Simplified Pre- and Post-processing Technique for Performing Finite-element Analyses of Deep Underground Mines

Marc T. Filigenzi, Mechanical Engineer
National Institute for Occupational Safety and Health
Spokane Research Center
Spokane, WA

ABSTRACT

Two of the major ground control safety issues confronting underground mine operations today are shaft pillar stability and the failure of rock around active mine openings. Failure of a mine shaft can lead to the entrapment of workers. Failure of rock around active underground mine openings can lead to roof falls, which in turn can result in worker injuries and fatalities.

Finite-element analysis has proven to be a reliable method for predicting stresses and displacements around underground mine openings. However, this is a complex and time-consuming technique and is not used as often as it could be in mine planning. The purpose of this paper is to demonstrate one technique developed at the Spokane Research Center that allows the user to create a finite-element model of a two-dimensional section of an underground mine in a relatively straightforward manner. This model is then used to calculate stresses, displacements, and safety factors around mine openings. With this information, mine planners can evaluate the stability of mine openings as well as the stability of pillars and mine shafts. This analysis will help develop mining sequences and layouts that minimize stresses in that section of the mine. This, in turn, should minimize the occurrence of shaft failure, roof falls, and other hazards associated with underground mining.

INTRODUCTION

The prediction of stresses and displacements around underground mine openings has proven to be a difficult challenge. However, the rewards of such predictions can be substantial. If a mine planner knows that a particular sequence or configuration of excavations will lead to unacceptably large displacements in the rock mass, and that these displacements may affect shaft stability, the planner can modify that sequence or configuration in order to maintain the integrity of the shaft. Such modifications can also minimize stresses in and around mine openings, which, in turn, can reduce the occurrence of roof falls and other hazards associated with the failure of rock.

Finite-element analysis has proven to be an effective method for predicting stresses in rock mass caused by underground mining *(1)*. These earlier studies relied upon the UTAH2PC finite-element program for the analysis of underground mining near a mine shaft. The UTAH2PC code can analyze a two-dimensional section of an underground mine in which there are several formations having anisotropic material properties. The program can also analyze multiple cut-and-fill operations.

However, UTAH2PC does not have an automatic finite-element mesh generator. Each node of the finite-element mesh must be manually located, and each element must be explicitly defined. This leads to a tremendous amount of labor and bookkeeping for an analyst using this program, as well as a reduction in the flexibility of the analysis. If the analyst wishes to modify the geometry of the mine openings or geology, the nodes and elements of the finite-element mesh must be manually relocated and redefined, or the mesh must be regenerated.

Many finite-element analysis programs are available today that include automatic mesh generators and intuitive, user-friendly interfaces (ANSYS, COSMOS/M, PATRAN, etc.). These programs will automatically create a finite-element mesh with all the necessary nodes and elements. However, many of these programs are intended for mechanical engineering design and are not suitable for models possessing the anisotropic material properties or brittle-failure criteria associated with rock.

As a solution to these problems, researchers at the Spokane Research Center, Spokane, WA, have developed a finite-element pre- and postprocessing procedure that takes advantage of the automatic mesh-generating capabilities of ANSYS, a commercially available, finite-element analysis program, while at the same time utilizing the analytical capabilities of the UTAH2PC finite-element code. With this procedure, a user can model two-dimensional sections of underground hard-rock mines, even those with multiple formations and anisotropic materials. These models can be used to plan mining sequences and layouts that will avoid the creation of regions of unacceptably high displacements or stress concentrations.

REQUIREMENTS FOR SOFTWARE

1. Computer-aided design (CAD) software capable of saving drawing files in IGES format. Third-party programs are available that will convert other drawing formats (such as DXF) into an ANSYS format.

2. ANSYS version 5.0 or later (pre- and postprocessor only). This program is used to create the finite-element mesh and view the results of the analysis.

3. ANS2UT2.EXE. This program is used to convert ANSYS output files into UTAH2PC input files and create a UTAH2PC runstream file.

4. UTAH2PC.EXE.[1] This program is used to perform the finite-element analysis.

5. CONVERT.EXE.[2] This program is used to convert UTAH2PC results files into ANSYS input files.

REQUIREMENTS FOR ANALYSIS

1. Vertical section map of the area of the mine to be analyzed. This map should illustrate the major geologic features of that section.

2. The location of all existing and proposed mine openings.

3. Material property data along each axis of anisotropy for each of the important geologic features of the section map. These data should include–

   A. Elastic modulus.
   B. Shear modulus.
   C. Poisson's ratio.
   D. Compressive strength.
   E. Tensile strength.
   F. Shear strength.
   G. Angle of inclination of the axis of anisotropy with the x-axis.
   H. Specific gravity, if gravity loading is to be applied to the model.

4. In situ stress. This stress can be caused solely by gravitational loading or it can be input as a formula where vertical, horizontal, and shear stresses are functions of depth.

5. Measurements of displacement of the rock mass as mining progresses. This information is required to calibrate the finite-element model.

SYMBOLS

σ_v = Vertical stress.

σ_h = Horizontal stress.

X = Depth in feet.

EXAMPLE PROBLEM

An example analysis has been provided to illustrate the technique. Figure 1 shows the geometry of the geology and proposed mine openings used in this example. The mine consists of three major formations. In situ stress is given by the following formula.

$$\sigma_v = 0.1X + 1000$$

and

$$\sigma_h = 0.01X + 1000.$$

The model used to illustrate this technique contained 11,955 nodes and 11,952 elements and took approximately 6 hours to create and analyze on a 66 MHZ Pentium PC with 64 megabytes of RAM. Text appearing inside square brackets [ ] is a command issued to ANSYS.

PROCEDURE

Step 1: Create Data for Analysis

1-1. Begin by digitizing mine drawings using a CAD program. Either horizontal or vertical sections may be digitized and modeled. If vertical sections are to be analyzed, then the drawing should be digitized so that mine depth increases along the positive x-axis, as shown in figure 1. This is necessary to accommodate UTAH2PC's coordinate system, which defines positive x as pointing down. The drawing should include only major geologic formations. Note that in the example, an extra line segment was added to break formation 1 into two sections. This was done to isolate the area of formation 1 in which mining is to occur. The purpose for this isolation will be explained later. Avoid overlapping and concurrent line segments (figure 2). ANSYS will not generate a finite-element mesh if these conditions are present.

Step 2: Create Finite-Element Mesh

2-1. Export the final drawing in an IGES format. If the CAD program does not have IGES export capabilities, then contact your ANSYS distributor for other available translators.

---

[1]UTAH2PC was written by William Pariseau of the University of Utah under contract with the U.S. Bureau of Mines and is available at no cost from the Spokane Research Center, NIOSH, Spokane, WA 99207.

[2]ANS2UT.EXE and CONVERT.EXE were both written by researchers at the Spokane Research Center of the U.S. Bureau of Mines and are available at no cost from the Spokane Research Center, NIOSH, Spokane, WA 99207.

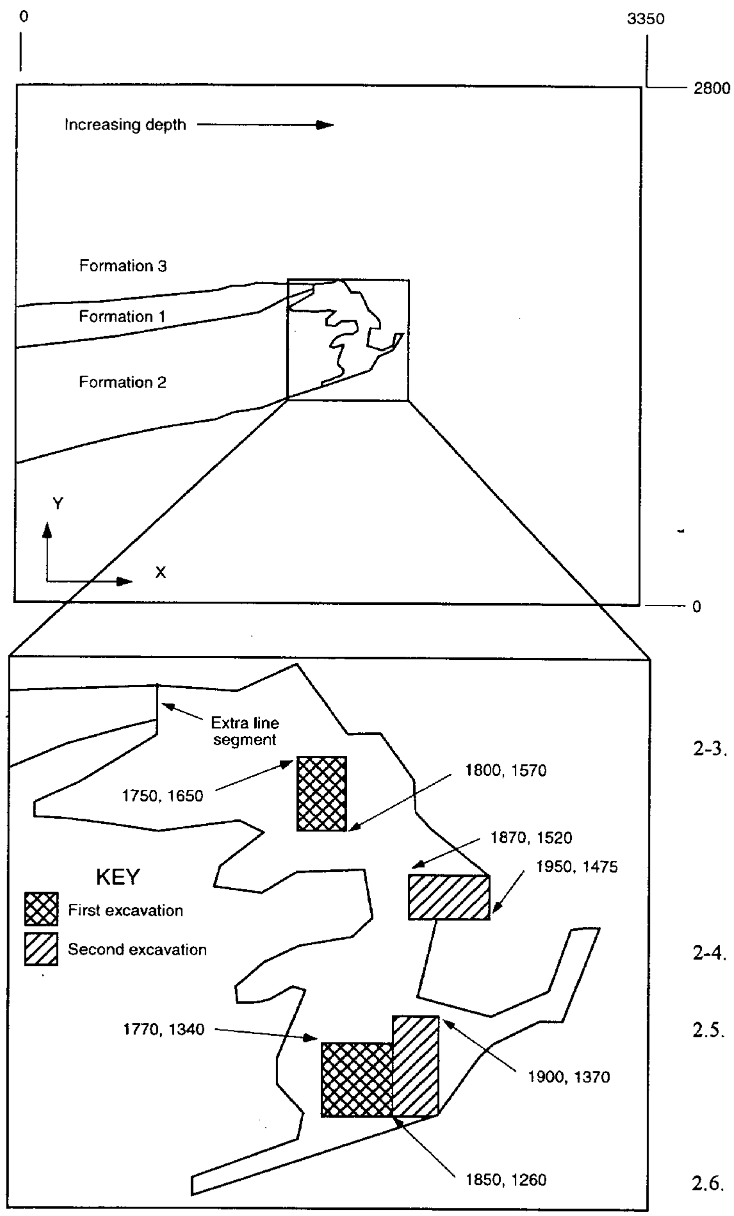

Figure 1.—Mine plan

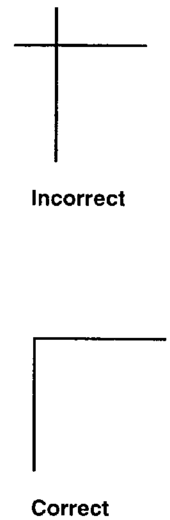

Figure 2.—Line configuration

2-2. Run ANSYS from the directory that contains the IGES format file created in step 1.

2-3. Input the IGES format file. This step will read in all of the line segments created by the digitizing process in step 1.
[/aux15]
[iges,*fname,ext*] (*filename,ext* is the name and extension of the IGES file created in step 2-1.)
[finish]

2-4. Enter the ANSYS preprocessor.
[/prep7]

2.5. Define element types to be used. This step will tell ANSYS to use three- and four-node, two-dimensional elements when creating the finite-element mesh.
[et,1,plane42]

2.6. View line segments. The plot should look identical to the digitized mine elevation drawing created in step 1-1.
[lplot]

2.7. Merge coincident keypoints. Keypoints are the points that define the ends of each line segment. By issuing the following command, any keypoints which are within a radius of 1 unit from each other will be merged together into one keypoint. This step will eliminate extraneous keypoints that might have been created during the digitizing process.
[nummerge,kp,1]

2-8. Create areas using the lines created in the previous steps. This is done by selecting the lines that bound a region of the model. In this example, formation 1 was divided into two areas: area 1, where mining is to occur, and area 2, which is located in the same formation as area 1, but which will not be mined. Formation 2 was defined as area 3, and formation 3 was defined as area 4 (figure 3).
[al,p]

2-9. Mesh the model.

A. Define element size. There are several factors to consider when setting the element size. The smaller the element size, the greater the number of elements generated for a given area. The more elements, the more accurate the results, but the greater the demand on computer resources. Another consideration when setting element size is that stresses will concentrate in the region of mine openings and diminish away from mine openings. Therefore, the elements near mine openings should be of a relatively small size (for greater accuracy), while elements away from mine openings can be kept large (to reduce demand on computer resources). It is up to operator experience to determine exactly how to size elements throughout the model. Note that an analysis can be run with a relatively course mesh (larger element size) and then rerun with a finer mesh (smaller element size). If there is no appreciable change in the results, then there is likely no need to further refine the element size. However, if the results do change appreciably, then the model should be recreated with smaller elements and the analysis repeated. In the example, mining will take place in area 1. Therefore, this is the area in which stresses will concentrate, and smaller elements will be required. In addition, smaller elements will be used to better define the mine openings. In the example, elements were given an edge length of 5 feet.
[esize,5]

B. Assign proper material numbers to the areas where a finite-element mesh is to be created. At this point, only area 1 is to be meshed, and area 1 is in formation 1, so each element generated should be assigned material number 1. The material properties associated with material number 1, as well as all other materials used in the model, will be listed in a file that will be used later in the analysis (see step 4-3).
[mat,1]

C. Select and create a mesh for area 1. During this phase, ANSYS will create a finite- element mesh for the area selected.
[amesh,p]

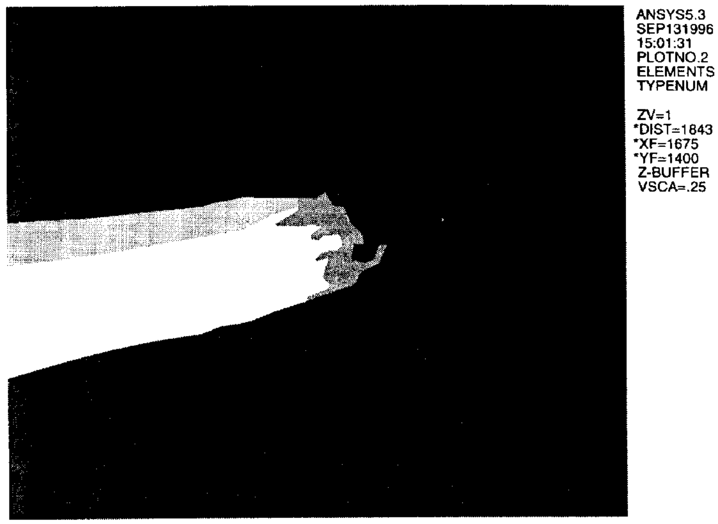

Figure 3.—Line plot

D. Create a finite-element mesh for each of the outer areas. These areas do not require the small element size that was used for area 1. Therefore, a different meshing strategy is used. In addition to specifying the element size, as was done in step 2-9-A, the user can also force ANSYS to create a mesh with a set number of elements along each line segment. This technique allows the user to vary the size of elements throughout the model. There is no rule to specify how many elements should be placed along each line segment. The operator must use judgment and experience to make this determination. Again, because stresses will concentrate around mine openings, smaller elements should be used around the area of mining, and larger elements may be used away from the area of mining. In the example, ANSYS was told to place 20 elements along the outsides of the model. Ten-foot elements were placed along the lines closer to area 1 (figure 4).
[lesize,p]

E. Specify the number of element faces to be placed along each line division as 1.
[esize,1]

F. Complete the mesh of areas in formation 1 by generating a mesh of area 2.
[amesh,p]

G. Set the material number to 2 (for formation 2) and generate a mesh of area 3.
[mat,2]
[amesh,p]

H. Set the material number to 3 (for formation 3) and generate a mesh of area 4.
[mat,3]
[amesh,p]

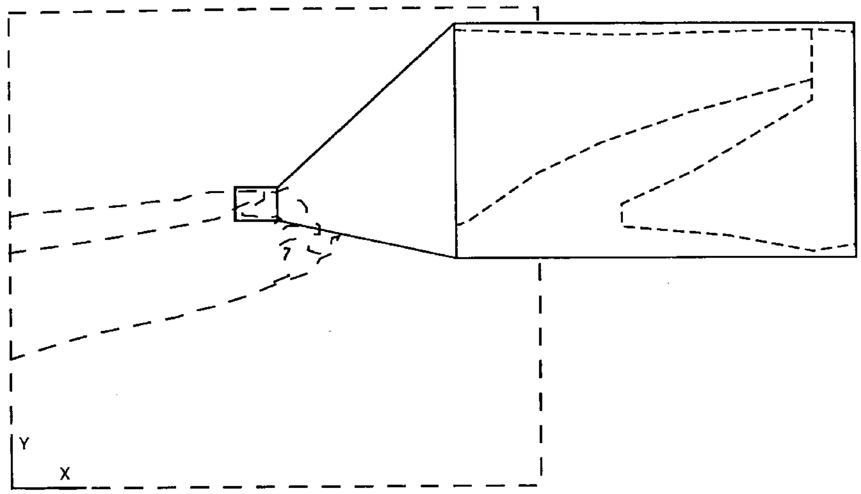

Figure 4.—Area plot

I. View all elements.
[eplot]

J. If the element sizes were not properly determined, then ANSYS will warn that poorly shaped elements were created. UTAH2PC will not run an analysis if the model contains poorly shaped elements. This problem may occur if the element size changes too rapidly near the area of mining. To correct this problem, the operator must first delete the elements generated in the areas outside the mining regions. Second, the operator must force ANSYS to slow the rate at which the elements size increases as the elements radiate away from the area of mining. Finally, the areas outside the mining regions must be re-meshed.
[aclear,p] Deletes elements from selected areas.
[mopt,trans,1] Slows rate of element growth.
[amesh,p] Re-meshes areas.

Step 3: Output Data

3-1. Copy a list of the elements to a file named ELIST.DAT.
[ewrite,elist,dat,,0]

3-2. Copy a list of nodes to a file named NLIST.DAT.
[nwrite,nlist,dat,,0]

3-3. Select the nodes to be constrained only in the x-direction. In this example, these are the nodes lying along line x = 3350 (the "bottom" of the model). Do not include the two nodes at the corners of the model. These nodes are constrained in the x- and y-directions. Copy a list of these nodes to a file named UX.DAT.
[nsel,s,p]
[nwrite,ux,dat,,0]

3-4. Select the nodes to be constrained only in the y-direction. In this example, these are the nodes lying along lines y = 0 and y = 2800. Again, do not include the two nodes at the bottom corners of the model. Copy a list of these nodes to a file named UY.DAT.
[nsel,s,p]
[nwrite,uy,dat,,0]

3-5. Select the nodes that are to be constrained only in the x- and y-directions. In this example, these are the two nodes at (3350,0) and (3350,2800). Copy a list of these nodes to a file named UXUY.DAT.
[nsel,s,p]
[nwrite,uxuy,dat,,0]

3-6. Select the elements to be cut during the first excavation. In the example, the cut elements are contained within two rectangles, the first with coordinates (1850,1260), (1900,1370), and the second with coordinates (1950,1775), (1870,1520) (figure 1). There are several steps involved in selecting these cut elements.

A. Select nodes within the first set of coordinates. First, select all nodes between x = 1850 and x = 1900. Then, from these nodes, reselect the nodes lying between y = 1260 and y = 1370.
[nsel,s,loc,x,1850,1900]
[nsel,r,loc,y,1260,1370]

B. Select all elements in contact with these nodes.
[esln,s,0]

C. Group these elements into a component named "ekill1a."
[cm,ekill1a,elem]

D. Select the nodes within the second set of coordinates. First, select the nodes between x = 1950 and x = 1870.

Then, from these nodes, reselect the nodes lying between y = 1475 and y = 1520.
[nsel,s,loc,x,1950,1870]
[nsel,r,loc,y,1475,1520]

E. Select all elements in contact with these nodes.
[esln,s,0]

F. Group these elements into a component named "ekill1b."
[cm,ekill1b,elem]

G. Group the two components into an assembly called "ekill1."
[cmgrp,ekill1,ekill1a,ekill1b]

H. Select all nodes and elements in the model.
[allsel]

I. Select the elements stored in the assembly "ekill1." These are the elements to be excavated in the second step of the analysis.
[cmsel,s,ekill1]

J. Copy a list of these elements to a file named EKILL1.DAT.
[ewrite,ekill1,dat,,0]

3-7. Repeat this process for the elements to be cut for the third step of the analysis, saving the element listing as EKILL2.DAT.

3-8. Save ANSYS database and exit the program.
[save]
[finish]
[/exit]

Step 4: Prepare Data for Analysis

Run ANS2UT2 from the directory that contains the files created in step 3. This program will convert the ANSYS output files into UTAH2PC input files. The program also creates the master runstream file for UTAH2PC. Consult the UTAH2PC documentation for more information on UTAH2PC files. ANS2UT2 will request the following information:

4-1. Job title. This is an arbitrary title for the UTAH2 runstream file and can be up to 80 characters long.

4-2. Master runstream file name. This is the user-supplied name of the master runstream file to be created by ANS2UT2 and provides the input for the UTAH2PC analysis. Three master runstream files were created for the example, one for each of the three load steps. The first (L1.DAT) contains all the data necessary for UTAH2PC to calculate the state of stress of the model prior to any excavation. The second (L2.DAT) contains all the data necessary for UTAH2PC to calculate the state of stress of the model after the first excavation. The third (L3.DAT) contains all the data necessary for UTAH2PC to calculate the state of stress of the model after the second excavation.

4-3. Material properties file name. This file is provided by the user and contains material properties for each of the materials used in the analysis.

4-4. Number of different material types. The user is to define the number of different material types used in the analysis. Note that each of the different material types should be defined in the material properties file. Three material types were used in the example analysis, one for each of the three geologic formations.

4-5. ANSYS element connection filename. This is the element listing generated by ANSYS (see step 3-2 above). The default is ELIST.DAT. ANS2UT2 will convert this file into a UTAH2PC formatted file named ELIST.UT2.

4-6. ANSYS node location filename. This is the node listing generated by ANSYS (see step 3-1 above). The default is NLIST.DAT. ANS2UT2 will convert this file into a UTAH2PC formatted file named NLIST.UT2

4-7. ANSYS cut or fill element listing. This is the list of elements to be cut or filled (see step 3-6-A above). In this example, the first load step has no cut or fill element listing. The elements cut in the second load step are listed in the file EKILL1.DAT. The elements cut in the third load step are listed in the file EKILL2.DAT. ANS2UT2 will convert these files into UTAH2PC formatted files. The name of the UTAH2PC format file is supplied by the user. ANS2UT2 will also prompt the user for the name of a file that UTAH2PC will create and store the number of each node that is attached to the cut elements.

4-8. Gravity loading. Select this option if the first load step is used to calculate loading by gravity. If gravity loading is selected, then the material properties file must include the specific gravity for each of the materials listed.

4-9. Initial stress. As an alternative to gravity loading, initial stress can be provided either from a file (such as the results from a previous load step) or calculated from known stress gradients. If the stress gradient is known, then ANS2UT2 will prompt the user for the following information.

A. Name of the initial stress file to be created.

B. Depth to top of model.

C. Slope and constant defining stress gradients. Stress is defined by the formula–

$$y = mx + b,$$

where y = stress gradient,
m = slope of gradient,

x = depth in feet,

and b = constant.

In the example, the initial stress for load step 1 was defined by the following formulae.

$$\sigma_v = 0.1X + 1000$$

and $$\sigma_h = 0.01X + 1000.$$

These formulas were provided to ANS2UT2, which applied these values to all of the elements in the model and saved the results in the file STRESS.UT2.

4-10. Boundary node listing. The boundary node listing is generated by ANSYS (see steps 3-3, 3-4, and 3-5 above). The default is UX.DAT, UY.DAT, UXUY.DAT. ANS2UT2 will convert these three files into a single UTAH2PC formatted file named BOUND.UT2.

4-11. Screen output filename. This is a file that will contain the UTAH2PC runstream output. This file name does not require an extension. The first four characters of this file name will be used as the first four characters for UTAH2PC stress and displacement output files.

4-12. Number of load steps. This is the number of increments to reach final load. Usually 10 increments are sufficient.

4-13. Maximum number of iterations per load. This is the number of iterations allowed per load step. A value of 80 was used in the example analyses.

4-14. Iteration interval. This is the number of iterations between printing of residuals and should be less than the maximum number of iterations. A value of 20 was used in the example analysis.

4-15. Type of analysis. Analyses include plane stress, plane strain, axially symmetric, elastic, or elastic-plastic. The type of analysis used in the example problem was an elastic-plastic analysis with a quadratic yield condition.

4-16. Caving. An elastic-brittle analysis can be pursued by selecting the caving analysis. This program option is somewhat experimental and should be used with care.

4-17. Batch runstream. Some data files can be skipped during a sequential batch run consisting of several back-to-back program passes. Select this option if this action is desired.

4-18. UTAH2PC to write output files. Select this option if UTAH2PC is to save output files (recommended).

4-19. Input units. If metric units are used in the model, then "yes" should be selected. Otherwise, the default is "no."

4-20. Error value. The program will exit the equation solver when the residuals are reduced to a value less than the error value. A value of 1 is recommended.

4-21. Orf value. An over-relaxation factor is used to accelerate the equation-solving procedure. Orf should be between 1 and 2 (1.86 recommended).

4-22. Xfac and Yfac. These are the scale factors used to convert the measurement units in the mine plans into inches or millimeters, as appropriate.

4-23. Efac and Sfac. These are scale factors used to change elastic properties (efac) or strength properties (sfac).

Load step 1 consists of applying the initial stress field to the model. During load step 2, the resultant stresses from load step 1 are applied to the model, and the first excavation takes place. During load step 3, the resultant stresses from load step 2 are applied to the model, and the second excavation takes place.

Step 5: Analyze Data

Run UTAH2PC from the directory containing the files created in step 4. UTAH2PC will prompt the user for the name of a runstream file (as created in step 4-2) and will then perform the analysis. Output includes cartesian stresses and element safety factors, element principal stresses and strains, and node force and displacements. For the example analysis, UTAH2PC was executed three times, once for each of the three load steps.

Step 6: Prepare Results of Analysis

Run CONVERT from the directory that contains the files created in step 5. This program converts Cartesian stresses, element safety factors, and node displacement files generated by UTAH2PC into a format ANSYS can read. CONVERT will prompt the user for the name of the UTAH2PC output file to be converted and the name of the ANSYS input file to be created.

Step 7: Review Results of Analysis

7-1. Run ANSYS from the directory that contains the ANSYS files created in step 6.

7-2. Resume the original model database.
[resume,*filename,db*] *filename,db* is the file name and extension of the ANSYS database. The default is file.db.

7-3. Enter the ANSYS postprocessor.
[/post1]

7-4. Create element tables that are to contain the stresses and safety factors generated by UTAH2PC in step 5 and converted into ANSYS format in step 6.
[etable,Sxx]
[etable,Syy]
[etable,Szz]
[etable,Txy]
[etable,Fac]

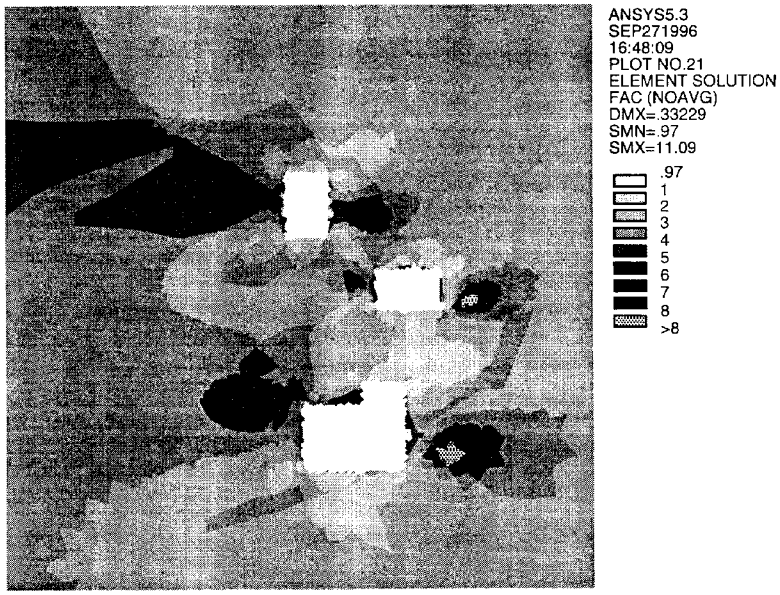

Figure 5.—Safety factors around openings

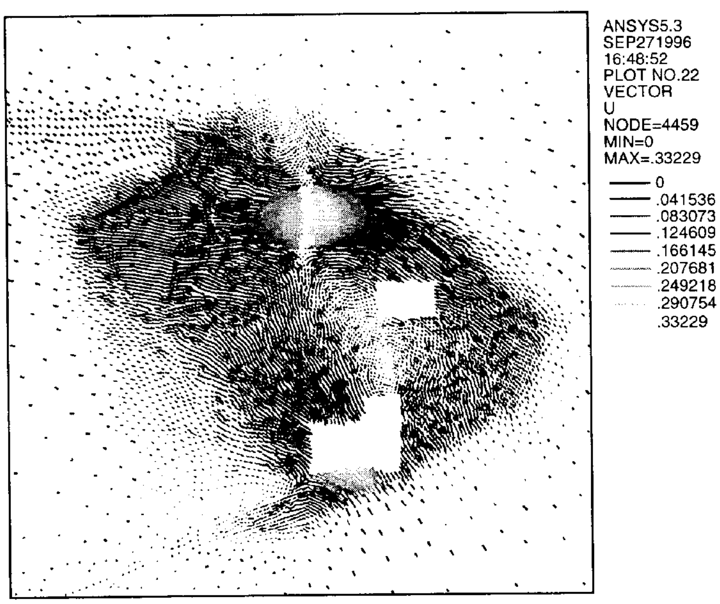

Figure 6.—Displacements around mine openings

7-5. Enter the converted stress and displacement files from load step 1 into the element tables.
[/input,*filename,ext*]

7-6. View analysis results. The following commands will cause ANSYS to plot the results of the analysis.
[pletab,sxx] Generate a plot of vertical stresses.
[pletab,syy] Generate a plot of horizontal stresses.
[pletab,txy] Generate a plot of shear stresses.
[pletab,fac] Generate a plot of safety factors.
[plvect,u,,,,vect] Generate a plot of displacements.

7-7. Repeat steps 7-5 and 7-6 to review the results from load steps 2 and 3. Figures 5 and 6 illustrate the resultant safety factors and displacements around the mine opening.

SUMMARY AND CONCLUSIONS

The technique explained above is one method that will allow mine planners to generate a two-dimensional, finite-element model of an underground section of a mine in a relatively straightforward manner. This model can be analyzed, and resultant stresses, displacements, and safety factors around mine openings can be viewed. This information allows a mine planner to develop mine plans that reduce the potential for rock failure around mine openings.

Cross sections consisting of several formations of anisotropic materials can be modeled, but many variables can affect the accuracy of the model, including the number and location of joints and fractures, the presence of water, the accuracy of measurements of material properties and in situ stress, and the actual geometry of the various formations and openings. It is therefore critical that a mine planner validate the results of the analysis by comparing the predicted response of the mine with the actual response of the mine. The elastic material properties may be validated by comparing predicted displacements with actual displacements. Strength properties may be validated by comparing predicted yield zones with observed yield zones *(2)*. These comparisons will allow a planner to determine if the material properties used in the model should be modified in order to create a more accurate model.

REFERENCES

1. Pariseau, W. G., J. C. Johnson, M. M. McDonald, and M. E. Poad. Rock Mechanics Study of Shaft Stability and Pillar Mining, Homestake Mine, Lead, SD. (In Three Parts). U.S. Bur. Mines RI 9531, 1995, 20 pp.

2. Pariseau, W. G. Research Study on Pillar Design for Vertical Crater Retreat (VCR) Mining (contract JO215043, Univ. UT). U.S. Bur. Mines OFR 44-86, 1986, 232 pp.; NTIS PB 86-210960.