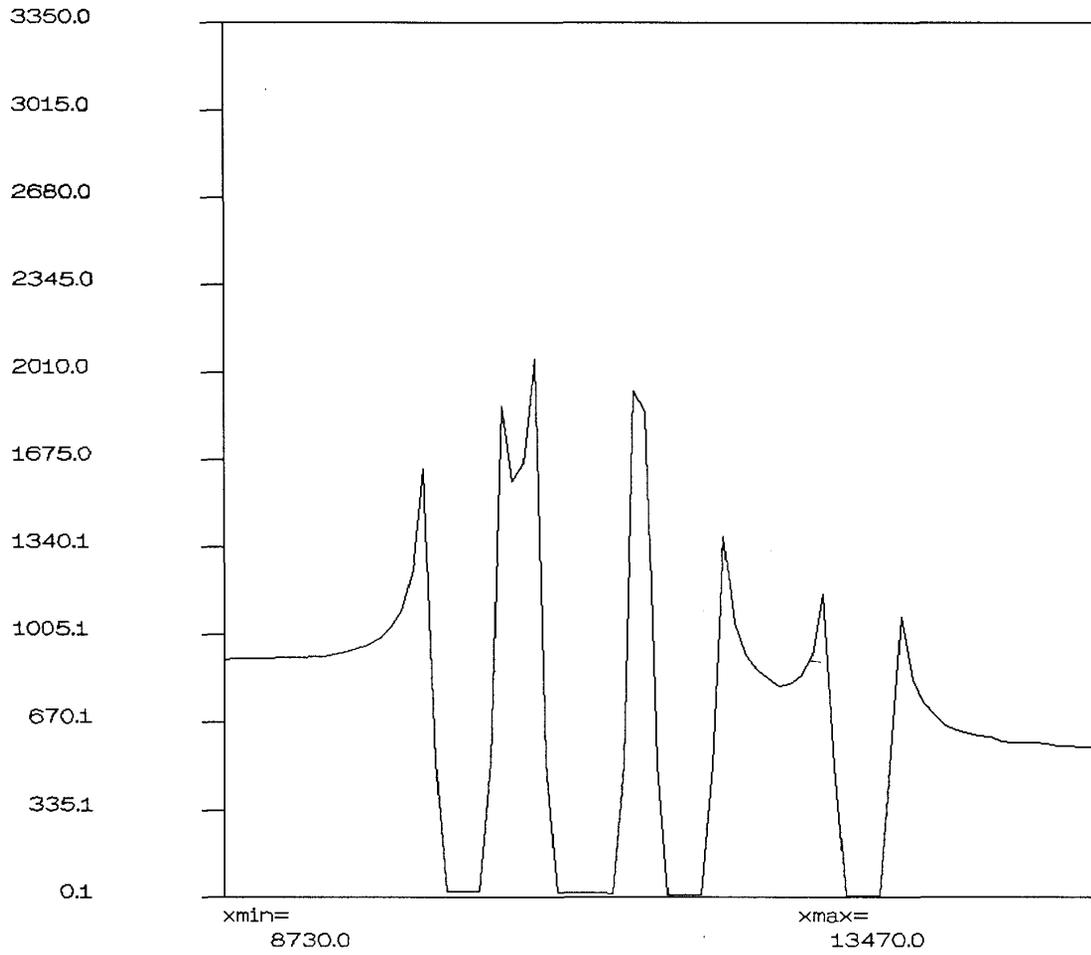


Figure 25.—Longwall example—normal stresses along cross section just ahead of face during mining step 2. Stresses are in pounds per square inch. Postprocessor program MULPLT/NL generates this computer drawn plot.

USBM MULPLT OF longwall.fmd
PILLAR RECOVERY PANEL
TOTAL CLOSURE-Z DIRECTION
SECTION PARALLEL TO X - ROW

MINING STEP 2
51

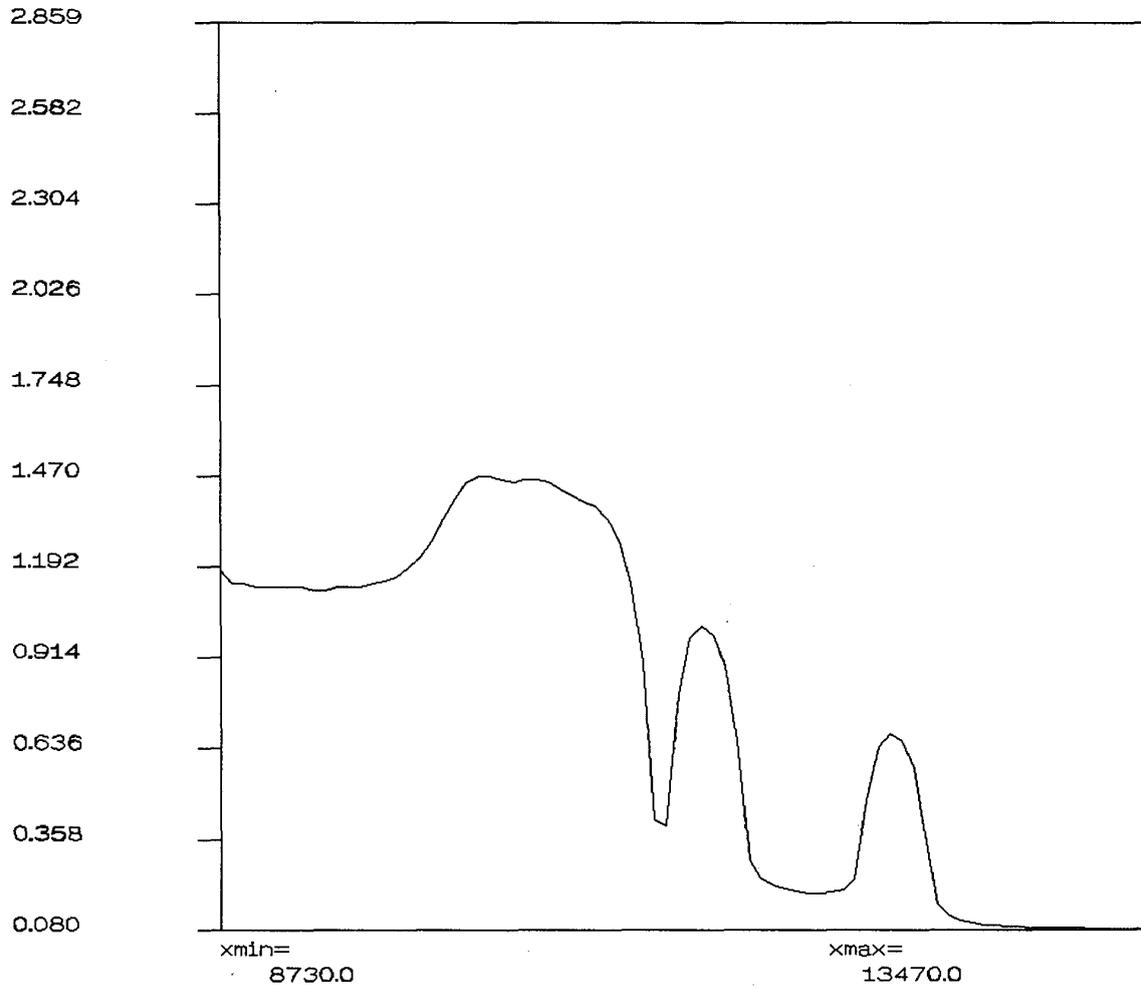


Figure 26.—Longwall example—normal displacements along cross section just behind face during mining step 2. Displacements are in inches. Postprocessor program MULPLT/NL generates this computer drawn plot.

MULTIPLE SEAM MINING EXAMPLE

```

0.2500000.200000E+05      2
  3
  1.00.500E+03.200E+03.000E+00.000E+00.000E+00
  1.00.250E+03.100E+03.000E+00.000E+00.000E+00
  6.00.500E+02.200E+02.400E+01.000E+00.000E+00
  0.0.0000  0.0.0000  0.0.0000  0.0.0000  0.0.0000  0.0.0250
75.00      30      26      6      25      6      21
  0.0      0.0  -560.0    2.0
  0.0      0.0  -600.0    3.0
1.00000  0.00000  0.00000  0.00000  1.00000  0.00000  0.00000  0.00000  1.00000
  1.35      .20      100      1      1
  
```

Figure 27.—Control section of input file for multiple-seam mining example.

The mine geometry lies askew to the grid orientation. One small area of full extraction (gob - C) appears in the lower right of the fine-mesh.

Figure 29 shows the fine-mesh for the lower seam where several longwall panels have been developed. This model covers a huge seam area. Again, A represents solid coal whereas B, used to define the gateroad areas, is 50% extracted coal. Small open areas (1) exist at the ends of each panel. For the equation solver in MULSIM/NL to work properly, a small amount of open area (1) *must* exist in each model. Finally, figures 30 and 31 show the coarse mesh for the upper and lower seams.

Running this example consumed several hours of SUN III computing time. Next, MULPLT/NL was invoked to examine the fine-mesh output files for each seam. The following graphics stem directly from MULPLT/NL. Figure 32 shows the normal stresses in megapascals on each seam and clearly illustrates the stress interactions between the seams. Similarly, figure 33 shows normal displacements for each seam and the seam interactions. Again, these calculation results are an example only, and interpretations are not intended at this time.

RANDOM ROOM-AND-PILLAR MINING EXAMPLE

The third and final example shows a highly irregular mine geometry. Furthermore, the tabular orebody dips down the page in the model. Figure 34 shows the input file control section. This example illustrates use of the program with the pound per square inch and inches unit system. Four in-seam materials are defined starting with

A for linear elastic rock at 0% extraction. Materials B, C, and D are also linear elastic rock, but at 75, 50, and 25% extraction, respectively. The actual tabular orebody dips at 20° into a steeply rising hillside. To account for the orebody dip, the direction cosines specify that the orebody plane dips about 20° down the page. To account for the rising hillside, an artificially high stress gradient is used. Vertical stress increases at 0.2342 psi/in (2.81 psi/ft), while horizontal stresses increase at one-fourth this gradient. (If the topography was flat, then the vertical stress gradient would be about 1.1 psi/ft.) The origin of the orebody is at a depth of -6,744 in (562 ft), and the orebody thickness is 240 in. This example has three mining steps.

Figures 35 and 36 show the fine- and coarse-mesh for the first step. This step provides calculations of the pre-mining stress and displacement fields. Figures 37 and 38 show the fine- and coarse-mesh for the mine geometry in the second mining step. Note the highly irregular mine geometry undergoing analysis. Finally, figures 39 and 40 show the fine- and coarse-mesh geometry for the third mining step. In this step, additional extraction has occurred in the lower, central region of the fine-mesh.

Running this example consumed about 8 h of SUN III computing time. Figure 41 shows stress and displacement plots produced by MULPLT/NL for the second mining step. At the edges of the model, stresses clearly increase with depth as one moves down the page. As before, these calculations are for illustrative purposes only and are not meant for interpretation at this time.

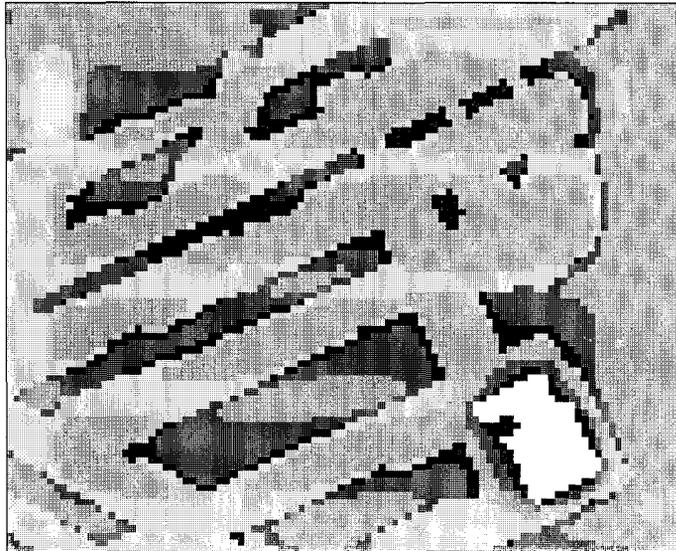
AAAAA... (A large block of text representing a fine-mesh section of an input file, consisting of many rows of characters, likely a grid of 'A's and 'B's.)

Figure 29.—Seam 2 (lower seam) fine-mesh section of input file for multiple-seam mining example.


```

USBM MULPLT OF multism.fm1
MULTIPLE SEAM LONGWALL PANELS
TOTAL NORMAL STRESS-Z DIRECTION   MINING STEP 1
xmin= 382.4                      xmax= 1882.4
ymax= 1582.4                     ymax= 1582.4

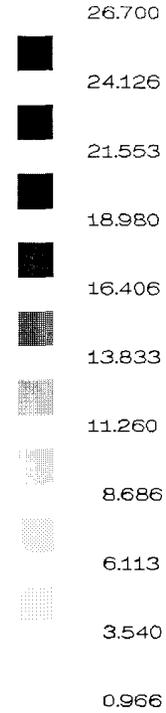
```



```

xmin= 382.4                      xmax= 1882.4
ymin= 382.4                      ymin= 382.4

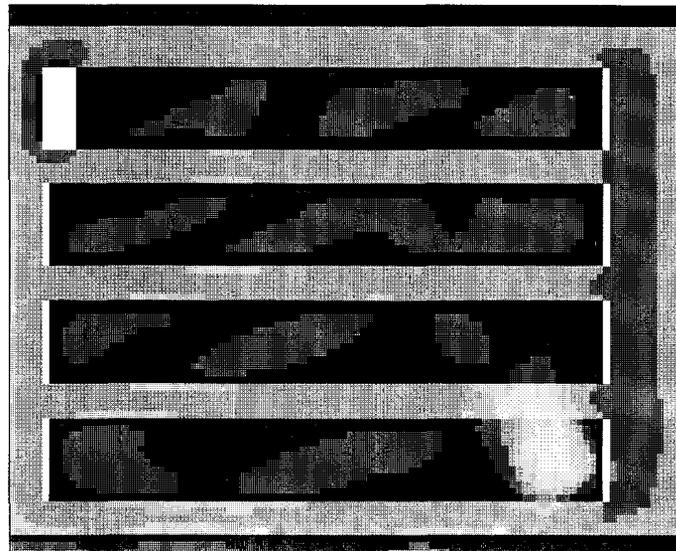
```



```

USBM MULPLT OF multism.fm2
MULTIPLE SEAM LONGWALL PANELS
TOTAL NORMAL STRESS-Z DIRECTION   MINING STEP 1
xmin= 382.4                      xmax= 1882.4
ymax= 1582.4                     ymax= 1582.4

```



```

xmin= 382.4                      xmax= 1882.4
ymin= 382.4                      ymin= 382.4

```

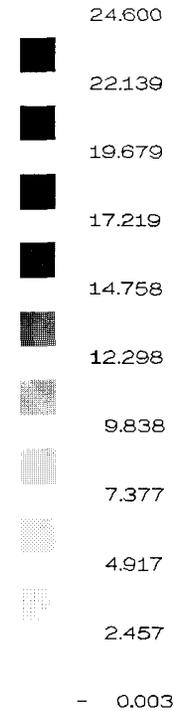


Figure 32.—Multiple-seam mining example—normal stresses. Top, Upper seam; bottom, lower seam. Stresses are in megapascals. Postprocessor program MULPLT/NL generates these computer drawn plots.

RANDOM ROOM AND PILLAR MINING EXAMPLE

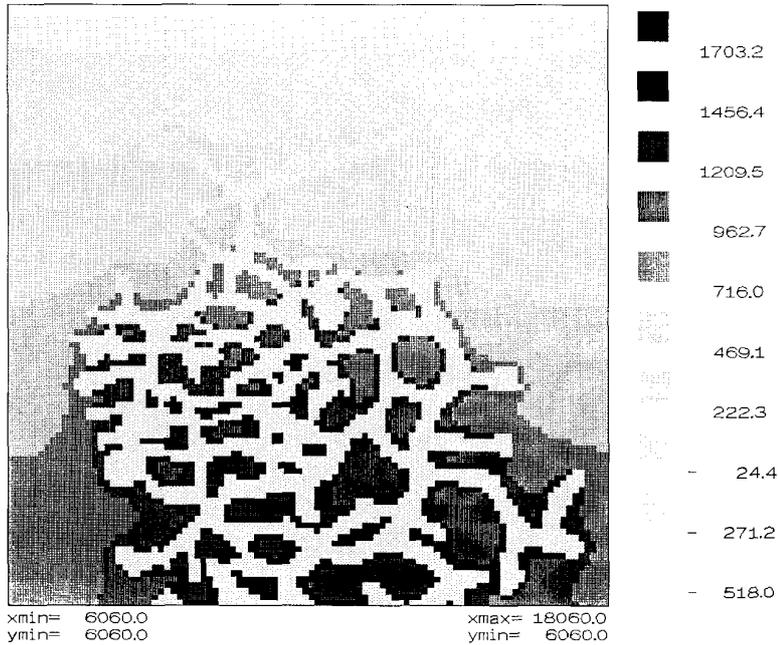
```

0.2500000.600000E+07      1
  4
  1.00.200E+07.800E+06.000E+00.000E+00.000E+00
  1.00.500E+06.200E+06.000E+00.000E+00.000E+00
  1.00.100E+07.400E+06.000E+00.000E+00.000E+00
  1.00.150E+07.600E+06.000E+00.000E+00.000E+00
  0.0.0583      0.0.0000      0.0.0000      0.0.0583      0.0.0000      0.0.2342
600.00      40      40      11      30      11      30
  0.0      0.0 -6744.0      240.0
-1.00000 0.00000 0.00000 0.00000-0.92718 0.37461 0.00000 0.37461 0.92718
  1.35      50.00      100      1      3

```

Figure 34.—Control section of input file for random room-and-pillar mining example.

USBM MULPLT OF rndrmpil.fmd
 RANDOM ROOM AND PILLAR MINING
 TOTAL NORMAL STRESS-Z DIRECTION MINING STEP 2
 xmin= 6060.0 xmax= 18060.0
 ymax= 18060.0 ymin= 6060.0



USBM MULPLT OF rndrmpil.fmd
 RANDOM ROOM AND PILLAR MINING
 TOTAL CLOSURE-Z DIRECTION MINING STEP 2
 xmin= 6060.0 xmax= 18060.0
 ymax= 18060.0 ymin= 6060.0

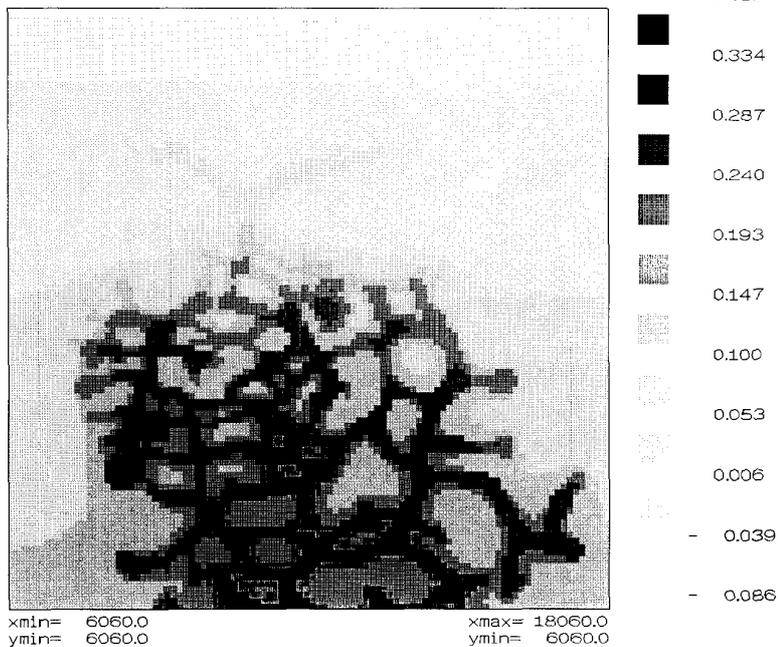


Figure 41.—Random room-and-pillar mining example—normal stresses (top) and normal displacements (bottom) during mining step 2. Stresses are in pounds per square inch, and displacements are in inches. Postprocessor program MULPLT/NL generates these computer drawn plots.

SUMMARY

This manual describes MULSIM/NL, which is a boundary-element program for calculating stresses and displacements in coal seams or thin tabular vein-like deposits. The manual gives detailed operating instructions for MULSIM/NL and then it illustrates its use with several practical examples. While this manual concentrates on the practical aspects of actually running and using MULSIM/NL, another related document titled "MULSIM/NL - Theoretical and Programmer's Manual" provides mathematical and programming details to those engineers and programmers who need to fully understand the FORTRAN program or desire to alter and enhance it.

MULSIM/NL calculates stresses and displacements throughout a coarse- and fine-mesh modeling grid. It can follow any one of six linear and nonlinear stress-strain relationships. A multiple mining step capability enables correct simulations of a changing mine geometry. In addition, MULSIM/NL performs energy release rate calculations that follow Salamon's (1) theoretical work.

This manual discusses the basic modeling process in which an actual mine plan is first idealized, then approximated by a coarse- and fine-mesh modeling grid. The in-seam material that make up this array of blocks and/or elements can follow any one of MULSIM/NL's six different stress-strain laws. Understanding the basic modeling process enables a user to create an input file for program MULSIM/NL. This input consists of a control section, fine-mesh geometry sections, and coarse-mesh geometry sections. The control section defines all the basic model

generation and operating parameters for MULSIM/NL such as problem size, material properties, in situ stress fields, and seam orientation. The coarse-mesh section defines the mine geometry and assigns material properties in an array of coarse-mesh blocks that comprise the overall mine model. The fine-mesh section defines geometry and assigns properties to a subarray of fine-mesh elements within the coarse-mesh.

The command sequence to actually run MULSIM/NL depends heavily on the host computer and its operating system. This manual gives example command sequences for an HP Engineering Workstation with a UNIX operating system and a VAX computer with the VMS operating system. Successful execution of MULSIM/NL produces large output data files containing calculated stresses, displacements, and energies for each mining step. The postprocessor MULPLT/NL provides a simple, very fast graphical means to examine these voluminous MULSIM/NL output files. MULPLT/NL is a menu-driven, graphics program that can display pseudo-three-dimensional plots or two-dimensional cross sections of the calculated stresses, displacements, and energies.

Last, the manual gives three complete examples that illustrate how to run MULSIM/NL. These examples show the complete input file for MULSIM/NL and various sample plots of the output data generated by the postprocessor MULPLT/NL. The discussions of the input file and these examples will provide the potential user with sufficient instructions to use MULSIM/NL for new purposes.

REFERENCES

1. Salamon, M. D. G. Energy Considerations in Rock Mechanics: Fundamental Results. *J. S. Afr. Inst. Min. Metall.*, v. 84, No. 8, 1984, pp. 233-246.
2. Beckett, L. A., and R. S. Madrid. MULSIM/BM - A Structural Analysis Computer Program for Mine Design. BuMines IC 9168, 1988, 302 pp.
3. Sinha, K. P. Displacement Discontinuity Technique for Analyzing Stresses and Displacements Due to Mining in Seam Deposits. Ph.D. Thesis, Univ. MN, 1979, 311 pp.
4. Crouch, S. L., and C. Fairhurst. The Mechanics of Coal Mine Bumps and the Interaction Between Coal Pillars, Mine Roof, and Floor (contract H0101778, Univ. MN). BuMines OFR 53-73, 1973, 88 pp.; NTIS PB 222 898.
5. Heasley, K. A., and K. Barron. A Case Study of Gate Pillar Response to Longwall Mining in Bump-Prone Strata. Paper in Proceedings of Longwall USA, Pittsburgh, PA, 1988, Industrial Presentation, Inc., Aurora, CO, 1988, pp. 91-105.
6. Cook, N. G. W., E. Hoek, J. P. G. Pretorius, W. D. Ortlepp, and M. D. G. Salamon. Rock Mechanics Applied to the Study of Rock Bursts. *J. S. Afr. Inst. Min. Metall.*, v. 66, No. 10, 1966, pp. 435-528.
7. Wang, F. D., W. A. Skelly, and J. Wolgamott. In Situ Coal Pillar Strength Study (contract H0242022, CO Sch. Mines). BuMines OFR 107-79, 1976, 243 pp.
8. Iannacchione, A. Behavior of a Coal Pillar Prone to Burst in the Southern Appalachian Basin of the United States. Paper in Proceedings of Second International Symposium of Rockbursts and Seismicity in Mines. Univ. MN, June 1988, pp. 427-439.
9. Dahlquist, G., and A. Bjork. Numerical Methods. Prentice-Hall, 1974, 571 pp.

APPENDIX.—SPECIAL INSTRUCTIONS FOR PERSONAL COMPUTER VERSION OF MULSIM/NL

OVERVIEW OF MULSIM/NL ON A PC

This appendix provides additional information necessary to run MULSIM/NL on a personal computer (PC). The U.S. Bureau of Mines downloaded this program to the PC to make it more readily available to the mining industry. In addition, the Bureau developed preprocessor and post-processor programs to help create input files and examine graphical output from MULSIM/NL in a PC environment. The PC-based version of MULSIM/NL contains three parts: The main BEM analysis program called MULNLPC, a simple preprocessor called MULPREPC, and a graphical postprocessor called MULPLTPC. Functionally and operationally, the PC version of MULSIM/NL is almost identical with its predecessors that run in a mainframe or workstation environment. MULNLPC and MULPLTPC have small and large versions for use on PC's with 4 or 8 Mb of random access memory (RAM). The smaller version can handle a model with 110 by 110 fine-mesh elements, whereas the larger version has capacity for 150 by 150 fine-mesh elements.

Minimum hardware requirements for the PC version of MULSIM/NL include (1) a 386 processor with a 387 math coprocessor, (2) 4 Mb of RAM for the small memory version and 8 Mb of RAM for the large memory version, and (3) a VGA or SVGA color monitor. In addition, if hardcopies of the color output from the postprocessor MULPLTPC are desired, software with a screen capture utility and a rasterizing color printer are required. Various software vendors market programs with screen capture utilities, including Pizazz Plus, PC Paintbrush, and Corel Draw. Similarly, many color raster printers such as the Hewlett-Packard Paintjet or the Tektronix Color Image Printers are available.

In developing the PC version of MULSIM/NL, Bureau researchers used the Microway Inc., Kingston, MA, NDP FORTRAN Compiler to generate an executable of MULNLPC, MULPREPC, and MULPLTPC. This compiler contains a royalty-free disk operating system (DOS) memory extender that gives the programs access to memory beyond the normal 640 Kb. This memory extender is linked into the executable. Other loaded memory managers, memory extenders, and memory resident programs will interfere with the Bureau programs! The Microway compiler also has its own royalty-free graphics library that was used to generate the graphics in MULPLTPC.

Before using the PC version of MULSIM/NL, the CONFIG.SYS file may require slight modification to include these lines.

FILES=30
BUFFERS=32

Make sure that other memory managers, memory extenders, and memory resident programs in the CONFIG.SYS and/or the AUTOEXEC.BAT files are disabled. After making these kinds of modifications, reboot the system, and then the MULSIM/NL programs are ready to run on the PC.

CREATING AN INPUT FILE WITH THE PRE- PROCESSOR PROGRAM MULPREPC

An earlier section of this report, "INPUT FILE STRUCTURE FOR MULSIM/NL," discusses the three parts to a MULSIM/NL (or MULNLPC) input file, namely the control section, the fine-mesh geometry, and the coarse-mesh geometry as shown in figure 7. The preprocessor MULPREPC helps generate the control section of this input file. The control section defines all the basic variables for MULSIM/NL, such as problem size, material properties, in situ stress fields, and seam orientation. MULPREPC creates a MULSIM/NL input file and places these variables in their proper position, freeing the user from this tedious task. MULPREPC only makes the control section of the input file. The user must then use a text editor or word processor to create the fine-mesh and coarse-mesh sections of the MULSIM/NL input file.

The PC version of MULSIM/NL has the same input parameters as described earlier in this report, with two minor exceptions. First, the PC version is restricted to *two* seams. Therefore, the variable NSEAM, defined in record 2 of the input file, must be the integer 1 or 2. Second, the fine-mesh size is restricted to 110 by 110 fine-mesh elements (or 22 by 22 fine-mesh blocks) in the small memory version of the programs. Therefore, the variables IFXS, IFXE, IFYS, and IFYE must be chosen according to $IFXE - IFXS + 1 \leq 22$ and $IFYE - IFYS + 1 \leq 22$. The larger memory version can still handle a fine-mesh size of 150 by 150 elements, as in the workstation or mainframe versions of the program. In both the small and large memory versions, maximum coarse-mesh size remains the same at 50 by 50 blocks.

MULPREPC is run by changing to the appropriate directory and typing the command MULPREPC followed by return. At that point, use of the program becomes self-explanatory. After the user chooses a consistent unit system, the program prompts for all the requisite MULSIM/NL input parameters using typical default values for the

chosen unit system. For a full discussion of the requested input variables, refer to the section of this report titled "INPUT FILE STRUCTURE FOR MULSIM/NL."

RUNNING MAIN PROGRAM MULNLPC

An earlier section of this report, "RUNNING MULSIM/NL," discusses the files used by MULSIM/NL and gives examples of the computer operating commands necessary to run the program in either a mainframe or a workstation environment. The subsection on "MULSIM/NL FILES" along with table 3 give details on all input, printout, coarse-mesh data, and fine-mesh data files used and created by the main program. MULNLPC uses the same files as the mainframe and workstation versions with one minor exception. In the PC version, FORTRAN unit 07 contains the basic program printout as opposed to FORTRAN unit 06. The PC version also sends some printed output directly to the computer screen, which is defined as FORTRAN unit 06. This output consists of a disclaimer and data for each equation solver iteration as the program converges on a solution. With these iteration data, the user can monitor progress of model.

Running MULNLPC requires the following .BAT files. For a single-seam model, create a file called RUNMULNL.BAT that contains the following DOS commands:

```
MULNLPC < %1.INP
REN FOR07.DAT %1.PRT
REN FOR31.DAT %1.CMD
REN FOR41.DAT %1.FMD
DEL FOR*.*
```

Next, create a file called RUNALL.BAT that contains the following command:

```
CALL RUNMULNL.BAT TESTA.
```

Finally, by typing the command RUNALL (return), the PC will execute the command line in the RUNALL.BAT file, which in turn executes the RUNMULNL.BAT file with the argument TESTA substituted for %1. RUNMULNL will then execute MULNLPC using the file TESTA.INP as input. After the program has converged to a solution, it will rename the printed output file TESTA.PRT, the coarse-mesh data file TESTA.CMD, and the fine-mesh data file TESTA.FMD. Finally, it will delete any remaining files of the form FOR*.* before returning control of the PC back to the user.

With multiple jobs, the file RUNALL.BAT might contain the following commands:

```
CALL RUNMULNL.BAT TESTA
CALL RUNMULNL.BAT TESTB
*
*
etc.
```

With this batch file, the user can solve many MULNLPC jobs sequentially.

For a two-seam model, the RUNMULNL.BAT file should contain the following DOS commands:

```
MULNLPC < %1.INP
REN FOR07.DAT %1.PRT
REN FOR31.DAT %1.CM1
REN FOR32.DAT %1.CM2
REN FOR41.DAT %1.FM1
REN FOR42.DAT %1.FM2
DEL FOR*.*
```

For a two-seam job with the argument TEST2, this batch file will execute MULNLPC using the file TEST2.INP as input. When it is done, the file TEST2.PRT will contain the printed output, and the files TEST2.CM1, TEST2.CM2, TEST2.FM1, and TEST2.FM2 will contain coarse- and fine-mesh data for seams 1 and 2, respectively.

Execution times with MULNLPC depend on many factors including the machine itself, size of the problem, and degree of nonlinearity in the model. Typical problems with a 120 by 120 fine mesh and some nonlinear elements can require 24 h of computing time with a 386 system.

EXAMINING OUTPUT FILES WITH POST-PROCESSOR PROGRAM MULPLTPC

The program MULPLTPC on a PC operates the same as MULPLT on a mainframe or workstation; therefore, the previous subsection on "OUTPUT FILE EXAMINATION WITH THE POSTPROCESSOR MULPLT/NL" also applies to MULPLTPC. This postprocessor provides a simple very fast graphical means to examine fine- and coarse-mesh data files from MULNLPC. It is menu-driven and can display psuedo-three-dimensional color plots or two-dimensional cross sections of the calculated stresses, displacements, and energies. The previously mentioned subsection provides greater detail on the capabilities of MULPLTPC.

To run **MULPLTPC**, the user should first change to the appropriate directory and then type the command **MULPLTPC** (return). The rest is self-explanatory. As with the main program **MULNLPC**, a small and large version exists for use on systems with either 4 or 8 Mb of RAM. The same limits on the number of fine-mesh

elements apply. To obtain a hardcopy of the color output from **MULPLTPC**, the user must first install appropriate software with a screen capture utility that can drive the available rasterizing color printer. The previous section on "**MULSIM/NL PRACTICAL EXAMPLES**" shows hardcopy of output produced by this postprocessor.