NUREG/CR-5429
Vol. 2

UDEC (Universal Distinct Element Code) Version ICG1.5

User's Manual

Prepared by Mark Board

Itasca Consulting Group, Inc.

Prepared for
U.S. Nuclear Regulatory Commission

AVAILABILITY NOTICE

Availability of Reference Materials Cited in NRC Publications

Most documents cited in NRC publications will be available from one of the following sources:

1. The NRC Public Document Room, 2120 L Street, NW, Lower Level, Washington, DC 20555
2. The Superintendent of Documents, U.S. Government Printing Office, P.O. Box 37082, Washington, DC 20013-7082
3. The National Technical Information Service, Springfield, VA 22161

Although the listing that follows represents the majority of documents cited in NRC publications, it is not intended to be exhaustive.

Referenced documents available for inspection and copying for a fee from the NRC Public Document Room include NRC correspondence and internal NRC memoranda; NRC Office of Inspection and Enforcement bulletins, circulars, information notices, inspection and investigation notices; Licensee Event Reports; vendor reports and correspondence; Commission papers; and applicant and licensee documents and correspondence.

The following documents in the NUREG series are available for purchase from the GPO Sales Program: formal NRC staff and contractor reports, NRC-sponsored conference proceedings, and NRC booklets and brochures. Also available are Regulatory Guides, NRC regulations in the *Code of Federal Regulations*, and *Nuclear Regulatory Commission Issuances*.

Documents available from the National Technical Information Service include NUREG series reports and technical reports prepared by other federal agencies and reports prepared by the Atomic Energy Commission, forerunner agency to the Nuclear Regulatory Commission.

Documents available from public and special technical libraries include all open literature items, such as books, journal and periodical articles, and transactions. *Federal Register* notices, federal and state legislation, and congressional reports can usually be obtained from these libraries.

Documents such as theses, dissertations, foreign reports and translations, and non-NRC conference proceedings are available for purchase from the organization sponsoring the publication cited.

Single copies of NRC draft reports are available free, to the extent of supply, upon written request to the Office of Information Resources Management, Distribution Section, U.S. Nuclear Regulatory Commission, Washington, DC 20555.

Copies of industry codes and standards used in a substantive manner in the NRC regulatory process are maintained at the NRC Library, 7920 Norfolk Avenue, Bethesda, Maryland, and are available there for reference use by the public. Codes and standards are usually copyrighted and may be purchased from the originating organization or, if they are American National Standards, from the American National Standards Institute, 1430 Broadway, New York, NY 10018.

NUREG/CR-5429
Vol. 2

UDEC (Universal Distinct Element Code) Version ICG1.5

User's Manual

Manuscript Completed: August 1989
Date Published: September 1989

Prepared by
Mark Board

Itasca Consulting Group, Inc.
1313 5th Street SE, Suite 210
Minneapolis, MN 55414

Prepared for
Division of High-Level Waste Management
Office of Nuclear Material Safety and Safeguards
U.S. Nuclear Regulatory Commission
Washington, DC 20555
NRC FIN D1016

ITASCA CONSULTING GROUP, INC.

TERMS AND CONDITIONS FOR LICENSING UDEC

YOU SHOULD READ CAREFULLY THE FOLLOWING TERMS AND CONDITIONS BEFORE USING THE UDEC PROGRAM. INSERTION OF THE UDEC DISK INTO YOUR COMPUTER OR READING THE SOURCE CODE INDICATES YOUR ACCEPTANCE OF THESE TERMS AND CONDITIONS. IF YOU DO NOT AGREE WITH THEM, YOU SHOULD PROMPTLY RETURN THE PACKAGE, AND YOUR MONEY WILL BE REFUNDED.

This program is provided by Itasca Consulting Group, Inc. Title to the media on which the program is recorded and to the documentation in support thereof is transferred to the customer, but title to the program is retained by Itasca. You assume responsibility for the selection of the program to achieve your intended results and for the installation, use and results obtained from the program.

LICENSE — Source Code

- You may use the program at only one site on only one machine at any one time.
- You may copy the program for back-up only in support of such use.
- You may not copy or transfer the program, or any copy, in whole or part, except as expressly provided in this document.
- The user is fully responsible for any modifications made to the code.
- You may not sublicense, rent, or lease this program.

LICENSE — Executable Code

- You may use the program on only one machine at any one time.
- You may copy the program for back-up only in support of such use.
- You may not copy, modify, or transfer the program, or any copy, in whole or part, except as expressly provided in this document.
- You may not sublicense, rent, or lease this program.

WARRANTY

Itasca will correct any errors in the code at no charge for twelve (12) months after the purchase date of the code. Notification of a suspected error must be made in writing, with a complete listing of the input and output files and description of the error. If, in the judgment of Itasca, the code does contain an error, Itasca will (at its option) correct or replace the copy at no cost to the user or refund the initial purchase price of the code.

LIMITATION OF LIABILITY

Itasca assumes no liability whatsoever with respect to any use of UDEC or any portion thereof or with respect to any damages or losses which may result from such use, including (without limitation) loss of time, money or goodwill which may arise from the use of UDEC (including any modifications or updates that may follow). In no event shall Itasca be responsible for any indirect, special, incidental or consequential damages arising from use of UDEC.

ABSTRACT

UDEC (Universal Distinct Element Code) is a two-dimensional distinct element program written for the static and dynamic analysis of the mechanical, thermal and hydrologic behavior of jointed rock masses. This program has been applied to a wide variety of problems in civil construction, mining, nuclear waste disposal, and geologic modeling. This document presents the theoretical basis for the mathematical models, the details of solution procedures, user's manual and presentation of verification and example problems. A description of the program support and documentation methodology which is employed is also given. This document is given in three volumes: Volume 1 — Description of Mathematical Models and Numerical Methods, Volume 2 — User's Manual, and Volume 3 — Verification and Example Problems. These three volumes are intended to satisfy the requirements and guidelines set forth in Final Technical Position and Documentation of Computer Codes For High-Level Waste Management (NUREG-0856).

UDEC

UNIVERSAL DISTINCT ELEMENT CODE
(Version ICG1.5)

TABLE OF CONTENTS

TABLE OF CONTENTS
(Continued)

TABLE OF CONTENTS
(continued)

TABLE OF CONTENTS
(continued)

TABLE OF CONTENTS
(continued)

TABLE OF CONTENTS
(continued)

TABLE OF CONTENTS
(continued)

TABLE OF CONTENTS
(continued)

LIST OF FIGURES

LIST OF FIGURES
(continued)

LIST OF FIGURES
(continued)

LIST OF FIGURES
(continued)

LIST OF FIGURES
(continued)

PAGE

Appendix A (continued)

LIST OF FIGURES
(continued)

LIST OF FIGURES
(continued)

LIST OF FIGURES
(continued)

LIST OF FIGURES
(continued)

LIST OF TABLES

U D E C

(Version ICG1.5)

1989

1.0 INTRODUCTION

The distinct element method is a recognized discontinuum modeling approach for simulating the behavior of jointed media subjected to quasi-static or dynamic conditions. The method has three distinguishing features which make it well suited for discontinuum modeling:

- The medium is simulated as an assemblage of blocks which interact through corner and edge contacts.
- Discontinuities are regarded as boundary interactions between these blocks; discontinuity behavior is prescribed for these interactions.
- The method utilizes an explicit timestepping (dynamic) algorithm which allows large displacements and rotations and general non-linear constitutive behavior for both the matrix and discontinuities.

Since the time the method was proposed, several forms of distinct element codes have been developed to cover a range of rock mass strengths and confining pressures which are encountered in situ. UDEC is a two-dimensional version of the method which is specifically designed to simulate the predominant features of fractured rock masses, including

- variable rock deformability
- complex joint structures
- non-linear, inelastic joint behavior
- plastic behavior of intact rock
- fluid flow in joints
- far-field static or dynamic boundary conditions

Specific features of UDEC are as follow:

- automatic tunnel generator
- structural element logic to simulate reinforcement and tunnel lining
- gravity loading
- non-reflecting and free-field boundary conditions for dynamic analyses
- transient heat flow and thermally-induced stresses
- automatic damping and timestep determination
- programmed error messages

UDEC is written in FORTRAN77, and two forms of the executable code are available. One version is compiled for operation on an 80386-based microcomputer running DOS3.x, using SVS FORTRAN 386, the PHARLAP linker, and ICG X-AM DOS extender. The screen graphics support for this version is handled through a FORTRAN-linkable library (SCITECH plotting package).

UDEC is also available for operation on a DSI-780 coprocessor board (manufactured by Definicon Systems Inc.). This version is compiled using SVS FORTRAN V2.6. The screen graphics support is handled through a FORTRAN-linkable library (SCIGRAF modules, which are sold by Definicon Systems Inc.)

The maximum problem size for UDEC is dependent on the available computer memory. This version of the code, with 4 megabytes of RAM memory available, can handle approximately 2500 rigid blocks or 1000 fully-deformable blocks. Previous versions of the code could handle approximately 400 rigid blocks or 225 fully-deformable blocks. These limits were defined based on memory allocations for typical minicomputers (e.g., IBM-4341, VAX11-780, PRIME-750, etc.). In any case, the code size can be adjusted by increasing or decreasing the size of the main array, as described in Section 6.5. The size of the main array is given by the parameter MTOP (e.g., MTOP=750,000 for 4 megabyte RAM). The main array is the IA array.

This manual is organized in the following fashion. A simple example problem is described in Section 2, for the new user to try some of the basic features of UDEC. The formulation of the two-dimensional distinct element model is given in Section 3, along with descriptions of the specific features in UDEC. In Section 4, a detailed discussion of the input commands is given. This section is the primary source for information on control of the UDEC program. Note that a full description of input commands for problem execution is given in Section 4.5, while a full description of commands for plotting is given in Section 4.6. Section 5 describes techniques for the use of UDEC in problem solving. The guidelines for setting up and executing engineering problems are discussed in this Section. Section 6 presents a program guide of the data structure utilized in UDEC. Thermal-mechanical modeling with UDEC is discussed in Section 7, and verification tests of this logic in the code are also presented. Appendix A contains a variety of mechanical verification tests comparing UDEC results to closed-form solutions. In Appendix B, example practical applications of UDEC are given.

2.0 INSTANT GRATIFICATION — A SIMPLE TUTORIAL ON USING UDEC

This section is provided for the new user who wishes to jump in and begin experimenting with UDEC. A simple, fast-running example problem has been chosen which demonstrates some basic aspects of the UDEC code. UDEC may be executed in either the "batch" or "interactive" mode, as explained in the following sections.

2.1 A Simple Slope Stability Problem — "Batch" Execution

The following four-block slope stability problem can be executed after UDEC has been compiled, linked and loaded on your system. First, create the following input file called "DATA".

```
BLOCK     (0,0) (0,20) (20,20) (20,0)
SPLIT     (0,2) (20,8)
SPLIT     (5,3) (5,20)
SPLIT     (5,12) (20,18)
FIX       0,20  0,5
FIX       0,5   0,20
PROP      MAT=1  DENS=2000
PROP      JMAT=1  JKN=1.33E7  JKS=1.33E7  JFRIC=0.35
GRAVITY   0  -10
DAMP      AUTO
HIST      YVEL(10,20)  TYPE 1
CYCLE     100
PLOT      HISTORY 1
COPY      HIS.PLT
PLOT      BLOCKS
COPY      BLOCKS.PLT
SAVE      SLOPE.SAV
STOP
```

After running UDEC, the output will be in a file called "OUT". The function of each command* in the input file is explained as follows.

---

*See Section 4.5 for full descriptions of input commands.

BLOCK (0,0) (0,20) (20,20) (20,0)

This command creates a square block with side lengths of 20 units (in this case, meters) in the first quadrant.

SPLIT (0,2) (20,8)
SPLIT (5,3) (5,20)
SPLIT (5,12) (20,18)

These commands split existing blocks along the line with end points specified by coordinates in the parentheses.

FIX 0,20 0,5
FIX 0,5 0,20

These commands fix the current velocity (zero) of all blocks with centroids in the range $0<x<20$, $0<y<5$ and $0<x<5$, $0<y<20$. In this case, the lowermost and leftmost blocks are immobilized.

PROP MAT=1 DENS=2000
PROP JMAT=1 JKN=1.33E7 JKS=1.33E7 JFRIC=0.35

These commands assign various material properties to material number one. Note: by default, all blocks and joints are assigned material number one; thus, for this problem, we have specified the mass density of all blocks (MAT=1) to be 2000 units (kg/m^3, in this case). Note, also, that we are specifying the mass density— not the unit weight of the block material. We have also specified all joints (JMAT=1) to have normal (JKN) and shear (JKS) stiffnesses equal to 1.33e7 (here, Pa/m) and friction coefficients equal to 0.35.

As will be seen, different material properties could be assigned to various joints and intact blocks. It will also be seen that it is not necessary for the full keyword (i.e., PROPERTY, MATERIAL, DENSITY, etc.) to be typed in order to be recognized by the program.

GRAVITY 0 -10

This command specifies gravitational acceleration in the x- and y-directions (here, units are m/sec^2).

DAMP AUTO

This command specifies viscous mass damping to absorb vibrational energy. The keyword AUTO indicates that the damping parameters are determined automatically by the program.

HIST YVEL (10,20) TYPE 1

This command records the motion (y-velocity) of a specified point in the rock mass. Such records are useful for judging behavior (e.g., equilibrium, stability, instability) of the rock mass. In this problem, it was decided to monitor the y-velocity of a point at the top of the model. Following execution of this command, the program returns information about the selected monitoring point (5,20). This point was selected by the program as being the closest available point to that desired. The keywords TYPE 1 instructs the program to print the value (in this case, the y-velocity of point (5,20)) on the screen at specified intervals (default = every 10 cycles).

CYCLE 100

This command instructs the program to perform 100 complete calculation cycles. During execution, the current cycle count, the maximum out-of-balance force, and y-velocity of point (5,20) are printed on the screen every 10 cycles. Inspection of these values indicates that equilibrium has been obtained (i.e., the velocity and maximum out-of-balance force generally approach zero).

PLOT **HISTORY 1**

This command creates a plot of history [i.e., y-velocity (y-axis) versus pseudo time (x-axis)] to be sent to the file given by the **COPY** command, which must follow immediately.

COPY **HIS1.PLT**

This command puts the plot data into a file called "HIS1.PLT".

PLOT **BLOCK**

This command creates a plot of the problem geometry (Fig. 2-1) to be sent to a file given by the **COPY** command, which follows immediately. Note that, because a rounding length is not specified, the default rounding length = 0.5 units (meters) is used for the corners of all blocks. Rounding is used in UDEC to give reasonable physical behavior to blocky assemblages. The rounding length may be specified by the user at the beginning of any problem.

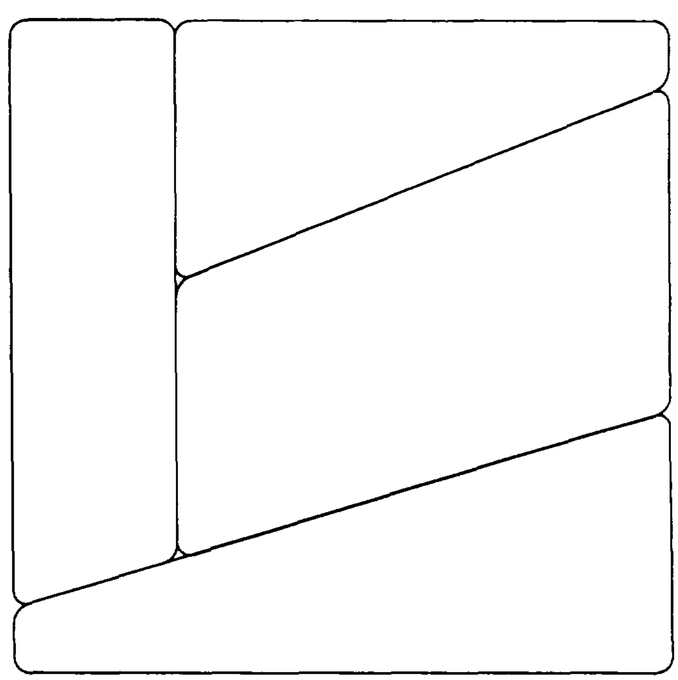


Fig. 2-1 A Simple Slope Stability Example: Equilibrium Stage (friction coefficient = 0.35)

COPY BLOCKS.PLT

The command puts the plot data into a file called "BLOCKS.PLT".

SAVE **SLOPE.SAV**

This command causes the current problem state to be saved so that it can be restarted at any time — for example, to perform parameter studies. The name of the file saved is "SLOPE.SAV".

STOP causes execution of UDEC to stop.

Next, another input file (called "DATA") can be created to study the behavior of the system following removal of the leftmost block.

```
RESTART  SLOPE.SAV
DELETE   0,5  0,20
CYCLE    1000
PLOT     BLOCK  VELOCITY
COPY     BLVEL.PLT
PLOT     HISTORY 1
COPY     HIST1.PLT
STOP
```

The function of each of these commands is as follows.

RESTART SLOPE.SAV

This command causes the initial save state to be restored.

DELETE 0,5 0,20

This command deletes blocks with centroids in the range 0<x<5, 0<y<20. In this case, the leftmost block is removed.

CYCLE 1000

This command causes an additional 1,000 calculation cycles to be performed.

PLOT BLOCK VELOCITY

This command causes the data for the plot shown in Fig. 2-2 to be created. The arrow at the centroid of the upper block gives the direction and velocity magnitude of this block. The plot shows that only the top block is sliding. This is the expected result because the friction coefficient (0.35) is less than the slope (0.4) of the joint between the two uppermost blocks.

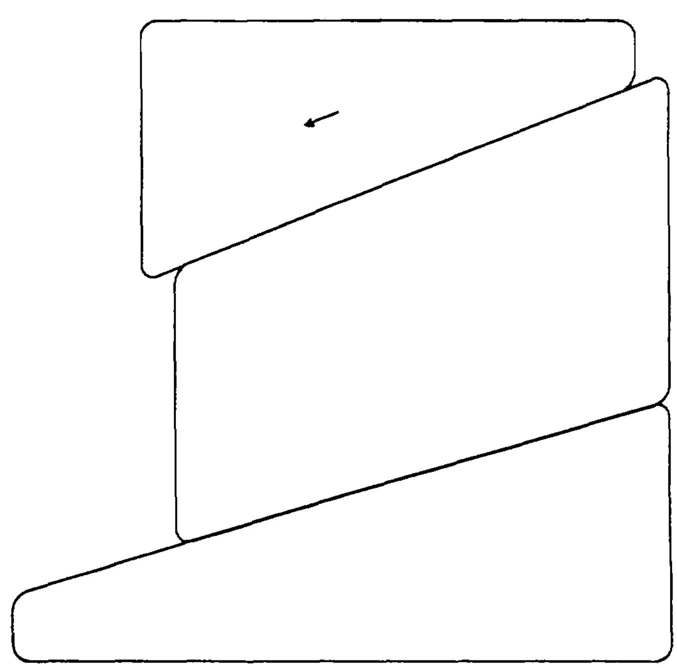


Fig. 2-2 A Simple Slope Stability Example: One Block Sliding (friction coefficient = 0.35)

COPY **BLVEL.PLT**

This command copies the plot data to a file called "BLVEL.PLT".

PLOT **HISTORY 1**

This command again creates a plot file of y-velocity versus pseudo-time.

COPY **HIST1.PLT**

This command copies the plot data to a file called "HIST1.PLT".

STOP causes execution of UDEC to stop.

Next, another input file called "DATA" can be created to examine the effect of other choices of problem parameters (in this case, joint friction):

```
RESTART   SLOPE.SAV
DELETE    0,5  0,20
PROPERTY  JMAT=1  JFRIC=0.2
CYCLE     1000
PLOT      BLOCK  VELOCITY
COPY      BLVEL.PLT
PLOT      HISTORY 1
COPY      HIST1.PLT
STOP
```

Here, the commands are as previously described with the exception of the third input command — i.e.,

PROPERTY JMAT=1 JFRIC=0.2

This command assigns a friction coefficient of 0.2 to joint material one (i.e., all joints). As seen in Fig. 2-3, the result of this change is that both top blocks are now sliding. (The top block is sliding faster than the middle block.) This is the expected result because the friction coefficient (0.2) is less than the joint slopes (0.4 for the top and 0.3 for the bottom).

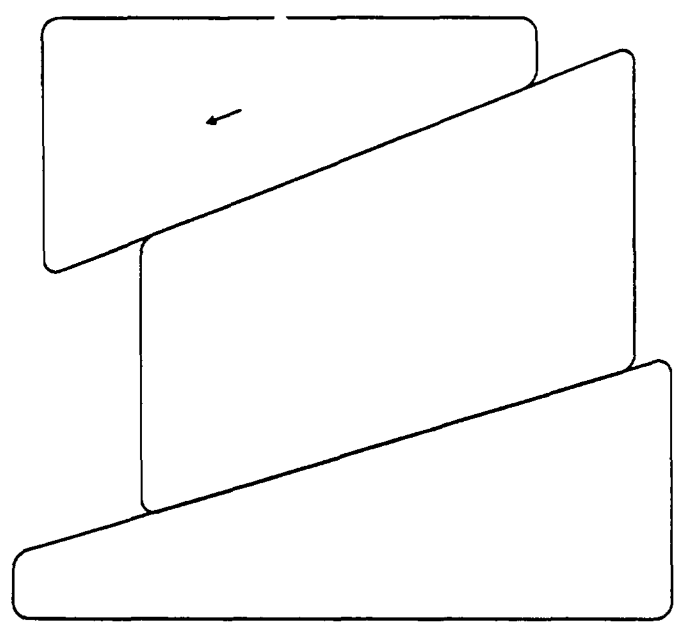


Fig. 2-3 A Simple Slope Stability Example: Two Blocks Sliding (friction coefficient = 0.2)

2.2 A Simple Slope Stability Problem — Interactive Execution

The preceding four-block slope stability problem can be executed interactively by simply running the UDEC program. [NOTE: The file DATA must not be present to run interactively. UDEC always looks for the file called "DATA". If the file called "DATA" is not found, then UDEC operates interactively.]

The problem begins by specifying a single block using the **BLOCK** command:*

UDEC> **BLOCK (0,0) (0,20) (20,20) (20,0)**

This command creates a square block with side lengths of 20 units (in this case, meters) in the first quadrant. To see the block, type

UDEC> **PLOT BLOCK**

A picture of the block will appear on the screen. Note that the corners appear slightly rounded. The rounding is used in UDEC to give reasonable physical behavior to blocks which just slightly overlap each other. The rounding length may be adjusted by the user. To continue the problem, strike the return (enter) key.

The problem is continued by splitting the initial block into smaller blocks by typing

UDEC> **SPLIT (0,2) (20,8)**
UDEC> **SPLIT (5,3) (5,20)**
UDEC> **SPLIT (5,12) (20,18)**

These commands split existing blocks along the line with endpoints specified by coordinates in parentheses. To see the resultant problem geometry, again type

UDEC> **PLOT BLOCK**

and strike the return key to continue.

---

*See command list in Section 4.5 for further details.

Next, the lowermost and leftmost blocks are immobilized by typing

UDEC> **FIX 0,20 0,5**
UDEC> **FIX 0,5 0,20**

This command fixes the current velocity (zero) of all blocks with centroids in the range 0<x<20, 0<y<5 and 0<y<20.

Then, we assign required material properties to the blocks and joints by typing

UDEC> **PROP MAT=1 DENS=2000**

UDEC> **PROP JMAT=1 JKN=1.33e7 JKS=1.33e7 JFRIC=0.35**

For this problem, we have specified the mass density of all blocks to be 2,000 units (kg/m^3, in this case). Note that we are specifying the mass density not the unit weight of the block material. We have also specified all joints to have contact normal (jkn) and shear (jks) stiffness equal to 1.33e7 (here, Pa/m) and friction coefficients equal to 0.35.

As will be seen later, different properties could be assigned to various joints and intact blocks. It will also be seen that it is not necessary for the full keyword (i.e., PROP, MAT, DENS, etc.) to be typed to be recognized by the program. Full keywords are given here for clarity.

Next, gravitational accelerations in x- and y-directions are specified by typing

UDEC> **GRAVITY 0 -10**

In order to absorb vibrational energy, viscous mass damping is introduced by typing

UDEC> **DAMP AUTO**

At this point, the problem is ready to be executed. As will be seen later, it is often helpful to judge behavior (i.e., equilibrium, stability, instability) by observing the motion of specified points in the rock mass. In this problem, it was decided to monitor the y-velocity of a point at the top of the model. The command used to record this motion is

UDEC> **HIST YVEL (10,20) TYPE 1**

Following execution of this command, the program returns information about the selected monitoring point (5,20). This point was selected by the program to be the closest available point to the desired point. The keyword TYPE 1 instructs the program to print the value [in this case, the y-velocity of point (5,20)] on the screen at specified intervals.

One hundred calculation cycles are executed by typing

UDEC> **CYCLE 100**

During execution, the current cycle count, the maximum out-of-balance force and y-velocity of point (5,20) are printed on the screen every 10 cycles. Inspection of these values indicates that equilibrium has been obtained (the velocity and out-of-balance force generally approach zero). A graphical representation of this behavior is obtained by typing

UDEC> **PLOT HIST 1**

and striking the return (enter) key to continue.

To give plots a heading, type

UDEC> **HEAD**

HEAD>
A SIMPLE SLOPE STABILITY EXAMPLE: EQUILIBRIUM STAGE (FRICTION COEF.=0.35)

Next, again type

UDEC> **PLOT BLOCK**

and strike the return (enter) key to continue.

If the plotter had been initialized previously, a hard copy of the plot can be obtained by typing

UDEC> **COPY**

Figure 2-4 should result. The legend of this figure gives

(1) the present clock time;
(2) the current cycle number;
(3) the plotting window; and
(4) the information being plotted.

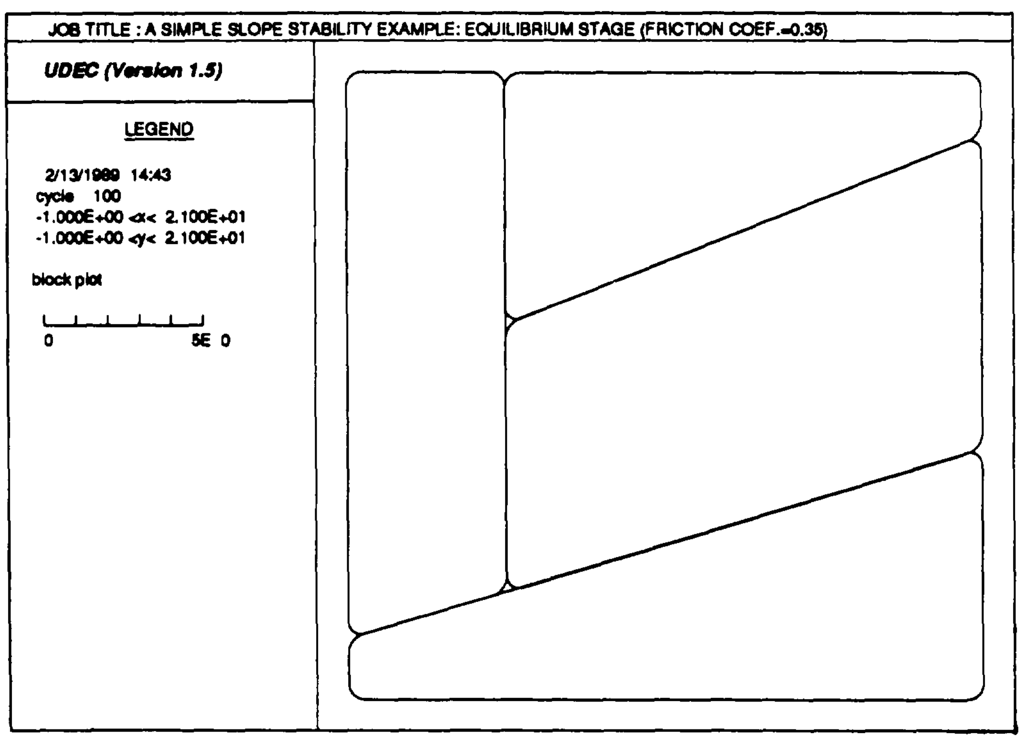


Fig. 2-4 A Simple Slope Stability Example: Equilibrium Stage (friction coefficient = 0.35)

Note that, in this case, the plotting window was automatically defined. The user may enlarge or reduce the plot using the **WINDOW** command described later.

It is often helpful to save this initial state so that it can be restarted at any time — for example, to perform parameter studies. To save the current state, type

UDEC> **SAVE SLOPE.SAV**

The name of the file saved is "SLOPE.SAV".

Next, the behavior of the slope can be studied by removing the leftmost block by typing

UDEC> **DELETE 0,5 0,20**

This command deletes blocks with centroids in the range $0<x<5$, $0<y<20$. Next, the calculation process continues using the CYCLE command. The problem state after 700 additional cycles (800 cycles total) is shown in Fig. 2-5. This figure was obtained following execution of the following commands.

UDEC> **CYCLE 700**

UDEC> **HEAD**

HEAD>
A SIMPLE SLOPE STABILITY EXAMPLE: ONE BLOCK SLIDING (FRICTION COEFF.=0.35)

UDEC> **PLOT BLOCK VELOCITY**

UDEC> **COPY**

The figure shows that only the top block is sliding. This is the expected result because the friction coefficient (0.35) is less than the slope (0.4) of the joint between the two uppermost blocks. A plot of the y-velocity history of the monitored point is obtained by typing

UDEC> **PLOT HIST 1**

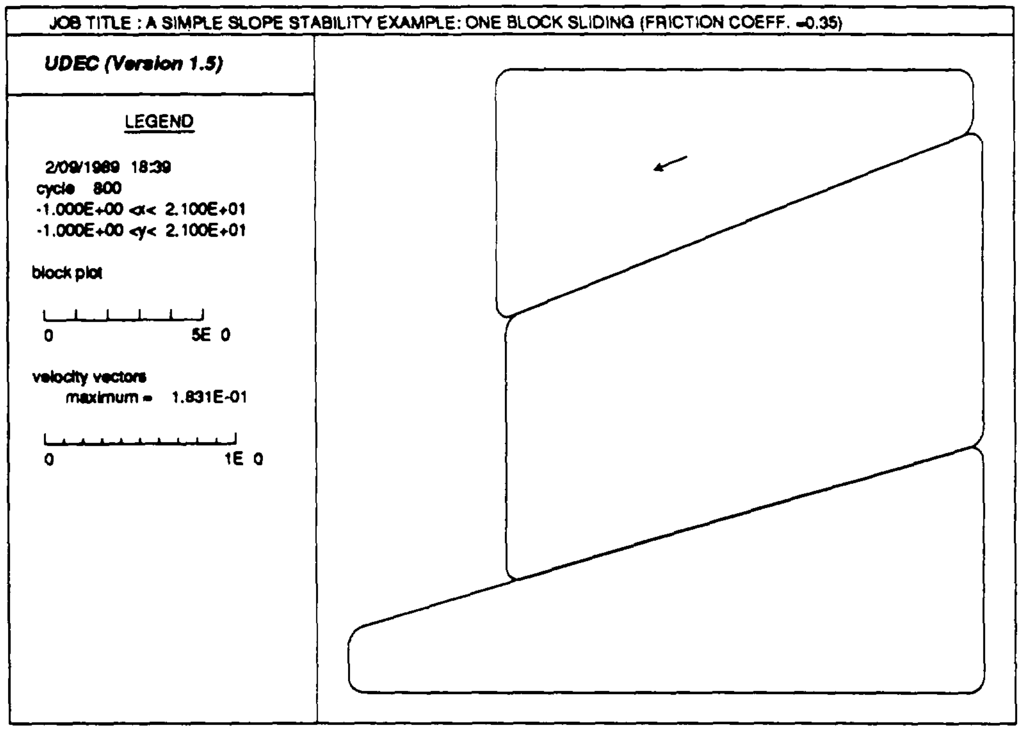


Fig. 2-5 A Simple Slope Stability Example: One Block Sliding (friction coefficient = 0.35)

The problem may be continued in the manner previously described, but it is interesting at this point to examine the effect of other choices of problem parameters. The initial save state may be recalled by typing

UDEC> **RESTART SLOPE.SAV**

The leftmost block may be removed as before but, in this case, the joint friction coefficient is reduced to 0.2. The following command sequence results in Fig. 2-6.

UDEC> **DELETE 0,5 0,20**

UDEC> **PROP JMAT=1 JFRIC=0.2**

UDEC> **CYCLE 700**

UDEC> **HEAD**

HEAD>
A SIMPLE SLOPE STABILITY EXAMPLE: TWO BLOCKS SLIDING (FRICTION COEF.=0.2)

UDEC> **PLOT BLOCK VELOCITY**

UDEC> **COPY**

As seen in Fig. 2-6, both blocks are sliding (the top block sliding faster than the middle block). This is the expected result as the friction coefficient (0.2) is less than the joint slopes (0.4, top; 0.3, bottom).

To exit UDEC, type

UDEC> **STOP**

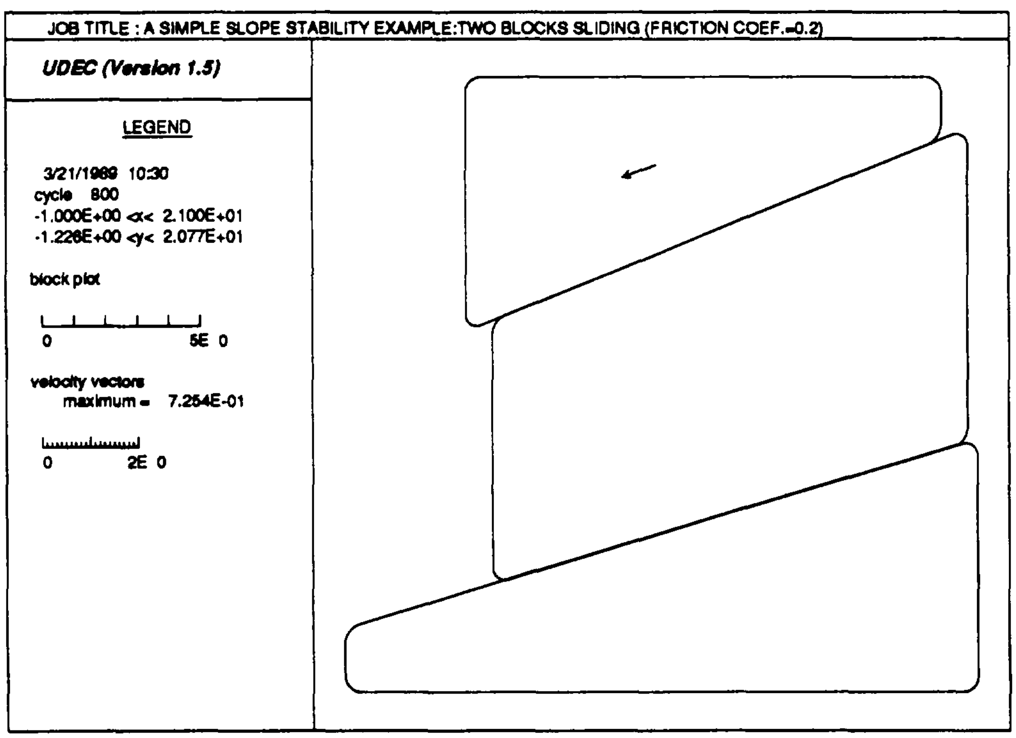


Fig. 2-6 A Simple Slope Stability Example: Two Blocks Sliding (friction coefficient = 0.2)

3.0 THEORETICAL BASIS AND FEATURES OF UDEC

The distinct element method is similar to the finite element method in that the problem domain is divided into a system of solid elements (blocks). The principal difference between the methods is that the distinct element method also permits the geometry of the elements to be defined by the spacing and orientation of the discontinuities in the rock mass, thereby allowing blocks to interact (or disconnect) from neighboring blocks. The distinct element algorithm includes not only continuum theory representation for the blocks but also force displacement laws which specify forces between blocks and a motion law which specifies motion of each block due to unbalanced forces acting on the block.

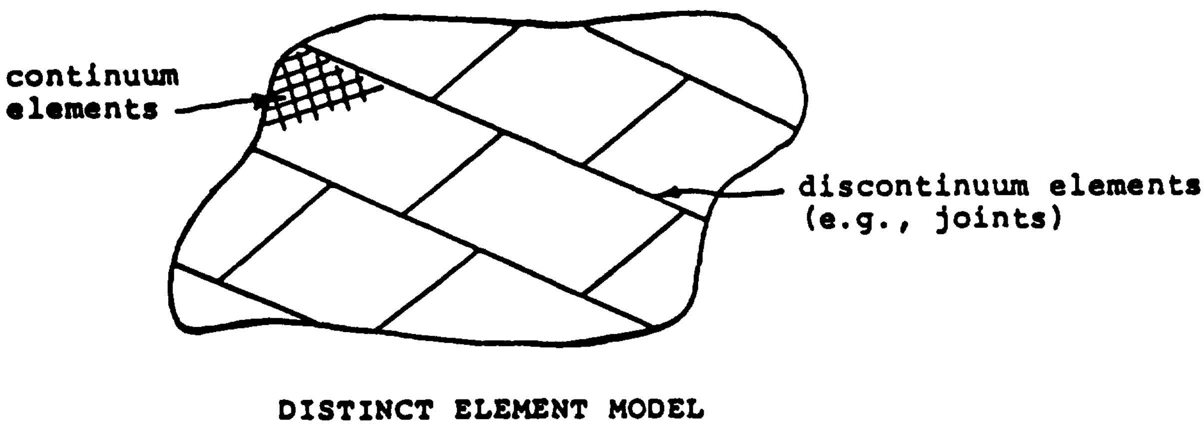


Fig. 3-1 Nature of Distinct Element Model

The Distinct Element Method was first proposed by Peter Cundall (1971). The method has undergone continuous development since that time. UDEC, which contains various features of separate special purpose codes, was first documented by Cundall (1980). Further development of UDEC is documented by Cundall and Hart (1985). Recent UDEC developments are documented by Lemos (1987). In this manual, major portions of Sections 3.3, 3.8 and 3.12 are taken, with only minor modifications, from Lemos (1987).

3.1 Explicit Solution Procedure

The distinct element method is based on an explicit solution procedure. The explicit procedure is similar to other explicit calculations in the time domain. "Explicit" refers to the nature of algebraic equations used in the numerical simulation of the physical system. In the explicit method, all quantities on one side of all equations are known, and each equation is simply evaluated to produce the result on the other side of the equation. Explicit formulations differ from implicit formulations, where unknown quantities exist on both sides of the equation; implicit formulations require the solution of simultaneous equations by some technique such as transpose elimination or Gauss elimination.

The explicit formulation relies on the fact that it takes a finite time for information to propagate through a system of blocks. The interdependence of variables over a selected time interval may be neglected if the time interval is small enough such that information passes between neighboring blocks at a speed less than physically possible. In other words, the numerical procedure is stable when the equations of motion for all blocks become uncoupled by selecting a time interval between subsequent integration intervals which is smaller than that required for adjacent blocks to communicate physically. The small timestep is the main disadvantage of the explicit method. Determination of the required timestep is based on block masses and stiffnesses present in the problem. An advantage of the explicit method is that, because matrices are never formed, large displacements and non-linear or post-elastic behavior are possible with no additional computing effort.

3.2 Block Deformability

Blocks may be rigid or deformable in the distinct element method. The basic formulation for rigid blocks is given by Cundall et al. (1978). Details of the formulation are given in Cundall (1971) and elsewhere. This formulation represents the medium as a set of distinct blocks which do not change their geometry as a result of applied loading. Consequently, the formulation is most applicable to problems in which the behavior of the system is dominated by discontinuities and where the material elastic properties may be ignored. Such conditions arise in low-stress environments and/or where the material possesses high strength and low deformability.

For many applications, the deformation of individual blocks cannot be reasonably ignored — i.e., blocks cannot be assumed to be rigid. Two approaches have been developed. In one approach, termed "simply deformable", each block is allowed three degrees of freedom to deform internally. In the second approach, termed "fully deformable", arbitrary deformation of blocks is permitted through internal discretization of blocks into finite difference zones. Both the simply-deformable and fully-deformable algorithms are used in UDEC and are described by Cundall et al. (1978). The fully-deformable logic is described as follows.

Fully-deformable blocks are internally discretized into finite difference triangles. The vertices of these triangles are gridpoints, and the equations of motion for each gridpoint are formulated as follows:

$$\ddot{u}_i = \frac{\int_s \sigma_{ij} n_j \, ds + F_i}{m} + g_i \tag{1}$$

where s is the surface enclosing the mass n lumped at the gridpoint,

n_j is the unit normal to s,

F_i is the resultant of all external forces applied to the gridpoint (from block contacts or otherwise), and

g_i is the gravitational acceleration.

During each timestep, strains and rotations are related to nodal displacements in the usual fashion:

$$\dot{\varepsilon}_{ij} = \frac{1}{2} (\dot{u}_{i,j} + \dot{u}_{j,i})$$

$$\dot{\theta}_{ij} = \frac{1}{2} (\dot{u}_{i,j} - \dot{u}_{j,i}) \tag{2}$$

Notice that, due to the incremental treatment, Eq. (2) does not imply a restriction to small strains.

The constitutive relations for deformable blocks are used in an incremental form, so that implementation on non-linear problems can be accomplished easily. The actual form of the equations is:

$$\Delta\tau^{e}_{ij} = \lambda\ \Delta\varepsilon_{v}\delta_{ij} + 2\ \mu\Delta\varepsilon_{ij} \tag{3}$$

where λ, μ are the Lame constants,

$\Delta\tau^{e}_{ij}$ are the elastic increments of the stress tensor,

$\Delta\varepsilon_{ij}$ are the incremental strains,

$\Delta\varepsilon_{v} = \Delta\varepsilon_{11} + \Delta\varepsilon_{22}$ is the increment of volumetric strain, and

δ_{ij} = Kronecker delta function.

Non-linear and post-peak strength models are readily incorporated into the code in a direct way without recourse to devices such as equivalent stiffnesses or initial strains, which need to be introduced into matrix-oriented programs to preserve linearity dictated by the matrix formulation. In an explicit program, however, the process is much simpler — after each timestep, the strain state of each zone is known. The program then needs to know the stress in each zone in order to proceed to the next timestep. The stress is uniquely defined by the stress-strain model whether it is a linearly-elastic relation or a complex, non-linear and post-peak strength model.

3.3 Interface Constitutive Relations

The deformability of the discontinuities or interfaces between blocks and the frictional characteristics are represented in the distinct element method by spring-slider systems with prescribed force-displacement relations which allow evaluation of shear and normal forces between blocks. In the model, spring-slider systems are located at contact points between blocks. The amount of penetration or overlap between two adjacent blocks can be defined directly from block geometry and block centroid translation and rotation. The force-displacement relation at one contact is thus uncoupled from that at another on the same block.

The simplest force-displacement law relates incremental normal and shear forces (ΔF_n, ΔF_s) which develop at contacts directly to the amount of incremental relative displacement (Δu_n, Δu_s):

$$\begin{aligned} \Delta F_n &= K_n \ \Delta u_n \\ \Delta F_s &= K_s \ \Delta u_s \end{aligned} \tag{4}$$

where K_n and K_s are the contact normal and shear stiffnesses [see Fig. 3-3(a)].

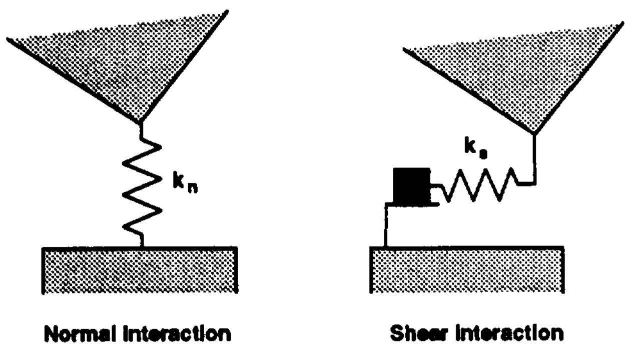


Fig. 3-2 Mechanical Representation of Interfaces in Distinct Element Method

Contact between two block edges [Fig. 3.3(b)] can be represented by two corner-edge contacts. The contact length, ℓ, calculated as

$$\begin{aligned} \sigma_n &= \frac{F_n}{\ell} \\ \sigma_s &= \frac{F_s}{\ell} \end{aligned} \tag{5}$$

and stress increments to be expressed in terms of the usual joint stiffnesses k_n and k_s [stress/length] as

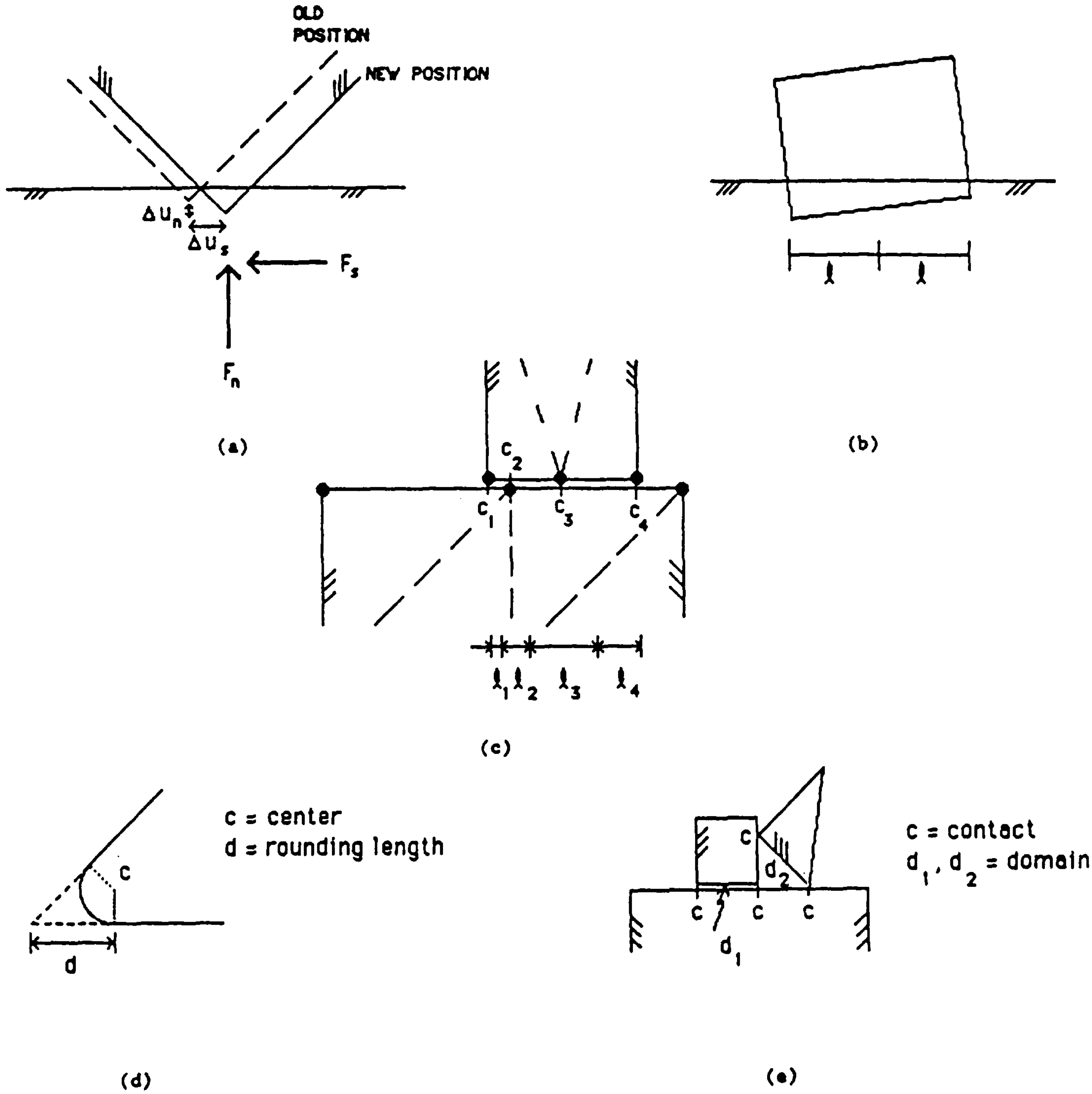


Fig. 3-3 Block Contact Geometry: (a) corner-edge contact; (b) edge-edge contact; (c) contact lengths for fully-deformable blocks; (d) rounded corner; and (e) domains

$$\sigma_n = k_n \, \Delta u_n$$
$$\Delta\sigma_s = k_s \, \Delta u_s \qquad (6)$$

When blocks are discretized into a fine internal mesh, grid-points may be placed along the original edges [Fig. 3-3(c)]. These grid-points are treated as new corners, since the edge is now able to deform into a polygonal line. The same expressions (6) are used, with contact lengths defined as shown in Fig. 3-3(c).

The overlaps in Figs. 3-3(a) and 3.3(b) represent only a mathematically convenient way of measuring relative normal displacements. In finite element or displacement discontinuity models, joints are similarly assigned a zero thickness, with overlapping indicating compressive joint stresses and separation indicating tension. If normal joint stiffnesses are increased, overlaps can be made as small as desired.

At each timestep, the incremental stresses calculated by expressions (6) are added to the existing stresses, and the constitutive criteria are checked. In the simplest model, no tensile stresses are allowed —i.e., defining tensile stresses as positive,

$$\sigma_n \leq 0 \qquad (7)$$

and the shear stresses are limited by a Mohr-Coulomb friction law:

$$|\sigma_s| \leq c - \sigma_n \tan \phi \qquad (8)$$

where c and ϕ are the joint cohesion and friction angle.

In general, the joint constitutive relations must provide the stress increments as a function of the displacement increments, current stresses and possibly other state parameters

$$\Delta\sigma_n, \; \Delta\sigma_s = f(\Delta u_n, \Delta u_s, \sigma_n, \sigma_s, \ldots) \qquad (9)$$

One such model, the continuously-yielding joint model, is described later in this section.

In principle, an interaction logic based on corner-edge contacts is sufficient, even for complex geometries. However, particular problems may arise, for instance, when the contact point approaches one of the edge endpoints. Then, the determination of which edge and corner are in contact may become ambiguous. Also, the contact normal may not be uniquely defined, or it may experience sudden jumps as the blocks rotate with respect to each other. It is important that the contact normal varies in a smooth way for any relative motion between blocks, so that the normal and shear components of the interaction force are physically meaningful. These problems are overcome in UDEC by assuming that the block corners are rounded, for the purpose of analyzing the interaction mechanics. A corner is approximated by an arc of circle tangent to the two adjacent edges [Fig. 3-3 (d)]. The distance between the tangency points and the actual corner, the rounding length d, is prescribed by the user and is the same for all corners (i.e., circle radii vary according to the corner angle). This scheme permits corner-corner contacts to be handled without ambiguity. The contact normal is defined by the line connecting the centers of the rounded corner circles. A smooth transition between corner-edge and corner-corner contacts is also achieved.

Rounded corners have the added advantage of eliminating the problem of closely-packed systems being "locked" by very small corner-corner overlaps. In a real situation, such sharp corners would probably be crushed. Rounded corners may thus provide a better approximation of the physical reality. The rounding length, if kept small (typically around a few percent of the average edge length), has no practical influence on the results of the analysis.

Because distinct element codes allow large block motion, elaborate procedures are required in order to update the contact structure, detecting new contacts and deleting others. The main difficulty is to make such updates computationally efficient, since checking all possible interactions would be impractical. UDEC takes advantage of the network of "domains" created by a two-dimensional block assembly. Domains are the regions of space between blocks which are defined by the contact points, as d_1 and d_2 in Fig. 3.3(e). During a small time increment, new contacts can only be formed between corners and edges within the same domain, so local updates can be executed efficiently whenever some prescribed measure of motion within the domain is attained. The contact updating procedure is further facilitated by a linked list data structure which follows closely the physical arrangement of corners and contacts. Details of the code organization can be found in Cundall (1980). The main disadvantage of this scheme is that it cannot be used for very loose systems, since the domain structure no longer exists.

The Continuously-Yielding Joint Model

Numerical modeling of practical problems may take the joints through rather complex load paths. Many empirical models only provide the response to simple loading conditions. More general situations require either interpolation between curves or other arbitrary assumptions. The continuously-yielding joint model, proposed by Cundall and Hart (1984) and revised by Lemos (1987), is intended to simulate the intrinsic mechanism of progressive damage of the joint under shear. This approach produces consistent responses in the varied conditions encountered in numerical modeling. The model also provides continuous hysteretic damping for dynamic simulations.

The response to normal loading is expressed incrementally as

$$\Delta\sigma_n = k_n \, \Delta u_n \qquad (10)$$

where k_n is the normal stiffness, given by $k_n = a_n \, \sigma_n^{en}$, a simple relation representing the observed increase of stiffness with normal stress, and

a_n and en are model parameters.

In general, zero tensile strength is assumed.

For shear loading, the model displays a continuous accumulation of plastic displacement from the onset of shearing. Figure 3-4 shows a typical stress-displacement curve for monotonic loading under constant normal stress. The shear stress increment is calculated as

$$\Delta\sigma_s = F \, k_s \, \Delta u_s \qquad (11)$$

where the shear stiffness can also be taken as a function of normal stress — for example,

$$k_s = a_s \, \sigma_n^{es} \qquad (12)$$

The instantaneous slope is governed by the factor F which depends on the distance from the actual stress curve to the "target" or bounding strength curve τ_m represented in the figure,

$$F = \frac{(1 - \sigma_s / \tau_m)}{(1 - r)} \tag{13}$$

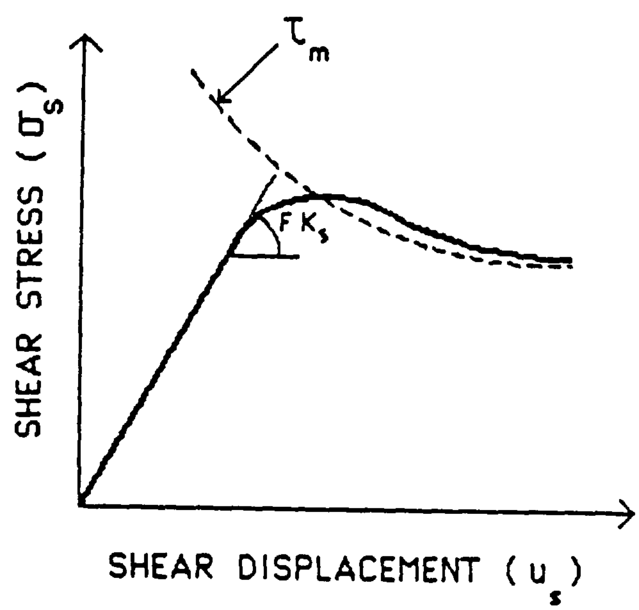


Fig. 3-4 Continuously-Yielding Joint Model: Shear Stress-Displacement Curve and Bounding Shear Strength

The factor r, which is initially zero, is intended to restore the elastic stiffness immediately after a load reversal. That is, r is set to σ_s/τ_m (and, therefore, F to 1) whenever sign (Δu_s) is not equal to sign (Δu_s^{old}). In practice, r is limited to 0.75 in order to avoid numerical noise when the shear stress is approximately equal to the bounding strength.

The bounding strength is given by

$$\tau_m = \sigma_n \tan \phi_{eff} \operatorname{sign}(\Delta u_s) \tag{14}$$

The parameter ϕ_{eff} can be understood as the friction angle that would apply if no more asperities were sheared off. As damage accumulates, this angle is continuously reduced according to the equation

$$\Delta\phi_{eff} = - 1/R \ (\phi_{eff} - \phi) \ \Delta u_s{}^p \tag{15}$$

where the plastic displacement is defined as

$$\Delta u_s{}^p = (1 - F) \ \Delta u_s \tag{16}$$

ϕ is the basic friction angle of the rock surfaces, and

R is a material parameter (with dimension of length) which expresses the joint roughness. A large value of R leads to a slower reduction of ϕ_{eff} and, therefore, to a larger peak. The peak is reached when the bounding strength equals the shear stress. After this point, the value of F becomes negative, and the joint enters the softening regime.

The above incremental relation for ϕ_{eff} is equivalent to

$$\phi_{eff} = \exp(-u_s{}^p/R) \ (\phi_{effo} - \phi) + \phi \tag{17}$$

where ϕ_{effo} is the initial value of ϕ_{eff} and represents the in-situ state of the joint.

The dilatancy angle is calculated as

$$i = \tan^{-1} [\,|\sigma_s|/\sigma_n] - \phi \tag{18}$$

— i. e., dilation takes place whenever the stress is above the residual strength level, and is obtained from the actual apparent friction angle.

Laboratory shear tests have shown that more damage is done under higher normal stress than at lower normal stress. This effect could be included by modifying the incremental equation governing the evolution of ϕ_{eff}. For example, parameter R could be multiplied by a factor of the form

$$\log_{10} (JCS/\sigma_n) \tag{19}$$

as in the so-named Barton-Bandis model (Bandis et al., 1983).

The present formulation may produce unacceptable results when large variations of normal stress accompany reversals in the direction of shearing. For example, consider the case of shearing at a given σ_n , followed by a substantial reduction in normal stress without shear motion, and then by shearing in the opposite direction. If the change in σ_n causes a large drop in the bounding strength, or a stress-dependent shear stiffness is used, in principle it is possible for the unloading curve to be above the loading curve, leading to energy production. Applications to date have not involved load paths capable of producing such behavior. However, further research and modifications of the model are required to avoid this problem.

3.4 Equations of Motion

The motion of an individual block is determined by the magnitude and direction of resultant out-of-balance moment and forces acting on it. In this section, the equations of motion which describe translation and rotation of the block about its centroid are developed. Consider the motion of a single mass acted on by a varying force, F(t). Newton's second law of motion can be written in the form

$$\frac{\partial \dot{u}}{\partial t} = \frac{F}{m} \tag{20}$$

The central difference scheme for the left-hand side of Eq. (20) at time t can be written as

$$\frac{\partial \dot{u}}{\partial t} = \frac{\dot{u}^{(t + \Delta t/2)} - \dot{u}^{(t - \Delta t/2)}}{\Delta t} \tag{21}$$

Substituting Eq. (21) in Eq. (20) and re-arranging yields

$$\dot{u}^{(t + \Delta t/2)} = \dot{u}^{(t - \Delta t/2)} + \frac{F(t)}{m} \Delta t \tag{22}$$

With velocities stored at the half-timestep point, it is possible to express displacement as

$$u^{(t + \Delta t)} = u^{(t)} + \dot{u}^{(t + \Delta t/2)} \Delta t \tag{23}$$

Because the force depends on displacement, the force/displacement calculation is done at one time instant. Figure 3-5 illustrates the central difference scheme with the order of calculation indicated by the arrows.

For blocks which are acted upon by several forces as well as gravity, the velocity equations become:

$$\begin{aligned} \dot{u}_i^{(t + \Delta t/2)} &= \dot{u}_i^{(t - \Delta t/2)} + \left(\frac{\Sigma F_i^{(t)}}{m} + g_i \right) \Delta t \\ \dot{\theta}^{(t + \Delta t/2)} &= \dot{\theta}^{(t - \Delta t/2)} + \frac{\Sigma M^{(t)}}{I} \Delta t \end{aligned} \tag{24}$$

where $\dot{\theta}$ = angular velocity of block about centroid,

I = moment of inertia of block, and

$\dot{u}_i$ = velocity components of block centroid.

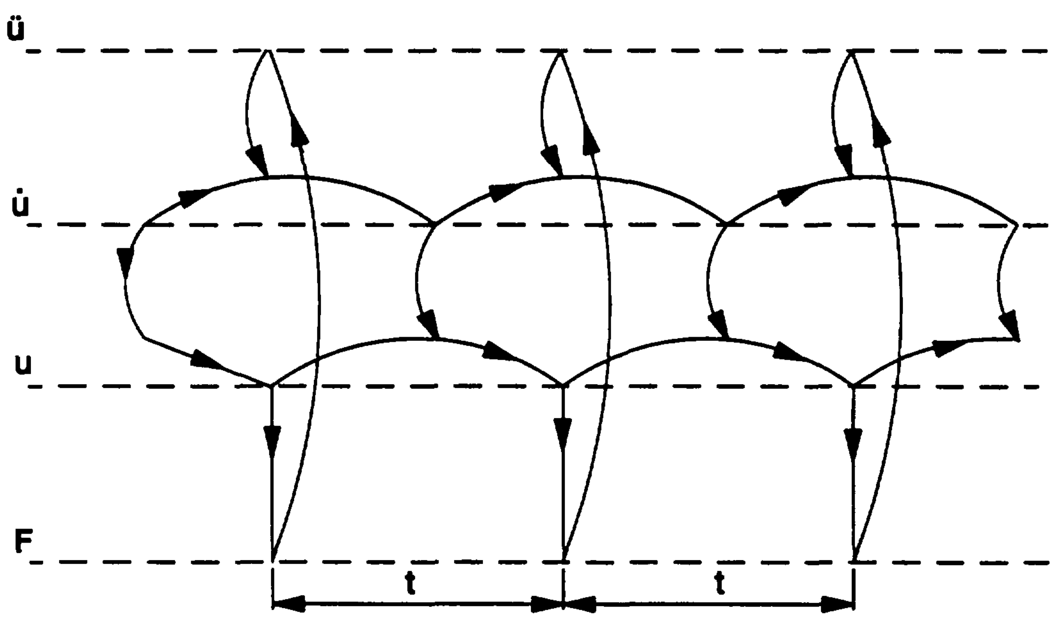


Fig. 3-5 Interlaced Nature of Calculation Cycle Used in Distinct Element Formulation

The new velocities in Eq. (24) are used to determine the new block location according to

$$x_i(t + \Delta t/2) = x_i(t) + \dot{u}(t + \Delta t/2)\ \Delta t$$

(25)

$$\theta_i(t + \Delta t/2) = \theta_i(t) + \dot{\theta}^{(t + \Delta t/2)}\ \Delta t$$

where θ_i = rotation of block about centroid, and

x_i = coordinates of block centroid.

Thus, each iteration produces new block positions which generate new contact forces. Resultant forces and moments are used to calculate linear and angular accelerations of each block. Block velocities and displacements are determined by integration over incremental timesteps. The procedure is repeated until a satisfactory state of equilibrium or mode of failure results.

3.5 Calculation Sequence

In all explicit, time-marching schemes, the main calculation cycle consists of applying the law of motion to all mass points, followed by the calculation of force increments from displacement increments for all spring-like elements (contacts and continuum zones). The cycle progresses as follows:

For All Blocks

- accelerate centroids from force-sums
- accelerate gridpoint masses from internal and boundary forces for deformable blocks
- update corner velocities and displacements
- apply new relative velocities to surrounding contacts
- reset force-sums to zero

For All Contacts

- update contact forces from known relative contact velocities using constitutive model
- accumulate centroid force-sums and gridpoint force-sums
- compute strain rates; hence, new stresses; hence, gridpoint forces

3.6 Static Analysis

For static solutions, damping must be used to dissipate vibrational energy in order that the system converges to a steady state; otherwise, the system will oscillate indefinitely. Two forms of viscous damping are available in the distinct element formulation: mass proportional damping and stiffness proportional damping. Mass-proportional damping has an effect similar to that of immersing the block assembly in a viscous fluid — i.e., absolute motion relative to the frame of reference is damped. Stiffness proportional damping is physically equivalent to dashpots across contacts and serves to damp block relative motion.

The dashpots across the contacts operate both in the shear and normal directions; the shear dashpot is "switched off" during sliding. For an elastic continuous system (one in which there is no slip or breaking and making of new contacts) the damping scheme described above is termed Rayleigh damping. For a discontinuous system that dissipates energy in slip, the theory does not apply, but damping still occurs and can be understood in terms of the physical effects of each type of dashpot.

Either type of damping can be used separately or together. Mass-proportional damping is effective in reducing low-frequency motion, where the whole block assembly "sloshes" from side to side. Stiffness-proportional damping is more effective against the high-frequency noise of individual blocks "rattling" against their neighbors.

Physically, mass proportional damping can be regarded as a set of viscous dashpots connected to the centroid of each block. The dashpots generate a force that opposes the block velocity and is proportional to both the velocity and the block mass. The equation of motion, including viscous damping, can be written as

$$\frac{\partial \dot{u}}{\partial t} = \frac{F}{m} - \alpha \dot{u} + g \tag{26}$$

where α is the damping constant.

A new difference equation in time can be written as:

$$\frac{\dot{u}^{(t+\Delta t/2)} - \dot{u}^{(t-\Delta t/2)}}{\Delta t} = \frac{F}{m} - \alpha \left[\frac{\dot{u}^{(t+\Delta t/2)} + \dot{u}^{(t-\Delta t/2)}}{2} \right] + g \tag{27}$$

Note that the damping force in the equation is centered at time t. Re-arranging,

$$\dot{u}^{(t+\Delta t/2)} = \left[\dot{u}^{(t-\Delta t/2)} \left(1 - \frac{\alpha \Delta t}{2}\right) + \left(\frac{F}{m} + g \right) \Delta t \right] / (1 + \alpha \Delta t/2) \tag{28}$$

Rotations are damped in an identical manner.

<u>Boundary Element Representation of the Far Field</u>

When performing static analyses, the problem of defining boundary conditions for a finite numerical model of an unbounded medium can be adequately handled by coupling the block assembly to a boundary element representation of the far-field. Because non-linear behavior is usually confined to the vicinity of the structure or excavation under study, the assumption of a linear elastic far-field is justified. A hybrid rigid block-boundary element model was developed by Lorig (1984) for the analysis of underground excavations in rock. A half-plane formulation for the boundary element region was used by Lemos (1983) in a coupled distinct element-boundary element model appropriate for the study of foundations or shallow excavations. A similar scheme is implemented in UDEC. The boundary element formulation follows the work of Brady and Wassyng (1981).

The boundary element region is represented by a stiffness matrix K which relates the forces and displacements at the interface of the two domains. Either an infinite plane or an half-plane solution can be used. The elastic moduli of the far-field domain should reflect the deformability of the jointed rock system. At every time step, the motion of the blocks defines the displacements at the interface. The boundary element domain provides elastic reaction forces given by

$$F = - K u \tag{29}$$

The dynamic analysis reported in the next section usually starts from some in-situ condition. Normally, a simple uniform stress field is assumed. A more realistic stress distribution can be simulated with a hybrid distinct element-boundary element model. Then, before the dynamic input is applied, the boundary element boundaries can be replaced by non-reflecting boundaries, provided the boundary element reaction forces are maintained throughout the dynamic loading phase.

3.7 Dynamic Analysis

Experience with numerous dynamic analyses has shown that, for good accuracy with the distinct element method, the wave length of the highest frequency of interest in the model should be at least 8 times the width of the largest finite-difference zone. The appropriate relation for specifying the mesh is determined by the frequency of the input record and the elastic properties of the medium. For example, assume that a maximum frequency of 100 Hz is to be propagated through a rock column with a bulk modulus of 1 GPa, a shear modulus of 0.15 GPa, and a density of 2610 kg/m^3. The compressional wave speed would be 664 m/sec, and the wavelength at 100 Hz would be 6.64 m. If eight zones per wavelength are required, then the maximum zone size would be 0.83 m.

For dynamic input with a high peak velocity and short rise-time, this requirement may necessitate a very fine spatial mesh and correspondingly fine integration time mesh. The effect is compounded in discontinuum codes because the wave propagation across discontinuities can produce higher frequency components than are provided in the input wave. The consequence is that reasonable analyses may be prohibitively time and memory consuming as well as extremely expensive. In such cases, it may be possible to adjust the input by recognizing that most of the power for the input history is composed of lower frequency components. By filtering the history and removing high frequency components, a coarser mesh may be used without significantly affecting the results.

The filtering procedure can be accomplished with a low-pass filter routine such as the Fast Fourier Transform technique. For example, the unfiltered velocity record shown in Fig. 3-6 represents a typical waveform containing a very high frequency spike. The highest frequency of this input exceeds 50 Hz but, as shown by the power spectral density plot of Fourier amplitude versus frequency (Fig. 3-7), most of the power (approximately 99%) is made up of components of frequency 15 Hz or lower. It can be inferred, therefore, that by filtering this velocity history with a 15 Hz low-pass filter, less than 1% of the power is lost. The input filtered at 15 Hz is shown in Fig. 3-8(a), and the Fourier amplitudes are plotted in Fig. 3-8(b). The difference in power between unfiltered and filtered input is less than 1%, while the peak velocity is reduced 38% and the rise time is shifted from 0.035 sec to 0.09 sec. Analyses should be performed with input at different levels of filtering to evaluate the influence of the filter on model results.

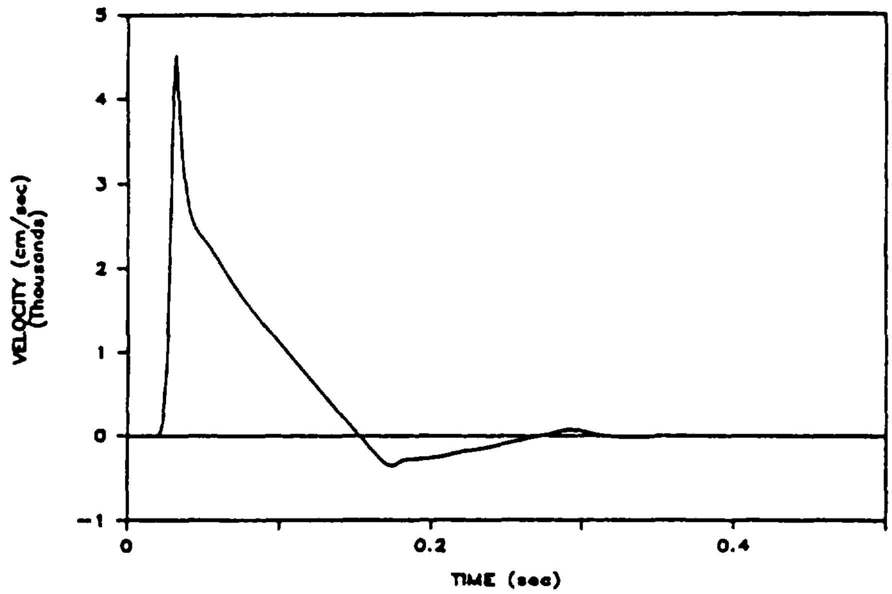


Fig. 3-6 Unfiltered Velocity History

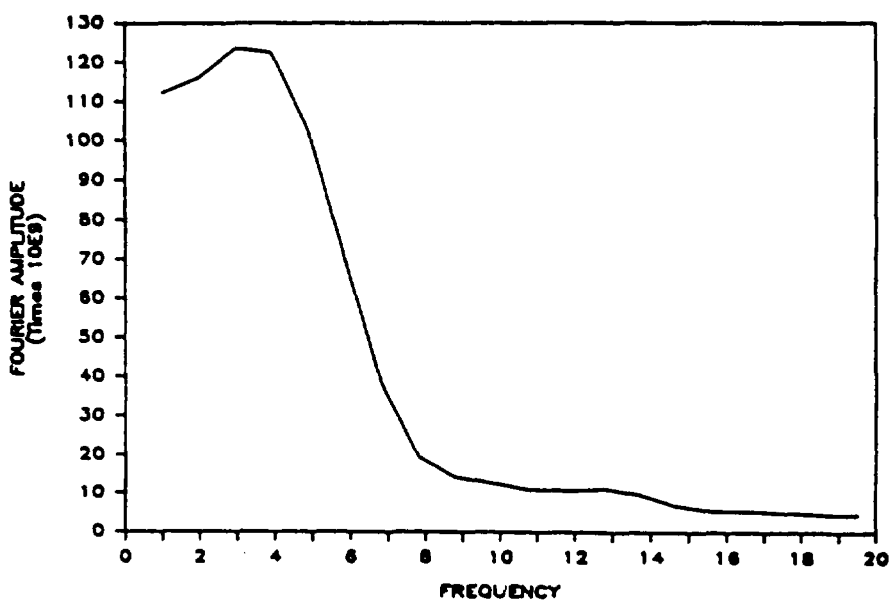


Fig. 3-7 Unfiltered Power Spectral Density Plot

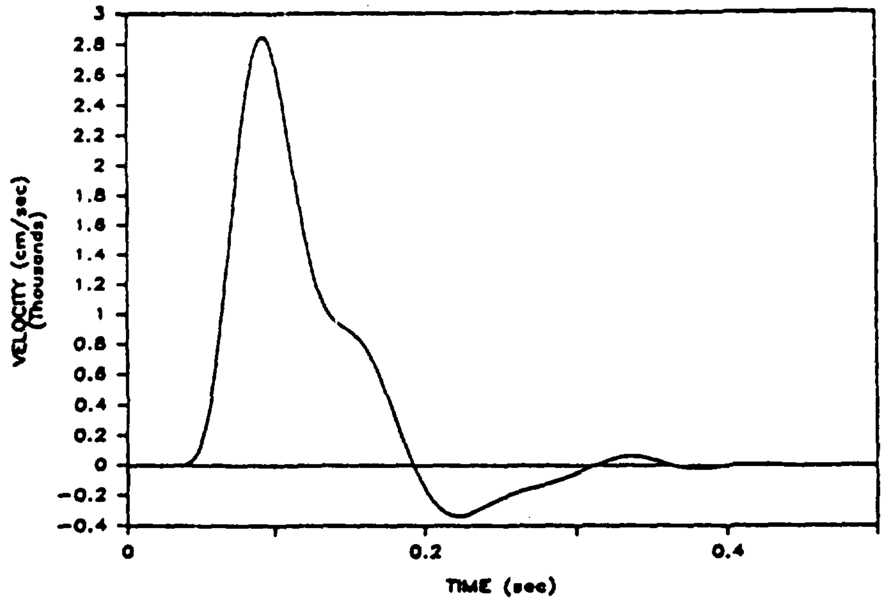


Fig. 3-8(a) Filtered Velocity History at 15 Hz

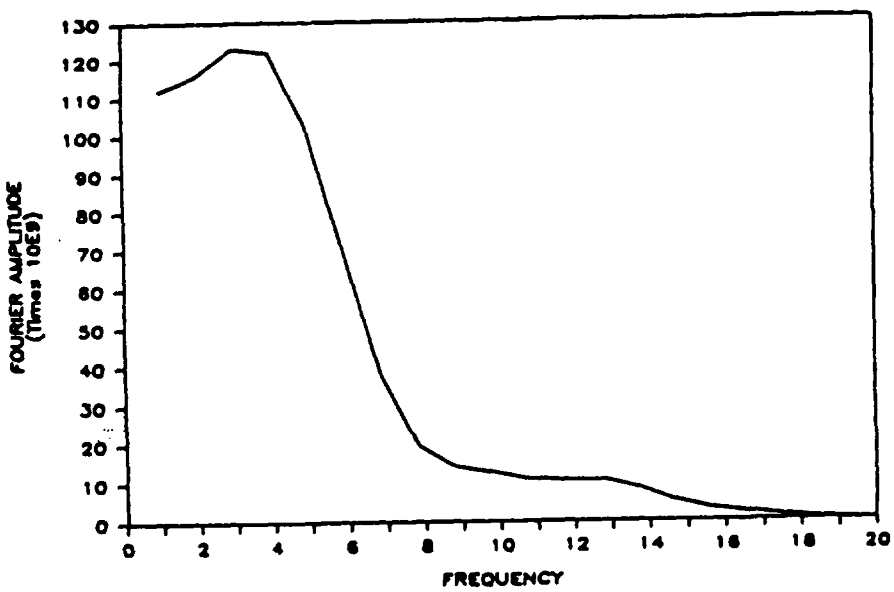


Fig. 3-8(b) Results of Filtering at 15 Hz

3.8 Boundary Conditions for Dynamic Analysis

3.8.1 Non-Reflecting Boundaries

3.8.1.1 Dynamic Modeling of Unbounded Media — The modeling of geomechanics problems involves media which, at the scale of the analysis, are better represented as unbounded. Deep underground excavations are normally assumed to be surrounded by an infinite medium, while surface and near-surface structures are supposed to lie on a half-space. Numerical methods relying on the discretization of a finite region of space require that appropriate conditions be enforced at the artificial numerical boundaries. In static analyses, fixed or elastic boundaries (e.g., represented by boundary element techniques) can be realistically placed at some distance from the region of interest. In dynamic problems, however, such boundary conditions cause the reflection of outward propagating waves back into the model and do not allow the necessary energy radiation. The use of a larger model can minimize the problem, since material damping will absorb most of the energy in the waves reflected from distant boundaries. However, this solution leads to large computational costs. The alternative is to use non-reflecting (or absorbing) boundaries. Several formulations have been proposed. The viscous boundary developed by Lysmer and Kuhlemeyer (1969) is used in UDEC. It is based on the use of independent dashpots and is nearly totally effective for body waves approaching the boundary at angles of incidence above 30°. For lower angles of incidence or surface waves, the energy absorption is only approximate. However, it has the advantage of being an inexpensive technique which can be used in time domain analyses. Its effectiveness has been demonstrated in both finite element and finite difference models (Kunar et al., 1977). A variation of this technique proposed by White et al. (1977) is also widely used.

More efficient energy absorption (for example, in the case of Rayleigh waves) requires the use of frequency-dependent dashpots, which can only be used in frequency domain analyses (e.g., Lysmer and Waas, 1972). These are usually designated as consistent boundaries and involve the calculation of dynamic stiffness matrices coupling all the boundary degrees of freedom. Boundary element methods may be used to derive these matrices (e.g., Wolf, 1985). A comparative study of the performance of different types of elementary, viscous and consistent boundaries was reported by Roesset and Ettouney (1977).

A different procedure to obtain efficient absorbing boundaries for use in time domain studies was proposed by Cundall et al. (1978). It is based on the superposition of solutions with stress and velocity boundaries in such a way that reflections are canceled. In practice, it requires the use of two parallel, overlapping grids in a narrow region adjacent to the boundary, whose results are added. This method has been shown to provide effective energy absorption, but is difficult to implement for a blocky system with complex geometry.

3.8.1.2 Numerical Implementation — The viscous boundaries proposed by Lysmer and Kuhlemeyer (1969) consist of independent dashpots attached to the boundary in the normal and shear directions. They provide viscous normal and shear tractions given by

$$\begin{aligned} t_n &= -\rho\, c_p\, v_n \\ t_s &= -\rho\, c_s\, v_s \end{aligned} \tag{30}$$

where v_n and v_s are the normal and shear components of the velocity at the boundary, ρ is the mass density, and c_p and c_s are the P- and S-wave velocities.

These viscous terms can be introduced directly into the equations of motion of the gridpoints lying on the boundary. A different approach, however, was implemented in UDEC, in which the tractions t_n and t_s are calculated and applied at every timestep in the same way as the boundary loads. This alternative scheme allows the viscous boundaries to be used also with rigid and simply-deformable blocks. Tests have shown that this implementation is equally effective. The only potential problem concerns numerical stability, because the viscous forces are calculated from velocities lagging by half a timestep. In practical analyses to date, no reduction of timestep has been required by the use of the non-reflecting boundaries. Timestep restrictions demanded by high joint stiffnesses or small zones are usually more important.

3.8.2 Dynamic Free-Field

3.8.2.1 Boundary Conditions for Seismic Analysis — Seismic analysis of surface and embedded structures by numerical techniques requires the discretization of a region of the soil or rock adjacent to the foundation. The seismic input is normally represented by a plane wave propagating upwards (Fig. 3-9). This dynamic excitation can be applied as a stress wave at the base of the model (AC), a non-reflecting boundary, as explained in the previous section. The boundary conditions at the sides AB and CD must account for the free-field motion which would exist in the absence of the structure. A simple solution is to extend the model laterally so that free-field conditions are achieved. For soils with high material damping, this condition can be obtained within a relatively small distance (Seed et al., 1975). However, when the material damping is low, the required distance may lead to an impractical model. An alternative procedure is to "enforce" the free-field motion in such a way that boundaries AB and CD retain their non-reflecting properties — i.e., outward waves originating from the structure are properly absorbed. This approach was used in the continuum finite-difference code NESSI (Cundall et al., 1980). A technique of this type was developed for UDEC. It involves the execution of a one-dimensional free-field calculation in parallel with the blocky system analysis. The lateral boundaries AB and CD are coupled to the free-field grid by viscous dashpots (Fig. 3-9).

An application of showing use of the dynamic free-field boundary condition in UDEC is given in Appendix B (Problem D). A verification problem demonstrating that the technique provides proper lateral conditions for seismic analysis is given by Lemos (1987).

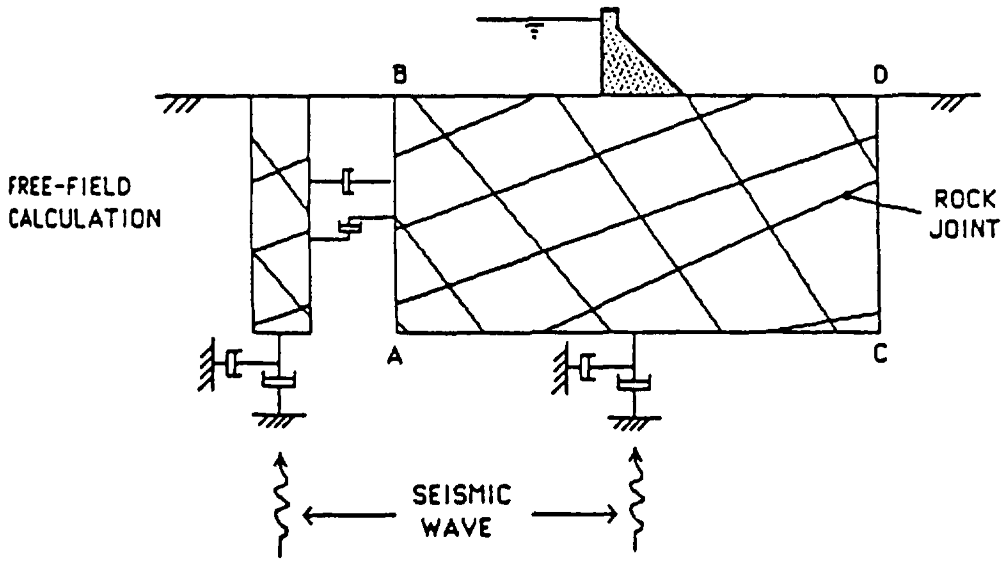


Fig. 3-9 Model for Seismic Analysis of Surface Structures: Block Assembly and Free-Field Mesh

3.8.2.2 Free-Field Representation — The free-field model consists of a one-dimensional "column" of unit width, simulating the behavior of the extended jointed medium. An explicit finite-difference method was selected for the model. The height of the free-field equals the length of the lateral boundaries of the blocky system. It is discretized into n elements of equal length Δy. Element masses are lumped at the n+1 gridpoints. A linear variation of the displacement field is assumed within each element, which are therefore in a state of uniform strain (and stress). Because all quantities are independent of the horizontal coordinate x, the element deformations are given by

$$\varepsilon_{xx} = 0 \quad , \quad \varepsilon_{yy} = \frac{\partial u_y}{\partial y} \quad \text{and} \quad \tau_{xy} = \frac{\partial u_x}{\partial y} \tag{31}$$

Finite-difference approximations to these expressions are

$$\varepsilon_{yy}{}^{i} = \frac{u_y{}^{i+1} - u_y{}^{i}}{\Delta y} \qquad \text{and} \qquad \tau_{xy}{}^{i} = \frac{u_x{}^{i+1} - u_x{}^{i}}{\Delta y} \tag{32}$$

where element i lies between gridpoints i and i+1.

Stresses are calculated by application of the constitutive relations. Gridpoint forces are assembled from the element stresses as

$$F_x{}^{i} = \sigma_{xy}{}^{i} + \sigma_{xy}{}^{i-1} \quad \text{and} \quad F_y{}^{i} = \sigma_{yy}{}^{i} + \sigma_{yy}{}^{i-1} \tag{33}$$

The time integration uses the same central-difference scheme described in Section 3.4. Damping and numerical stability are handled in a similar fashion.

At the gridpoint at the base of the free-field, shear and normal dashpots provide the absorbing boundary conditions, and the dynamic input is applied in the form of a stress record.

3.8.2.3 Coupling With the Blocks — The free-field calculation provides gridpoint velocities $v_x{}^f$ and $v_y{}^f$ and element stresses $\sigma_{xx}{}^f$ and $\sigma_{xy}{}^f$. In order to achieve the required boundary conditions along the l.h.s. AB, the following stresses must be applied

$$\begin{aligned} \sigma_{xx} &= \sigma_{xx}{}^f + \rho c_p \, (v_x - v_x{}^f) \\ \sigma_{xy} &= \sigma_{xy}{}^f + \rho c_s \, (v_y - v_y{}^f) \end{aligned} \tag{34}$$

where v_x and v_y are the components of the block gridpoint velocity.

Along the r.h.s. CD, the sign of the second term must be reversed.

Figure 3-10 illustrates how the coupling of the free-field mesh with a block edge EF is executed. The contribution from the free-field stresses in Eqs. (34) is applied in the form of concentrated forces at discrete points on the block edge, at the same y-coordinate of the free-field gridpoints. For example, at point i, the following forces are applied

$$P_x{}^i = \frac{1}{2}\left[(\sigma_{xx}{}^f)^{i-1} + (\sigma_{xx}{}^f)^i \right] \Delta y$$

$$P_y{}^i = \frac{1}{2}\left[(\sigma_{xy}{}^f)^{i-1} + (\sigma_{xy}{}^f)^i \right] \Delta y \qquad (35)$$

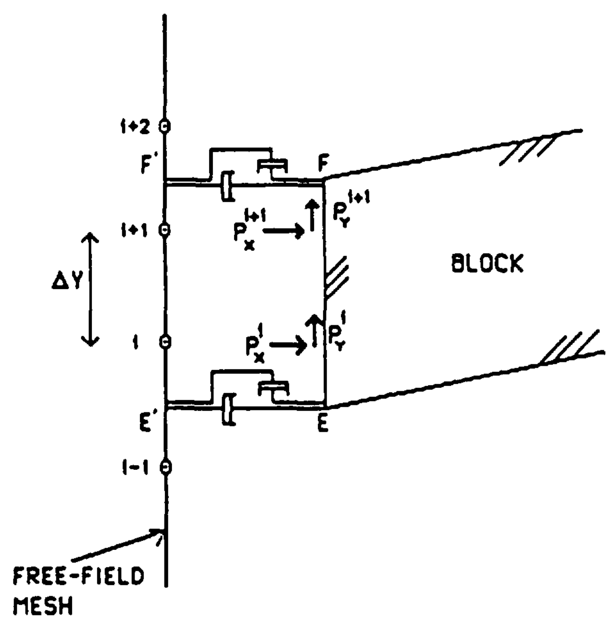


Fig. 3-10 Coupling of Block Model and Free-Field Mesh

For the r.h.s. boundary, the negative of these expressions is applied. The second term of Eq. (34) is applied at the block corners (gridpoints) E and F. For example, at E, the forces to be applied are

$$R_x = - \rho c_p \; (v_x^E - v_x^{fE}) \; l$$
$$R_y = - \rho c_s \; (v_y^E - v_y^{fE}) \; l \qquad (36)$$

where l is the block edge half-length.

The free-field velocities v_x^{fE} and v_y^{fE}, at point E', are obtained by linear interpolation between the adjacent gridpoint velocities (in this case, i-1 and i). The free-field mesh should be fine enough to guarantee an adequate number of support points for the blocks, otherwise excessive slip may occur at some boundary joints.

Normally, the dynamic loading follows a static calculation corresponding to the in-situ conditions. In this case, the in-situ free-field stresses are subtracted from $\sigma_{xx}{}^f$ and $\sigma_{xy}{}^f$ before forces $P_x{}^i$ and $P_y{}^i$ are calculated. At the same time, the reaction forces (and external loads) along the lateral boundaries which provided equilibrium to the block system in the in-situ state must be stored, so that they are applied to the blocks in addition to the free-field forces during the dynamic loading.

3.9 Tunnel Generator

Efficient analysis of problems involving excavations require a simple means for specifying the excavation boundary. The present generator logic creates a circular joint or crack pattern for the simulation of excavations with circular cross-sections. The user is allowed to choose the center point of the circular pattern, its radius, and the number of straight segments or sides defining the circle. An example of a tunnel generated in a regularly-jointed media is shown in Fig. 3-11.

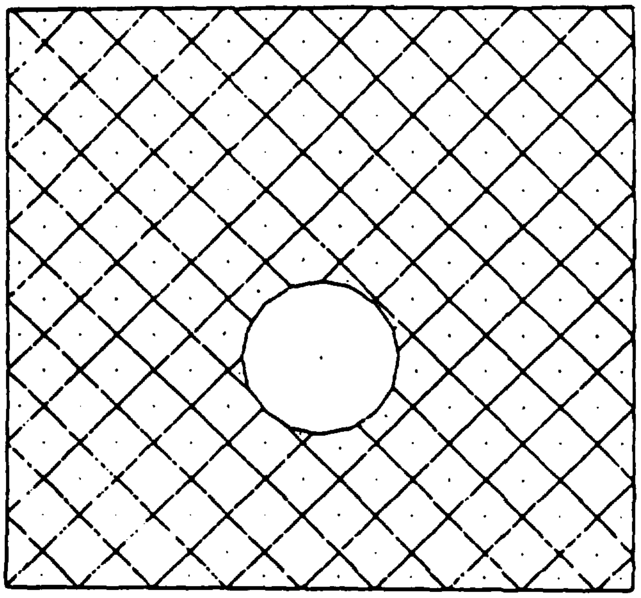


Fig. 3-11 Circular Tunnel in Jointed Rock

Other excavation geometries can be formed based on the arc generator. For example, a horseshoe profile can be obtained by using the arc generator and supplemental crack commands, as shown in Fig. 3-12.

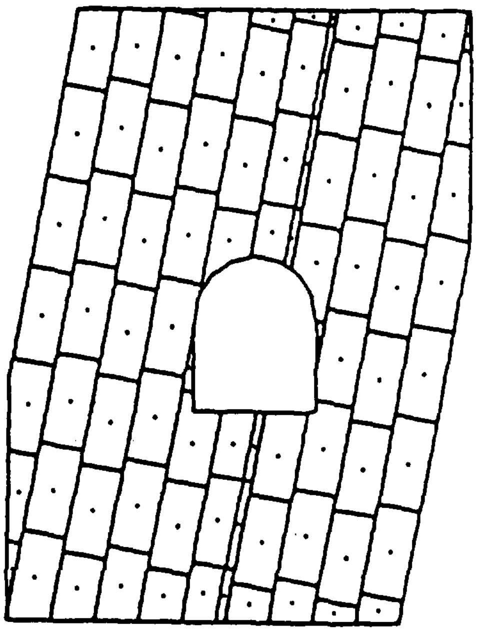


Fig. 3-12 Horseshoe-Shaped Excavation in Jointed Rock

3.10 Rock Reinforcement

In practice, several types of rock reinforcement systems exist which handle a great variety of ground conditions and applications. Two major types are considered representative of most reinforcement systems. One system, typified by reinforcing bars encapsulated with resin, exhibits significant resistance to shear at a discontinuity and, also, is capable of developing large axial loads in the reinforcement due to the high strength characteristics of the resin. The second system, typified by cement-grouted cables, exhibits little shear resistance at joints and requires significant bond length to develop large axial capacities.

The characteristic behavior of these two rock reinforcement systems have been incorporated into UDEC. The first formulation is based on that originally developed for a rigid-block version of the code (Lorig, 1985). This reinforcement formulation considers only the local effect of reinforcement where it passes through existing discontinuities. The second formulation considers the presence of the reinforcement throughout the rock mass and does not explicitly model resistance of the reinforcement to shear deformation at discontinuities (although shear resistance will result naturally from deformation of the reinforcement). The two formulations are described in the following subsections.

3.10.1 A Numerical Formulation for Rock Reinforcement Which Considers Local Restraint Across Discontinuities — Analysis of reinforced excavations in jointed rock requires that the functions of reinforcement be considered explicitly. This can be achieved in the computational model by calculating, for each element, the forces generated by displacements across the discontinuity through which the element passes. The following description of this approach is taken from Lorig (1985).

The reinforcement model presented here uses simple force-displacement relations to describe both shear and axial behavior of the reinforcement element. Large shear displacements are accounted for by assuming simple geometric changes develop locally in the reinforcement near a discontinuity. The resultant representation is most applicable to cases where deformation of individual rock blocks may be neglected in comparison with deformation of the reinforcing system. In such cases, attention may be focused reasonably on the effect of reinforcement near discontinuities.

3.10.1.1 Axial Force-Displacement Relation for Fully-Bonded Rock Reinforcement — Historically, testing of rock reinforcement has focused on pull-out tests for two reasons:

(1) ease of experimentation and interpretation or results; and

(2) provision of axial restraint (the main function of reinforcement in the prevailing conceptual models).

Consequently, a relatively good understanding of axial force-displacement relations has been achieved. The axial force-displacement relation used in the representation of rock reinforcement considered here is shown in Fig. 3-13.

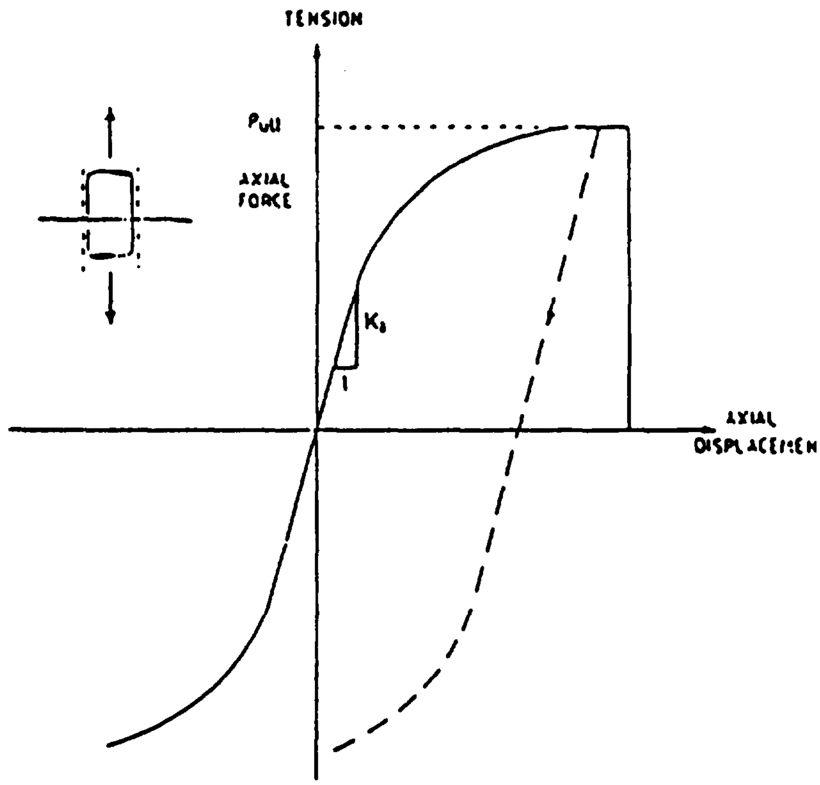


Fig. 3-13 Axial Behavior of Reinforcement System

Figure 3-13 indicates an identical response in tension and compression. This may not be the case for all reinforcing systems. The explicit formulation used in the distinct element method allows any force-displacement relation to be incorporated in the numerical scheme. In particular, for a specific reinforcement problem, the results of pull-out tests may be used to define the exact force-displacement relation to be used. If these results are not available, the force-displacement relation may be defined by a continuously-yielding model. The yield model used in the present study is a continuous, non-linear, force-displacement algorithm written in terms of the axial stiffness, the ultimate capacity, and a yield function. The yield function describes the force-displacement path followed in approaching the ultimate capacity.

The following theoretical expression given by Gerdeen et al. (1977) may be used to estimate the axial stiffness, K_a, for fully-bonded solid reinforcing elements:

$$K_a = \pi k d_1 \tag{37}$$

where d_1 = reinforcement diameter,

$k = [\frac{1}{2} G_g E_b / (d_2/d_1 - 1)]^{1/2}$,

G_g = grout shear modulus,

E_b = Young's modulus of reinforcement material, and

d_2 = hole diameter.

Comparisons with finite element analyses (Gerdeen et al., 1977), as well as limited comparisons with laboratory data for this investigation, indicate that Eq. (37) tends to slightly over-estimate axial stiffnesses.

The ultimate axial capacity of the reinforcement depends on a number of factors, including strength of the reinforcing element, bond strength, hole roughness, grout strength, rock strength, and hole diameter. In the absence of results of physical tests, empirical relations may be used to estimate the ultimate anchorage strength P_{ult}. One such relation for resin-grouted reinforcement is given by Littlejohn and Bruce (1975):

$$P_{ult} = 0.1\ \sigma_c\ \pi d_2\ L \tag{38}$$

where σ_c = uniaxial compressive strength of the host rock material; and

L = bond length.

3.10.1.2 Shear Force-Displacement Relation for Fully-Bonded Rock Reinforcement — Recognition that reinforcement also acts to modify the shear stiffness and strength of discontinuities has led to laboratory shear testing of reinforced discontinuities.

Experimental results and theoretical investigations indicate that shearing along a discontinuity induces bending stresses in the reinforcement that decay very rapidly with distance into the rock from the shear surface. Typically, within one to two reinforcing element diameters, the bending stresses are insignificant.

The shear force-displacement relation used in the present study is shown in Fig. 3-14. The figure shows representative responses for reinforcement at various attitudes with respect to the traversed discontinuity and direction of shear. Each curve is characterized by a continuously-yielding model defined by a shear stiffness, an ultimate strength, and a yield function.

If the results of physical tests are not available, the shear stiffness, K_s, may be estimated using the following expression from Gerdeen et al (1977):

$$K_s = E_b\ I\ \beta^3 \tag{39}$$

where $\beta = [K/(4E_bI)]^{1/2}$,

$K = 2E_g/(d_2/d_1-1)$,

I = second moment of area of the reinforcement element, and

E_g = Young's modulus of the grout.

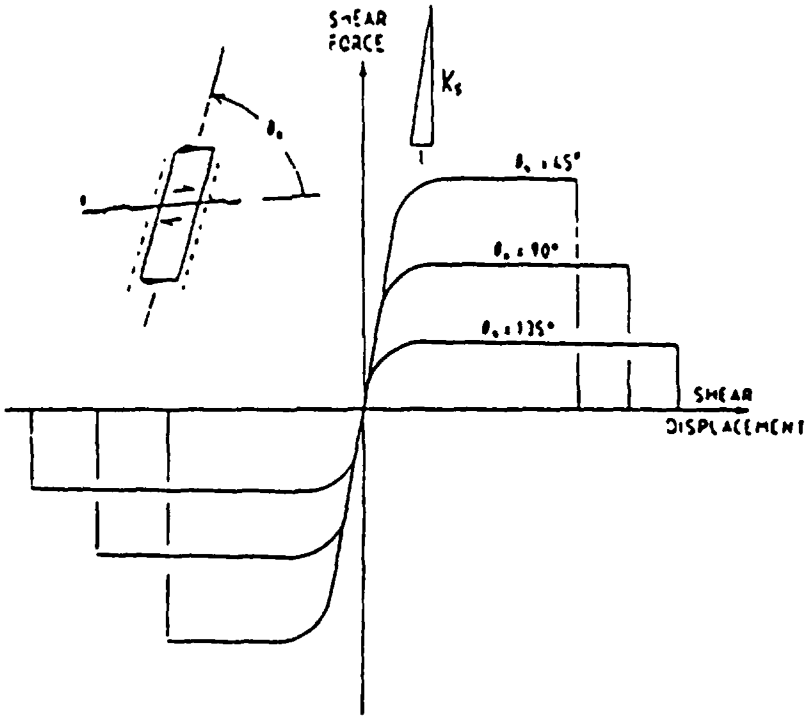


Fig. 3-14 Shear Behavior of Reinforcement System

Empirical relations can be used to estimate the maximum shear force, $F_{s,b}^{max}$, for a reinforcement element at various orientations with respect to a transgressed discontinuity and direction of shear. For example, Bjurstrom (1974) used the results of shear tests of ungrouted reinforcement perpendicular to a discontinuity in granite to develop the following expression:

$$F_{s,b}^{max} = 0.67\ d_1^2\ (\sigma_b \sigma_c)^{1/2} \tag{40}$$

where σ_b = yield strength of reinforcement.

In their assessment of maximum shear resistance, St. John and Van Dillen (1983) apply the results of Azuar et al. (1979). The latter found that the maximum shear force was about half the product of the uniaxial tensile strength of the reinforcement and its cross-sectional area, for reinforcement perpendicular to the discontinuity. The force increased to 80-90% of that product for reinforcement inclined with the direction of shear. Shear displacements causing rupture were reported after approximately two reinforcement diameters for the perpendicular case and one diameter for the inclined case. St. John and Van Dillen interpret differences between strength and amount of displacement before rupture in terms of the extent of crushing of rock around the reinforcement.

In the present formulation, the maximum shear force, $F_{s,b}^{max}$, for various orientations was calculated from the expression:

$$F_{s,b}^{max} = F_s^{max}\left[1 + [\mathrm{sign}(\cos\theta_o, \Delta u_s) \bullet \cos(\theta_o)]\right]/2 \qquad (41)$$

where $F_s^{max} = \pi\, d_1^2\, \sigma_b/4$,

Δu_s = incremental change in shear displacement (see Fig. 3-15), and

$\mathrm{sign}(\cos\theta_o, \Delta u_s)$ assigns the sign of Δu_s to $\cos(\theta_o)$.

It is seen that the maximum shear force, $F_{s,b}^{max}$, decreases from a maximum at

$$\theta_o = 0° \text{ to } 50\% \text{ of } F_s^{max} \text{ at } \theta_o = 90°$$

which is consistent with the results of Azuar et al. (1979).

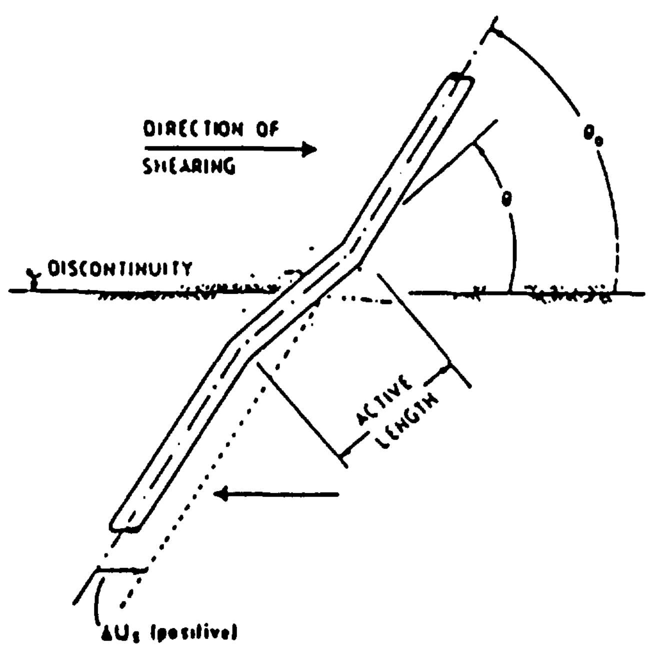


Fig. 3-15 Assumed Reinforcement Geometry After Shear Displacement, Δu_s

3.10.1.3 Numerical Modeling of Rock Reinforcement — Laboratory tests of fully-grouted untensioned reinforcement in good quality rocks with one discontinuity indicate that strains in the reinforcement are concentrated across the discontinuity [Bjurstrom (1979) and Pells (1974)]. This observation forms the basis of the numerical description of the behavior of rock reinforcement used here. The model assumes that, during shear displacement along a discontinuity, the reinforcement deforms as shown in Fig. 3-15. The short length of reinforcement which spans the discontinuity and changes of orientation during shear displacement is referred to as the active length. The assumed geometric changes were originally suggested in a derivation by Haas (1976) for conventional point-anchored reinforcement and adopted by Fuller and Cox (1978) in considering fully-grouted reinforcement.

In the model, it is assumed that the active length changes orientation only as a direct geometric result of shear and normal displacements at the discontinuity. Methods for estimating the active length are presented in the next section. The model may be considered to consist of two springs located at the discontinuity interface and oriented parallel and perpendicular to the reinforcement axis, as shown in Fig. 3-16(a). Following shear displacement, the axial spring is oriented parallel to the active length, while the shear spring remains perpendicular to the original orientation, as shown in Fig. 3-16(b). Similar geometric changes follow displacements normal to the discontinuity.

The force-displacement relations described previously are used to determine forces arising in the springs from incremental displacements of end points of the active length.

In computing the incremental axial displacement of the active length, it is necessary to account for crushing of the grout and/or rock near the discontinuity as shear displacement causes the reinforcement to bear against one side of the hole. In the present model, a reduction factor, r_f, was applied to incremental axial displacements arising from changes in orientation of the active length to account for the crushing. The reduction factor is computed from the following expression:

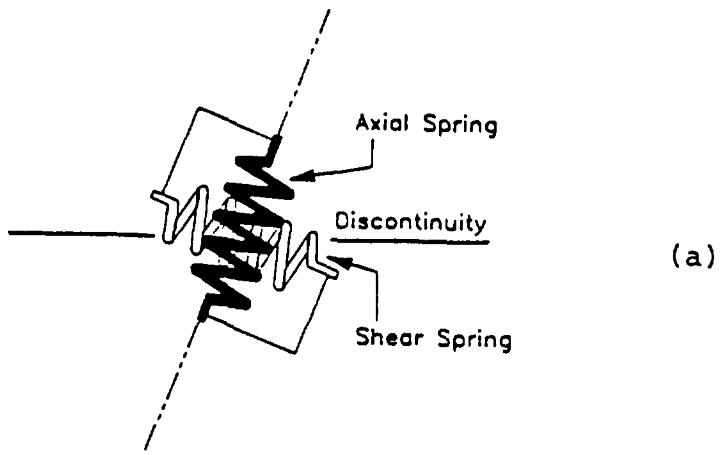
$$r_f = |u_{axial}| \, (u_s^2 + u_n^2)^{-1/2} \tag{42}$$

where u_{axial} = summation of axial displacement increments (i.e., discontinuity displacement increments resolved at each configuration in the direction of the active length),

u_s = total discontinuity shear displacement, and

u_n = total discontinuity normal displacement.

Note that no reduction ($r_f = 1.0$) is applied for cases in which there is no change in orientation of the active length.



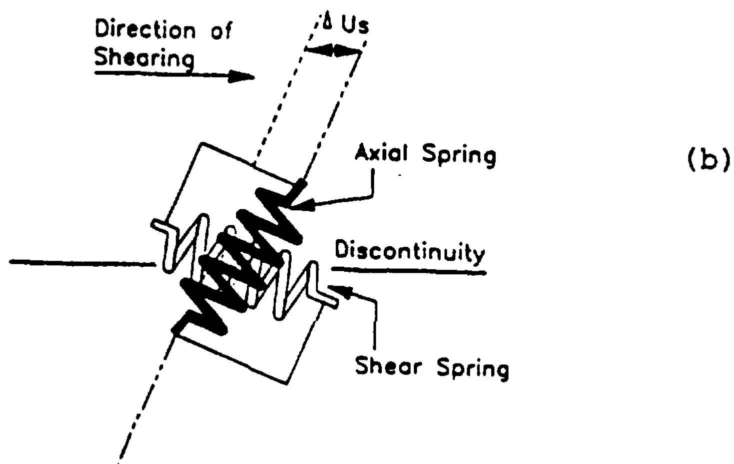


Fig. 3-16 Orientation of Shear and Axial Springs Representing Reinforcement Prior To and After Shear Displacement

Resultant shear and axial forces are resolved into components parallel and perpendicular to the discontinuity, as shown in Figs. 3-17(a) and (b). Forces are then applied to the neighboring blocks.

Both shear and axial force-displacement relations employ a continuously-yielding model. The yield model for shear behavior is described in incremental form by the expression:

$$\Delta F_s = K_s \; |\Delta u_s| f(F_s) \tag{43}$$

where ΔF_s is an incremental change in shear force,

Δu_s is an incremental change in shear displacement,

K_s is the shear stiffness, and

$f(F_s)$ is a function describing the path by which the shear force, F_s, approaches the ultimate or bounding shear force $F_{s,b}^{max}$.

In the present work, the function

$$f(F_s) = |F_{s,b}^{max} - F_s| (F_{s,b}^{max} - F_s) / (F_{s,b}^{max})^2 \tag{44}$$

is used to represent the yield curve. From Eq. (43), the shear force "seeks" the bounding force in an asymptotic manner. An identical yield function involving axial forces is used to describe axial behavior.

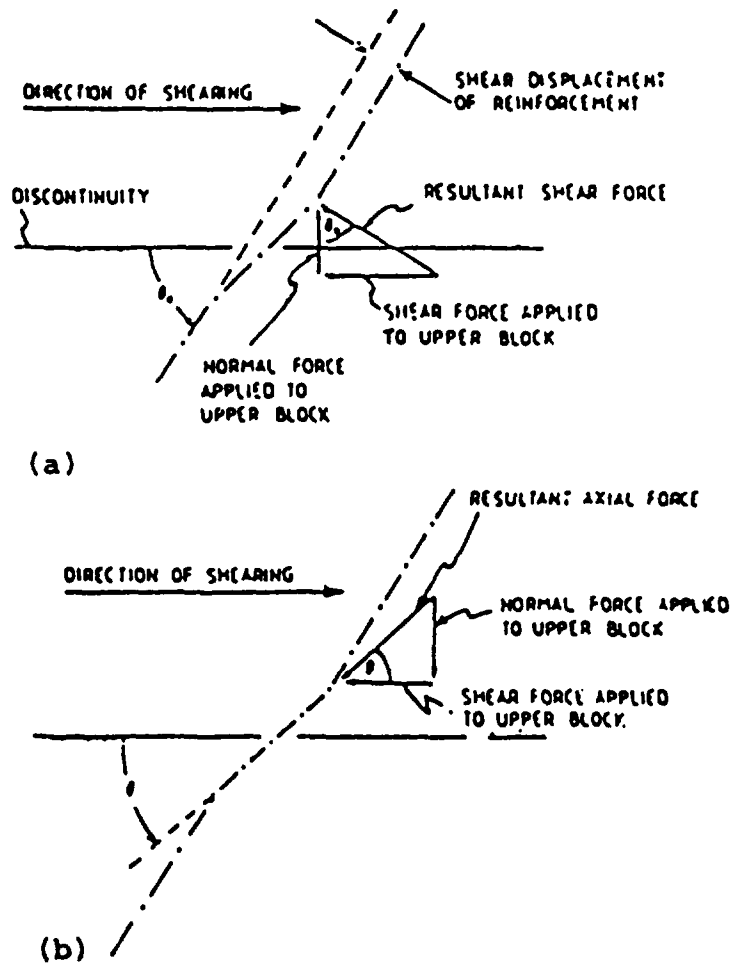


Fig. 3-17 Resolution of Reinforcement Shear and Axial Forces Into Components Parallel and Perpendicular to Discontinuity

3.10.1.4 Estimation of Active Length — An estimate of the active length is required to define the assumed local deformation illustrated in Fig. 3-15. It has been shown that the active length extends approximately one to two reinforcing element diameters either side of the discontinuity. In the absence of experimental data, results of theoretical analysis may be used to define the active length. For example, in defining the elastic shear stiffness, κ_s, Gerdeen et al. (1977) also determine a quantity, ℓ, called the load transfer length, or "decay length". If ρ_{max} is the proportion of maximum deflection in the reinforcement, the relation between it and the load transfer length may be expressed by

$$e^{-\beta \ell} = \rho_{max} \tag{45}$$

For example, the point at which the deflection decays to 5% of its maximum value is

$$e^{-\beta \ell} = 0.05$$

or

$$\ell = 3/\beta$$

This approach was developed for reinforcement oriented perpendicular to the shear plane.

Dight (1982) presents a theoretical analysis for determining the distance from the shear plane to maximum moment which corresponds with the location of the plastic hinge in the reinforcement element. This approach places no restrictions on the orientation of the reinforcement with respect to the shear plane. A significant result of this analysis is that the distance of the plastic hinge from the shear plane does not appear to vary greatly with shear displacement, especially for displacements greater than 10mm (0.4 in.) for typical reinforcement systems. This observation is in agreement with the assumed geometry changes described earlier.

3.10.2 <u>A Numerical Formulation for Rock Reinforcement Which Accounts for Inelastic Deformation of Intact Rock and Shear Behavior of the Grout Annulus</u> — In assessing the support provided by rock reinforcement, it is often necessary to consider not only the local restraint provided by reinforcement where it crosses discontinuities but also restraint to intact rock which may experience inelastic deformation in the failed region surrounding an excavation. The model described in the previous section considers only the local restraint provided by the reinforcement at discontinuities. Such a formulation is most applicable to hard rock situations in which the bonding agent (e.g., epoxy resin) is capable of developing very large axial forces in the reinforcement over relatively short distances. However, in many instances, it is necessary to consider more than just the local effect of the reinforcement — its presence in resisting deformation must be accounted for along its entire length. Such situations arise in modeling inelastic deformations associated with failed rock and/or reinforcement systems (e.g., cable bolts) in which the bonding agent (grout) may fail in shear over some length of the reinforcement. The numerical formulation for rock reinforcement which accounts for inelastic deformation of the intact rock and shear behavior of the grout annulus is described here.

3.10.2.1 Axial Behavior — The axial behavior of conventional reinforcement systems may be assumed to be governed entirely by the reinforcing element itself. The reinforcing element is usually steel and may be either a bar or cable. Because the reinforcing element is slender, it offers little bending resistance (particularly in the case of cable), it is treated as a one-dimensional member subject to uniaxial tension or compression. A one-dimensional constitutive model is adequate for describing the axial behavior of the reinforcing element. In the present formulation, the axial stiffness is described in terms of the reinforcement cross-sectional area and Young's modulus, E. At present, the reinforcement has unlimited tensile capacity and no compressive capacity. A more rigorous approach incorporating kinematic hardening is possible with little increase in computational effort.

In evaluating the axial forces developed in the reinforcement, displacements are computed at nodal points along the axis of the reinforcement as shown in Fig. 3-18. Out-of-balance forces at each nodal point are computed from axial forces in the reinforcement as well as shear forces contributed through the grout annulus. Axial displacements are computed based on integration of Newton's Second Law of Motion using the computed out-of-balance axial force and a mass lumped at each nodal point.

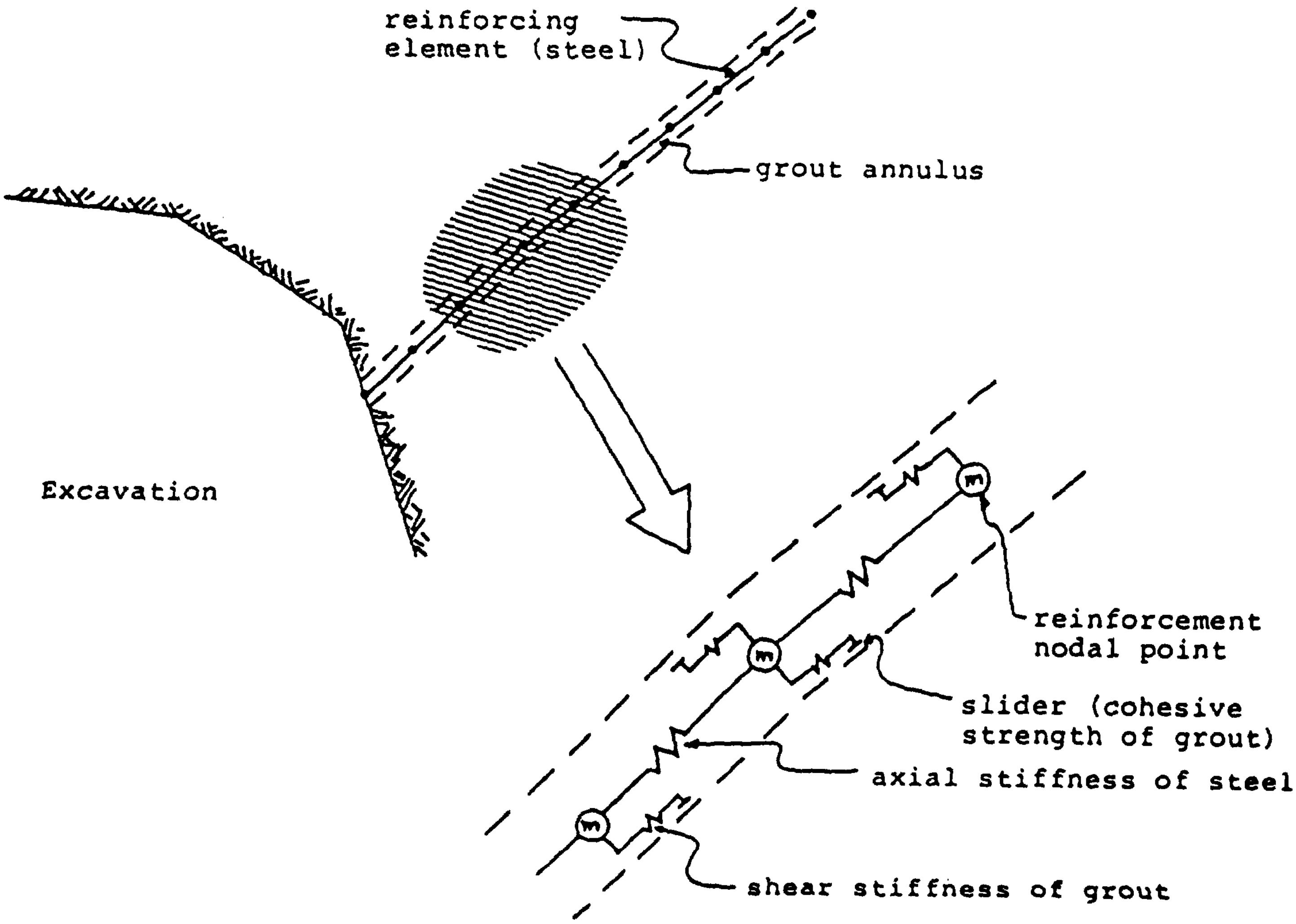


Fig. 3-18 Conceptual Mechanical Representation of Fully-Bonded Reinforcement Which Accounts for Shear Behavior of the Grout Annulus

3.10.2.2 Shear Behavior of Grout Annulus — The shear behavior of the grout annulus is represented at a spring slider system located at nodal points shown in Fig. 3-18. The shear behavior of the grout annulus during relative displacement between the reinforcing/grout interface and the grout/rock interface is described numerically by the grout shear stiffness. Numerical estimates for this parameter can be derived from an equation describing the shear stress at the grout/rock interface (St. John and Van Dillen, 1983):

$$\tau_G = \frac{G_g}{(D/2+t)} \bullet \frac{u_b - u_r}{\ln(1+2t/D)} \tag{46}$$

where u_b = axial displacement of the bolt,

u_r = axial displacement of the grout/rock interface,

G_g = grout shear modulus,

D = bolt diameter, and

t = annulus thickness.

Consequently, the required grout shear stiffness (JKS) per unit problem thickness is simply given by

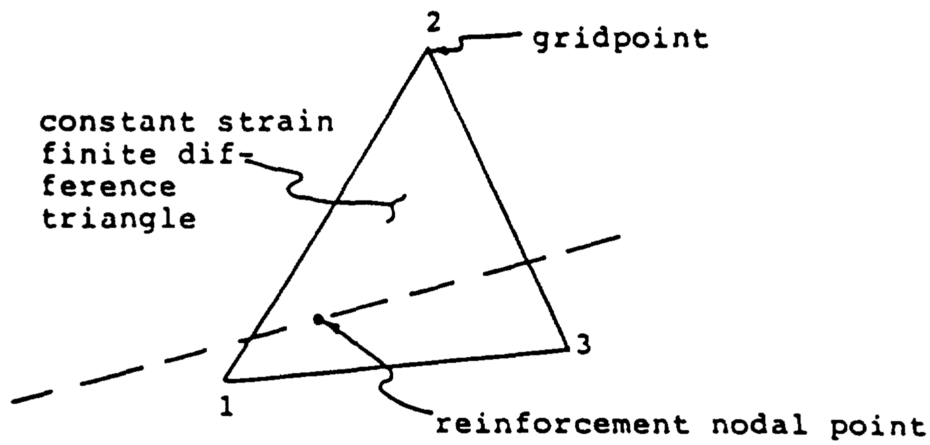
$$JKS = \frac{2\pi\ G_g}{\ln\ (1+2t/D)} \tag{47}$$

In computing the axial displacement of the grout / rock interface, the following interpolation scheme is used. Consider reinforcement passing through a constant strain-finite difference triangle making up part of the intact rock as shown in Fig. 3-19 (a). The incremental x-component of displacement (Δu_{xp}) at the nodal point is given by

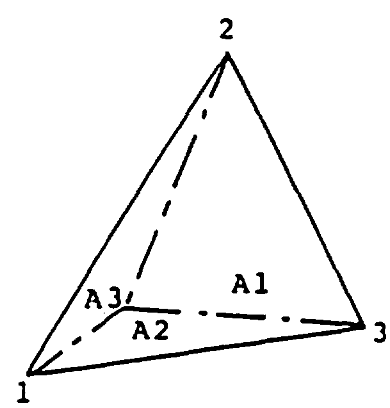
$$\Delta u_{xp} = W_1 \Delta u_{x1} + W_2 \Delta u_{x2} + W_3 \Delta u_{x3} \tag{48}$$

where Δu_{x1}, Δu_{x2}, Δu_{x3} are the incremental gridpoint displacements, and

W_1, W_2, W_3 are weighting factors.



(a) typical reinforcing element passing through a triangular zone



(b) areas used in determining weighting factors used to compute displacement of grout/rock interface

Fig. 3-19 Geometry of Triangular Finite Difference Zone and Transgressing Reinforcement Used in Distinct Element Formulation

A similar expression is used for y-component displacements. The weighting factors W_1, W_2, W_3 are computed from the position of the nodal point within the triangle as follows:

$$W_1 = A_1/A_T \tag{49}$$

where A_T is the total area of the finite-difference triangle, and

A_1 is the area of the triangle in Fig. 3-19(b).

Incremental x- and y-displacements [Eq. (48)] are used at each calculation step to determine the new local reinforcing orientation. The axial component of displacement of the grout/rock interface is computed from the current orientation of the reinforcing segment.

In the present formulation, the maximum amount of shear strength in the grout annulus is limited to a value τ_{peak}. The peak shear strength used may be estimated from the results of pull-out tests or, should such results not be available, the peak strength may be estimated as (St. John and Van Dillen, 1983):

$$\tau_{peak} = \pi \; (D + 2t) \; \tau_I \; Q_B \tag{50}$$

where τ_I is approximately one-half of the uniaxial strength of the weaker of the rock and grout, and

Q_B is the quality of the bond between the grout and rock (Q_B = 1 for perfect bonding).

The maximum shear force (JCOH) per unit problem thickness is, therefore, given by

$$JCOH = \pi \; (D+2t) \; \tau_{peak} \tag{51}$$

Because an explicit formulation is used throughout, more general grout shear constitutive relations, including residual shear strength, for the grout annulus are possible.

Forces generated at the grout/rock interface (F_{xp}, F_{yp}) are distributed back to gridpoints according to the same weighting factors used previously — i.e.,

$$
\begin{aligned}
F_{x1} &= W_1 \bullet F_{xp} \\
F_{x2} &= W_2 \bullet F_{xp} \qquad (52) \\
F_{x3} &= W_3 \bullet F_{xp}
\end{aligned}
$$

where F_{x1}, F_{x2}, and F_{x3} are forces applied to the gridpoints.

3.10.2.3 2-D/3-D Equivalence — Reducing 3-D problems with regularly-spaced reinforcement to 2-D problems involves averaging the reinforcement effect in 3-D over the distance between the bolts. Donovan et al. (1984) suggest that linear scaling of material properties is a simple and convenient way of distributing the discrete effect of reinforcement over the distance between bolts in a regularly-spaced pattern.

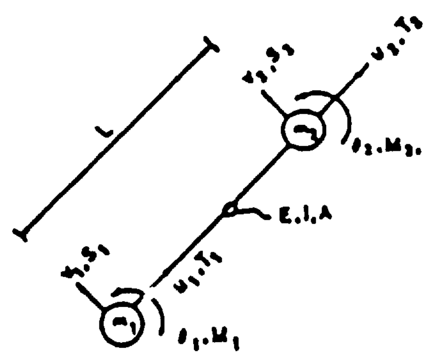
In the present formulation, a scaling factor (f), defined as the ratio of unit problem width (W) to reinforcing spacing (S) perpendicular to the plane of analysis–~~i.e.~~, f = W/S should be used to scale material properties E, G_g, and τ_I. For example, if the reinforcing spacing, S, is 2 meters and the unit problem width is 1 meter, the material properties E, G_g and τ_I should all be reduced by a factor of 2.

3.10.2.4 Failure at Grout/Reinforcing Interface — Failure of reinforcing systems does not always occur at the grout/rock interface. Failure may occur at the reinforcing/grout interface, as is often true for cable reinforcing. In such cases, the shear stress should be evaluated at this interface. This suggests that the expressions $(D+2t)$ be replaced by (D) in Eqs. (50) and (51).

3.11 Tunnel Lining Model

Tunnel linings are typically thin and their characteristic response to bending deformation cannot be neglected. A structural element representation for the tunnel lining provides a convenient method for including bending effects.

The structural element method is well documented in structural engineering texts. The use of beam elements in 2-D linear analysis of excavation support is reported by Dixon (1971), Brierley (1975), Monsees (1977), among others. Paul et al. (1983) presents analysis using beam elements which include non-linear behavior. Analysis of any support structure is initiated by discretization of the structure into a number of elements whose response to axial, transverse and flexural loads can be represented in matrix form, as shown in Fig. 3-20. The rock-structure interface is represented by springs oriented both radially and tangentially with respect to the support structure.



Structural Element Sign Convention

$$
\begin{bmatrix} T_1 \\ S_1 \\ M_1 \\ T_2 \\ S_2 \\ M_2 \end{bmatrix}
= \frac{E}{L}
\begin{bmatrix}
A & & & & & \text{SYM.} \\
0 & \frac{12I}{L^2} & & & & \\
0 & \frac{6I}{L} & 4I & & & \\
-A & 0 & 0 & A & & \\
0 & -\frac{12I}{L^2} & -\frac{6I}{L} & 0 & \frac{12I}{L^2} & \\
0 & -\frac{6I}{L} & 2I & 0 & -\frac{6I}{L} & 4I
\end{bmatrix}
\begin{bmatrix} u_1 \\ v_1 \\ \theta_1 \\ u_2 \\ v_2 \\ \theta_2 \end{bmatrix}
$$

Fig. 3-20 Local Stiffness Matrix for Structural Element Representation of Excavation Support

<u>Structural Element Formulation</u> — Either an implicit or explicit formulation may be used in analyzing the behavior of a support structure composed of beam elements and interface stiffnesses. In the first formulation (implicit), a global stiffness matrix is formed for the entire structure. The size of the stiffness matrix is reduced by deleting free nodes—i.e., those nodes which are not located at the rock-support interface. This is possible because these nodes are subjected to neither directly-imposed external loads nor displacements by the surrounding medium. The resultant efficiency, however, limits straightforward application of this formulation to quasi-static problems involving linear elastic behavior. This formulation does not provide information about failure mechanisms or ultimate capacities of interior supports. However, factors of safety based on lining stresses should be conservative since they do not take into account the highly indeterminate nature of a lining in contact with the rock. A detailed description of this formulation, its use with distinct elements, and numerous other examples are presented by Lorig (1984).

In the second formulation (explicit), local stiffness matrices are used following division of the structure into segments with the distributed mass of the structure "lumped" at nodal points, as shown in Fig. 3-21. Forces generated in support elements are applied to the lumped masses which move in response to unbalanced forces and moments in accordance with the equations of motion. This formulation has the following desirable characteristics: (1) slip between support and excavation periphery is modeled in a manner identical to block interaction along a discontinuity and (2) large displacements, with non-linear material behavior are readily accommodated. These capabilities are illustrated in Fig. 3-22, where a roof block loads and displaces a hypothetical four-element interior structural support.

The present version of UDEC uses the second (i.e., explicit structural element formulation) to model a support structure.

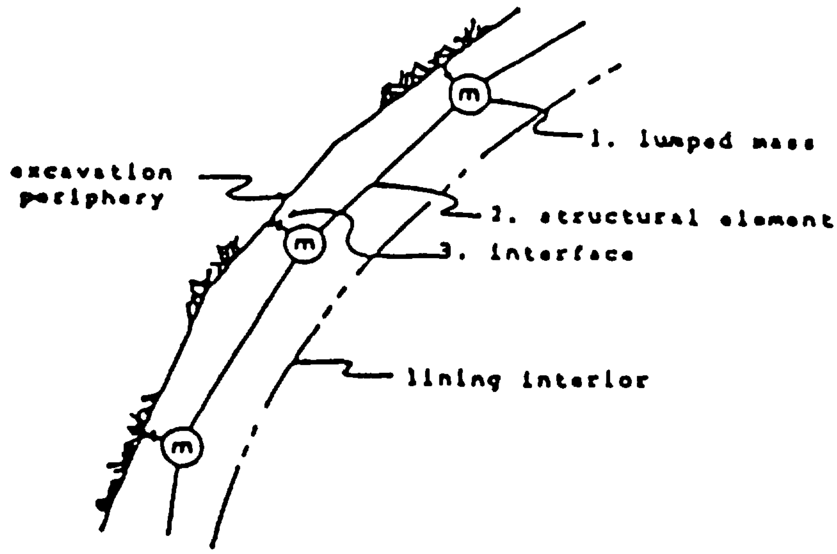


Fig. 3-21 Lumped Mass Representation of Structure Used in Explicit Formulation

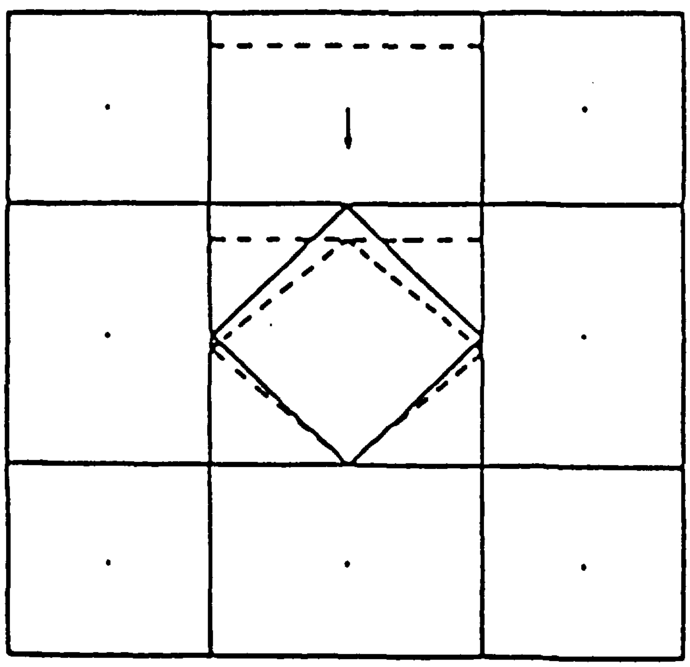


Fig. 3-22 Demonstration of Interface Slip and Large Displacement Capabilities of Explicit Structural Element Formulation

3.12 Fluid Flow in Joints

The code UDEC includes the capability to perform the analysis of fluid flow through the fractures of a system of impermeable blocks. A fully-coupled mechanical-hydraulic analysis is performed, where fracture conductivity is dependent on mechanical deformation and, conversely, joint water pressures affect the mechanical computations.

The numerical implementation makes use of the domain structure shown in Fig. 3-3(e). For a closely- packed system, there exists a network of domains, each of which is assumed to be filled with fluid at uniform pressure and which communicates with its neighbors through contacts. Fully-saturated conditions are assumed. When gravitational loading is present, the effects of the hydrostatic pressure are simulated by applying buoyancy forces to the blocks. In this case, domain pressures equal the total water pressure minus the hydrostatic pressure.

When a pressure differential exists between adjacent domains, flow will take place. The flow rate is calculated in two different ways depending on the type of contact. For a point contact (i.e., corner-edge or corner-corner), the flow rate is

$$q = - k_c \Delta p \tag{53}$$

where k_c is a contact permeability parameter.

When the contact possesses a length (i.e., an edge-edge contact), a more realistic calculation is possible, based on the cubic law of flow in fractures. The flow rate is then given by

$$q = - k_j a^3 \Delta p / l \tag{54}$$

where k_j is a joint permeability factor,

a is the contact hydraulic aperture, and

l is the contact length.

The hydraulic aperture is given, in general, by

$$a = a_o + \Delta a \tag{55}$$

where a_o is the aperture at zero normal stress, and

Δa is the mechanical change in aperture due to the contact normal displacement.

A minimum value, a_{res}, is assumed for the aperture, beyond which mechanical closure does not affect the contact permeability.

At every time step, flow rates through all the contacts are calculated based on these formulas. Then, domain pressures are updated, taking into account the net flow into the domain and possible changes in domain area due to the motion of the surrounding blocks. The new domain pressure is

$$p = p_o + K_w \, Q \, \Delta t \, / \, A - K_w \, \Delta A \, / \, A_m \tag{56}$$

where p_o is the old domain pressure,

Q is the sum of flow rates into the domain from all surrounding contacts,

K_w is the bulk modulus of fluid, and

$\Delta A = A - A_o \quad , \quad A_m = (A + A_o) \, / \, 2.$

where A and A_o are, respectively, the new and old domain areas.

The new domain pressures are then added to the forces to be applied to the surrounding block edges. This process results in effective normal stresses in all contacts and total stresses inside the impermeable blocks.

The numerical stability of the fluid flow algorithm, for slowly varying domain areas, requires that the timestep be limited to

$$\Delta t^f = \min \left[A \,/\, (K_w \, \Sigma \, k_i) \right] \tag{57}$$

where the sum of k_i is extended to all contacts surrounding the domain, with

$$k_i = \max (k_c \,,\, k_j \, a^3 / l)$$

The minimum value of Δt^f for any domain is taken and, if less than the mechanical timestep, is used in the analysis.

For joint contacts (edge-edge), the domain area is the product of joint length and aperture. Because the minimum joint aperture is a_{res}, the domain area is always positive, even if the blocks are overlapping. A minimum domain area can be set for computational efficiency. Large contact permeabilities or very small domain areas require very small timesteps. In addition, the fluid filling a joint increases the apparent stiffness by K_w/a. If a is small, this may be much larger than the joint mechanical stiffness and require the adjustment of the timestep. There are several ways of reducing the run times of fluid flow analyses. The use of density scaling of the blocks or the scaling of the bulk modulus of the fluid can improve efficiency by producing mechanical and hydraulic time steps of the same order of magnitude. In many cases, only the final steady-state condition is of interest. This condition does not depend on domain areas, and an equal area can be used for all domains, which leads to faster convergence. This scheme also eliminates the problem of the fluid stiffness affecting the mechanical time step.

Problems containing joints of very different apertures generally require a large number of timesteps for the steady-state condition to be reached. In many cases, the domain pressures change very slowly, and their mechanical effects are only felt after several timesteps. A procedure to be used in such cases was developed in which several fluid flow timesteps are executed between mechanical timesteps. It is based on an adaptive procedure which 'triggers' an update of the mechanical quantities whenever the maximum increment of pressure in any domain exceeds some prescribed tolerance.

Typical SI units for various parameters in this section are given in Table 3-1.

Table 3-1

TYPICAL SI UNITS FOR FLUID FLOW PARAMETERS

Parameter	SI Units
q	$m^3/sec \bullet m$
p	Pa
k_c	$m^2/sec \bullet Pa$
k_j	$(1/Pa) \bullet sec$
a	m
l	m
Q	m^3/sec
K_w	Pa

3.12.1 Hydraulic Behavior of Rock Joints — Flow in planar rock fractures may be idealized by means of the parallel plate model. The analytic solution for laminar viscous flow between parallel plates gives the mean velocity as

$$v = k_f \, J \tag{58}$$

where J is the hydraulic gradient, and the fracture hydraulic conductivity is given by

$$k_f = \frac{a^2 \, g}{12\nu} \tag{59}$$

where a is the fracture width,

ν is the kinematic viscosity of the fluid, and

g is the acceleration of gravity.

The flow rate per unit width can thus be expressed as

$$q = v\ e = \frac{a^3\ g}{12\ \nu}\ J \tag{60}$$

which is usually referred as the cubic flow law.

Experiments conducted by Louis (1969) showed that this law is essentially valid for laminar flow in rock joints. This author proposed an empirical correction factor for the above expression in order to account for fracture roughness. Witherspoon et al. (1979) tested both open and closed joints. They reported that the cubic law is still valid for the latter, provided that the actual mechanical aperture is used. Due to the effects of roughness and tortuosity of flow, the fracture conductivity in their experiments was reduced by a factor between 1.04 and 1.65. Barton et al. (1985) proposed an empirical formula which gives the hydraulic aperture (to be used in the cubic law) as a function of the mechanical aperture and the joint roughness coefficient (JRC).

3.12.2 Example Fluid Flow Parameter Values — The following discussion relates primarily to the gravity dam example problems presented in the appendix, but it can serve as a starting point for other problems as well. In the example, joint hydraulic apertures were calculated from Eq. (55).

A residual aperture a_{res} was taken as the minimum hydraulic aperture. A maximum value for the hydraulic aperture equal to 2 to 4 times the residual aperture was assumed for reasons of computational efficiency, since the time-step required for stability of the fluid flow algorithm is inversely proportional to joint conductivity. As conductivity is proportional to the cube of the aperture, considerable variation of permeability due to stress changes can still be modeled in spite of this constraint. In the base run, the following values were used

$$a_0 = 1 \text{ mm}$$

$$a_{res} = 0.5 \text{ mm}$$

The water properties were

$$K_w = 2000 \text{ MPa}$$

$$\rho_w = 1000 \text{ kg/m}^3$$

$$\mu = 2.8 \times 10^{-4} \text{ Pa}\bullet\text{sec}$$

The dynamic viscosity μ ($\mu = \nu \rho_w$) was used to calculate the joint permeability factor in UDEC (see Eq. 54):

$$k_j = 1 / 12\mu = 3 \times 10^8 \text{ MPa}^{-1}\text{sec}^{-1} \tag{61}$$

3.13 Visco-Plastic Flow in Joints

The flow of a Bingham body (or liquid) such as cement grout is of the visco-plastic type. The major difference between this model and a Newtonian liquid is that, for a Bingham fluid, a yield stress, τ_y, must be exceeded to initiate flow.

For Newtonian flow, it is assumed that the flow rate per unit width, q , is related linearly to the pressure gradient, J, as shown in Fig. 3-23. The general equation for fluid flow between planar surfaces is given by

$$q = \frac{b e^a J}{12\mu} \tag{62}$$

where e = fracture width (aperture),

b = empirical coefficient,

μ = dynamic viscosity of fluid, and

a = aperture exponent.

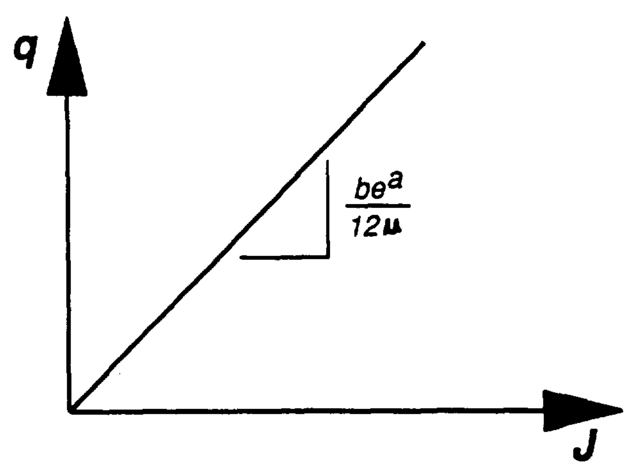


Fig. 3-23 Flow-Gradient Relation for Newtonian Fluid in UDEC

In the most widely used form of this relation, known as the cubic flow law, a = 3 and b = 1. The flow gradient relation of a Bingham body is similar, except that no flow occurs until the threshold gradient, J_o, is exceeded as shown in Fig. 3-24.

Considering the balance of forces on a rectangular element of fluid, the expression for the threshold gradient for flow between parallel sides of aperture, e, is given by

$$J = \frac{2\tau_y}{e} \tag{63}$$

The expression for the threshold gradient can also be obtained by considering the equation for steady laminar flow of a Bingham plastic in a circular pipe. This equation is known as Buckingham's equation (Wilkinson, 1960):

$$Q = \frac{\pi a^4 \Delta P}{8L \mu_p}\left[1 - \frac{4}{3}\left[\frac{2L\tau_y}{a \Delta P}\right] + \frac{1}{3}\left[\frac{2L\tau_y}{a \Delta P}\right]^4\right]$$

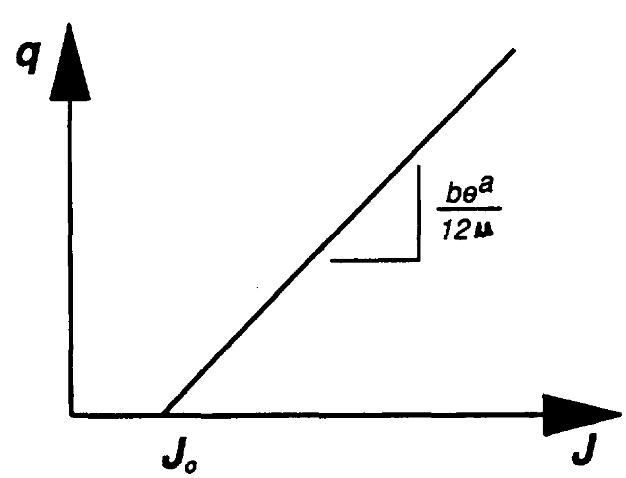


Fig. 3-24 Flow-Gradient Relation for Bingham Fluid in UDEC

where Q = volume rate of flow,

a = pipe radius,

μ_p = Bingham plastic viscosity,

$\Delta P/L$ = pressure gradient = J, and

τ_y = yield stress.

From this expression, it can be seen that no flow occurs if the pressure gradient J is zero or equals $2\tau_y/a$. It is not clear from the equation what occurs at pressure gradients between zero and $2\tau_y/a$, but it is reasonable to assume that no steady flow occurs within this range. Therefore, the threshold gradient, J_o, the gradient at which steady flow is possible, is given by

$$J_o = \frac{2\tau_y}{a} \tag{64}$$

Note that this expression can also be derived by considering the balance of forces acting on a cylindrical element of fluid with radius a and length L.

3.14 Strength/Stress Contour Plots

UDEC allows the user to make plots of strength/stress ratios for fully-deformable blocks. Two strength criteria are available: Mohr-Coulomb and Hoek-Brown.

3.14.1 Mohr-Coulomb Strength Criteria — The state of stress within any zone can be expressed in terms of principal stresses σ_1 and σ_2. This stress state, in general, will plot as a circle "a" with radius r_a on the Mohr diagram. Failure occurs if this circle just touches the failure envelope. The strength for the stress state represented by circle "a" is determined by holding σ_2 constant while increasing or decreasing σ_1 until circle "b" with radius r_b touches the envelope. The ratio of the radii of the two circles ($F=r_b/r_a$) is the strength/stress ratio. F is also known as the "failure index" or the "factor of safety". Note that $|F|<1$ for all circles "a" with points lying outside the envelope. UDEC gives F as a negative number whenever

σ_2 is tensile (i.e., σ_2 is positive) to identify the presence of tensile stresses. UDEC gives F as zero whenever σ_2 is greater than the tension cut-off (recall that tension is positive). The tension cut-off is taken to be the smaller of TENS or COH/FRIC (i.e., C/tanϕ) for each material specified.

3.14.2 Hoek-Brown Strength Criterion — The Hoek-Brown empirical failure criterion (Hoek and Brown, 1970) is given (for compressive stresses negative) by:

$$\sigma_1 = \sigma_2 - (-m\sigma_c\sigma_2 + s\sigma_c^2)^{1/2} \tag{65}$$

where σ_1 = major principal stress,

σ_2 = minor principal stress,

m = empirical value $(m>0)$,

s = empirical value $(0<s<1.0)$, and

σ_c = unconfined compressive strength $(\sigma_c>0)$.

The state of stress within any zone can be expressed in terms of principal stresses σ_1 and σ_2. This stress state, in general, will plot as a circle "a" with radius r_a on the Mohr diagram. Failure occurs if this circle just touches the failure envelope. The strength for the stress state represented by circle "a" is determined by holding σ_2 constant while increasing or decreasing σ_1 until circle "b" with radius r_b touches the envelope. The ratio of the radii of the two circles $(F=r_b/r_a)$ is the strength/ stress ratio. F is also known as the "failure index" or the "factor of safety". Note that $|F|<1$ for all circles "a" with points lying outside the envelope. UDEC gives F as a negative number whenever σ_2 is tensile (i.e., σ_2 is positive) to identify the presence of tensile stresses. F is zero whenever σ_2 is greater than $(s\bullet\sigma_c/m)$.

3.15 References

Azuar, Debreuille, Habib, Londe, Panet, Rochet, and Salembier. "Rock Mass Reinforcement by Passive Rebars," Proc. 4th International Congress on Rock Mechanics, Vol. 1, pp. 23-30. 1974.

Bandis, S. C., A. C. Lumsden, and N.R. Barton. "Fundamentals of Rock Joint Deformation," Int. J. Rock Mech. Min. Sci., 20, 244-268 (1983).

Barton, N., S. Bandis and K. Bakhtar. "Strength, Deformation and Conductivity Coupling of Rock Joints," Int. J. Rock Mech. Min. Sci., 22, 121-140 (1985).

Bjurstrom, S. "Shear Strength on Hard Rock Joints Reinforced by Grouted Untensioned Bolts," Proc. 3rd International Congress on Rock Mechanics, Vol 2, pp. 1194-1199. 1979

Brady, B.H.G., and A. Wassyng. "A Coupled Finite Element-Boundary Element Method of Stress Analysis," Int. J. Rock Mech. Min. Sci., 18, 475-485 (1981).

Brierley, G. "The Performance During Construction of the Liner of a Large, Shallow Underground Opening in Rock," Ph.D. Thesis, University of Illinois at Urbana-Champaign, 1975.

Cundall, P.A. "A Computer Model for Simulating Progressive Large Scale Movements in Blocky Rock Systems," Proceedings of the Symposium of the International Society of Rock Mechanics (Nancy, France, 1971), Vol. 1, Paper No. II-8.

Cundall, P. A. "UDEC - A Generalized Distinct Element Program for Modelling Jointed Rock," Report PCAR-1-80, Peter Cundall Associates; Contract DAJA37-79-C-0548, European Research Office, U.S. Army, March 1980.

Cundall, Peter A., and Roger D. Hart. "Analysis of Block Test No. 1 Inelastic Rock Mass Behavior: Phase 2 - A Characterization of Joint Behavior (Final Report)," Itasca Consulting Group, Subcontract SA-957, Rockwell Hanford Operations, Richland, Washington, March 1984.

Cundall, Peter A., and Roger D. Hart. "Development of Generalized 2-D and 3-D Distinct Element Programs for Modeling Jointed Rock." Itasca Consulting Group; Misc. Paper SL-85-1, U.S. Army Corps of Engineers, 1985.

Cundall, P. A., J. Marti, P. J. Beresford, N. C. Last, and M. I. Asgian. "Computer Modeling of Jointed Rock Masses," Tech. Report N-78-4 U. S. Army Engineer Waterways Experiment Station, Vicksburg, Mississippi, August 1978.

Cundall, P. A., H. Hansteen, S. Lacasse, and P. B. Selnes. "NESSI -Soil Structure Interaction Program for Dynamic and Static Problems," Report 51508-9, Norwegian Geotechnical Institute December 1980.

Dight, P. M. "Improvements to the Stability of Rock Walls in Open Pit Mines," Ph.D. Thesis, Monash University, 1982.

Dixon, J. D. "Analysis of Tunnel Support Structure with Consideration of Support-Rock Interaction," U.S. Dept. of Interior, Bureau of Mines Investigation RI7297, 1971.

Donovan, K., W. E. Pariseau, and M. Cepak. "Finite Element Approach to Cable Bolting in Steeply Dipping VCR Stopes," in Geomechanics Applications in Underground Hardrock Mining. New York: AIME, 1984.

Fuller, P.G., and R.H.T. Cox. "Rock Reinforcement Design Based on Control of Joint Displacement–A New Concept," Proc. 3rd Aust. Tunnelling Conference (Sydney, 1978), pp. 28-35. 1978.

Gerdeen, J. C., V. W. Snyder, G. L. Viegelahn, and J. Parker. "Design Criteria for Roof Bolting Plans Using Fully Resin-Grouted Nontensioned Bolts to Reinforce Bedded Mine Roof," U.S. Bureau of Mines OFR 46(4)-80, 1977.

Haas, C. J. "Shear Resistance of Rock Bolts, Trans. Soc. Min. Eng. AIME, 260(1), 32-41 (1976).

Hoek, E., and E. T. Brown. Underground Excavations in Rock. London: The Inst. of Mining and Metallurgy, 1980.

Kunar, R. R., P. J. Beresford, and P. A. Cundall. "A Tested Soil-Structure Model for Surface Structures," Proceedings of the Symposium on Soil-Structure Interaction (Roorkee University, India, January 1976), Vol 1, pp. 137-144. Meerut, India: Sarita Prakashan, 1976.

Lemos, José. "A Distinct Element Model for Dynamic Analysis of Jointed Rock with Application to Dam Foundations and Fault Motion," Ph.D. Thesis, University of Minnesota, June 1987.

Lemos, J. V. "A Hybrid Distinct Element-Boundary Element Computational Model for the Half-Plane," M.S. Thesis, University of Minnesota, 1983.

Littlejohn, G. S., and D. A. Bruce. "Rock Anchors—State of the Art. Part I: Design," Ground Engineering, 25-32 (May 1975), 41-48 (July 1975).

Lorig, L. J. "A Hybrid Computational Model for Excavation and Support Design in Jointed Media," Ph.D. Thesis, University of Minnesota, 1984.

Lorig, L. J. "A Simple Numerical Representation of Fully Bonded Passive Rock Reinforcement for Hard Rocks." Computers and Geotechnics, 1, 79-97 (1985).

Louis, C. "A Study of Groundwater Flow in Jointed Rock and Its Influence on the Stability of Rock Masses," Imperial College Rock Mech. Research Report No. 10, 1969.

Lysmer, J., and R. Kuhlemeyer. "Finite Dynamic Model for Infinite Media," J. Eng. Mech. Div. ASCE 95:EM4 (1969).

Lysmer, J., and G. Waas. "Shear Waves in Plane Infinite Structures," J. Eng. Mech. Div. ASCE, 98, 85-105 (1972).

Monsees, J. E. "Station Design for the Washington Metro System," Proceedings of the Engineering Foundation Conference—Shotcrete Support, ACI publication SP-54, 1977.

Paul, S. L., A. J. Hendron, E. J. Cording, G. E. Sgouros, and P. K. Saha. "Design Recommendations for Concrete Tunnel Linings," University of Illinois DOT Report No. DOT-TSC-UMTA-83-16, 1983.

Pells, P.J.N. "The Behaviour of Fully Bonded Rock Bolts," Proc. 3rd International Congress on Rock Mechanics, Vol. 2, pp. 1212-1217. 1974.

Roesset, J. M., and M. M. Ettouney. "Transmitting Boundaries : A Comparison," Int. J. Num. Anal. Meth. Geomech., 1, 151-176 (1977).

Seed, H. B., J. Lysmer, and R. Hwang. "Soil-Structure Interaction Analysis for Seismic Response," J. Geotech., Div. ASCE, 101, 439-457 (1975).

St. John, C. M., and D. E. Van Dillen. "Rockbolts: A New Numerical Representation and Its Application in Tunnel Design," Rock Mechanics —Theory - Experiment - Practice (Proceedings of the 24th U.S. Symposium on Rock Mechanics (Texas A&M University, June 1983), pp. 13-26. New York: Association of Engineering Geologists, 1983.

White, W., S. Valliappan, and I. K. Lee. "Unified Boundary for Finite Dynamic Models," J. Eng. Mech., Div. ASCE, 103, 949-964 (1977).

Wilkinson, W. L. Non-Newtonian Fluids: Fluid Mechanics, Mixing and Heat Transfer. London: Pergamon Press, 1960.

Witherspoon, P. A., J.S.Y. Wang, K. Iawi, and J. E. Gale. "Validation of Cubic Law for Fluid Flow in a Deformable Rock Fracture," Lawrence Berkeley Laboratory, University of California, Berkeley, 1979.

Wolf, J. P. Dynamic Soil-Structure Interaction. New Jersey: Prentice-Hall (1985).

4.0 INPUT INSTRUCTIONS AND COMMAND DESCRIPTIONS

4.1 Program Input Conventions

The input commands to UDEC differ from most computer programs written for numerical modeling; commands are specifically designed for simplicity and ease of use by the program operator. All input commands are word-oriented and consist of a primary command word followed by keywords and numerical input, as required. The commands, given on the following pages, are typed literally on the input line. You will note that only the first few letters are capitalized. The program requires only these letters to be typed for the command to be "recognized". Many of the keywords are followed by a series of numbers which provide the numeric input required by the keyword. Words that begin with a lower-case letter stand for numbers. Integers are expected when the word begins with i, j, k, l, m or n; otherwise, a real (or decimal) number is expected. The decimal point can be omitted from a real number but must not appear in an integer. The keywords and numbers may be separated by any number of spaces or by any of the following delimiters:

( ) , / =

< > denotes optional parameter(s) — the brackets are NOT to be typed.

Anything that follows a "*" is taken to be a comment and is ignored. It is useful to make such comments in the input file since the comments are reproduced on the output.

An ampersand ("&") at the end of a line denotes the next line as a continuation of keywords or numeric input. In previous versions of UDEC, a "+" at the beginning of a line denoted that line as a continuation of keywords or numeric input for the preceding line. This option is maintained in the present version of UDEC for compatibility with existing data files. However, the "+' continuation convention will not work with some commands, such as the **BLOCK** command.

Geometric locations are prescribed for some commands by coordinate points (i.e., x1,y1 x2,y2) or by rectangular regions or ranges (i.e., xl,xu yl,yu). When ranges are specified (see, for example, **CHANGE**, the x-range delimiters (xl and xu) or the y-range delimiters (yl and yu) cannot be equal. For example, the command (> **CHANGE 10,10 1,10 KEYWORD**) will not work because the x-range is not correctly specified.

The following sign conventions are used in UDEC and must be kept in mind when entering input. The sign is positive for:

motion upwards and to the right
tensile stress and force
extensional strain
shear strain as follows:

```
                      *  *  *  *
                    *        *
                  *        *
                *  *  *  *
```

joint normal force (stress) in compression
joint normal opening
joint shear displacement as follows:

```
                                  -->
                                  <--
```

joint shear force (stress) as follows:

```
                                  <--
                                  -->
```

Anti-clockwise block rotations and moments are positive. The sign for a force or velocity boundary condition is determined by the direction of the force or velocity vector (i.e., + when pointing in the positive axis direction).

SI units are recommended for the numeric input; however, any consistent set of engineering units may be used. No conversions are performed in the program. Table 4-1 illustrates examples of consistent sets of units.

The commands to UDEC are listed in two ways to assist the user in preparing code input. First, in Sections 4.2 to 4.4, a summary of commands is given according to the modeling function performed and the order in which the commands should be entered. This will help the user identify model conditions required for a specific analysis. Following this, full descriptions of all commands are given in alphabetical order. Section 4.5 describes all commands for problem execution; Section 4.6 describes all plotting commands.

Within certain limits, the commands are order independent. The **STOP** command should be the last command in a batch DATA file.

Table 4-1

SYSTEMS OF UNITS

	METRIC				BRITISH	
Length	m	m	m	cm	ft	in
Density	kg/m^3	10^3kg/m^3	10^6kg/m^3	10^6g/cm^3	slugs/ft^3	snails/in^3
Force	N	kN	MN	Mdynes	lb_f	lb_f
Stress	Pa	kPa	MPa	bar	lb_f/ft^2	psi
Gravity	m/sec^2	m/sec^2	m/sec^2	cm/sec^2	ft/sec^2	in/sec^2
Stiffness	Pa/m	KPa/m	MPa/m	bar/cm	lb_f/ft^3	lb/in^3

where 1 bar = 10^6 dynes/cm^2 = 10^5 N/m^2 = 10^5 Pa

1 atm = 1.013 bars = 14.7 psi = 2116 lb_f/ft^2 = 9.807×10^4 Pa

1 slug = 1 lb_f-s^2/ft = 14.59 kg

1 snail = 1 lb_f-s^2/in

1 gravity = 9.81 m/s^2 = 981 cm/s^2 = 32.17 ft/s^2

4.2 General Command Sequence and Simple Command Descriptions

Commands may, in general, be given in any logical order. The following command sequence is recommended. Only the most frequently used commands are presented in this section. More detailed descriptions of all commands are given in the next subsection.

I. SPECIFY PROGRAM CONTROL

The user has options to:

- **START** a new problem
- **RESTART** an existing saved state from a previously-executed problem
- **CALL** a previously-prepared remote input command file to read into UDEC
- **RETURN** program control from a remote input file to local mode
- **STOP** program execution

The command **THERMAL** must be given before the **BLOCK** command for any problem involving heat flow or thermally-induced stresses.

II. INPUT PROBLEM GEOMETRY

The general procedure to establish the problem geometry is to:

(1) specify a constant distance for corner rounding using the **ROUND** command (The default is 0.5.);

(2) create a single rigid block defining original boundary of modeled region using the **BLOCK** command; and

(3) divide this original block into smaller blocks to create the desired assemblage of blocks.

All possible future blocks must be created during this step as, once execution begins, blocks can no longer be created. Blocks may be deleted at any time.

Partially-penetrating cracks can be used while creating the problem geometry, but UDEC can only analyze assemblages of blocks. Therefore, upon execution of the first **CYCLE** command or upon internally discretizing any blocks to form fully-deformable blocks, UDEC will delete any joints which do not completely intersect a block.

Care must be taken when entering cracks and splits since no facility exists for deleting a crack once it has been formed. In addition, once rigid blocks are changed to fully-deformable blocks, they can no longer be cracked or split.

The following commands are used to establish geometries of block assemblage.

ARC creates a series of cracks in an arc pattern.

CRACK creates single crack in block.

DELETE deletes block(s).

EDGE specifies minimum value of block edge length.

II. INPUT PROBLEM GEOMETRY (continued)

JDELETE deletes joints which do not completely intersect in block.

JREGION defines region for automatic joint set generation.

JSET generates a joint set from statistically-based parameters.

SPLIT splits a block into two.

TUNNEL generates tunnel geometry by introducing a circular pattern of cracks.

III. SPECIFY BLOCK DEFORMABILITY

All blocks are rigid by default. Blocks are made "simply deformable" (SDEF) by using the **CHANGE** command. Blocks can be made "fully deformable" (FDEF) by using the **GENERATE** command. The **GENERATE** command discretizes specified blocks into constant strain finite difference triangular zones.

IV. SPECIFY ARTIFICIAL PASSIVE SUPPORT

Three types of artificial passive support may be specified:

(1) **STRUCTURE**, which generates structural (i.e., beam) elements modeling for tunnel lining;

(2) **REINFORCE**, which generates local reinforcement elements across joints for modeling fully-bonded passive reinforcement in hard rocks; and

(3) **CABLE**, which generates global reinforcing elements across joints and within fully-deformable blocks for modeling fully-bonded passive reinforcement which explicitly considers the shear behavior of the grout annulus.

V. ASSIGN CONSTITUTIVE RELATIONS AND PROPERTIES

Block and joint constitutive relations are specified using the **CHANGE** command. All block, joint and artificial support properties are designated using the **PROPERTY** command. Fluid properties are specified with the **FLUID** command.

VI. APPLY BOUNDARY AND INITIAL CONDITIONS

General boundary conditions are prescribed using the **BOUNDARY** command. The following commands are also used to specify initial and boundary conditions.

FFIELD	prescribes dynamic free field.
FIX	current velocities for specified blocks are fixed.
FREE	revokes the action of the **FIX** command.
GRAVITY	sets gravitational acceleration in x- and y-directions.
INSITU	sets initial zone stresses in all fully-deformable blocks and joints.
LOAD	prescribes loads at block centroids.
PFIX	prescribes pressure to domain regions.
PFREE	revokes actions of PFIX command.
RESET	resets certain variables to zero.

The **BE** command defines conditions for the boundary element model.

VII. SPECIFY CONDITIONS DURING SOLUTION

The following commands are used to specify conditions during solution.

DAMPING material damping (used to absorb kinetic energy)

FRACTION fraction of critical time-step used in solution

MSCALE sets zone and block masses for mass-scaling.

SET sets certain internal variables.

In addition, it is often useful to monitor certain variables through time by using the **HISTORY** command.

VIII. CYCLE THE PROBLEM

The **CYCLE** command is used to execute a given number of computation cycles.

IX. OUTPUT

The **PRINT** and **PLOT** commands may be utilized to examine the current problem state. Other output commands are listed below.

COPY copies a plot to a hard-copy device.

DUMP dumps memory from main array to screen.

HEADING prescribes heading for problem.

JPLOT generates joint plots on screen.

SAVE saves the current problem state in a remote file.

SET sets plotting parameters.

WINDOW defines model region to be plotted.

4.3 Setting Your Own Default Conditions ("UDEC.INI")

If you wish UDEC to assume certain parameters or output modes whenever you start the program, prepare a file called "UDEC.INI". This file may contain any valid UDEC commands(s). UDEC will read the file automatically on start-up and process the commands. For example, "UDEC.INI" might contain the following:

SET PLOT=PS

GRAVITY 0 -10

If the file "UDEC.INI" does not exist, UDEC simply continues without error.

4.4 Defining a Batch Mode DATA File

If you wish to run UDEC in the batch mode, you need to create a file called "DATA". This file may contain any valid UDEC commands except **CALL** and **RETURN**. The last command in the "DATA" file must be **STOP**. If the file "DATA" does not exist, UDEC will operate interactively.

4.5 Full Description of Input Commands for Problem Execution

Arc xc yc xb yb theta npoin

This command creates a pattern of npoin cracks which conform to an arc of a circle centered at (xc, yc), with a beginning point of (xb,yb) and a counterclockwise angle of theta degrees.

BE keyword <keyword>

Boundary element conditions are applied to the boundary domain generated by the keyword GEN. The keywords and associated parameters are:

Fix	xh yh xv yv	fix point (xh, yh) in x-direction and point (xv, yv) in y-direction, outside of model region (must precede STIF keyword)
Half		half-plane solution for boundary element domain, assuming surface at y=0 (must precede STIF keyword)
Gen	<x1 x2 y1 y2> <Corner ic1 ic2>	generate boundary element domain along outer boundary between region x1 to x2 and y1 to y2 or between boundary corners ic1 and ic2 (the affected boundary runs clockwise from ic1 to ic2)
Mat	n	material number n assigned to far-field properties
Stif		automatically generate stiffness matrix

BE (continued)

It is recommended that the **BE** commands be applied after the model comes to equilibrium under in-situ loading conditions (use **BOUNDARY STRESS** command, for example, to apply far-field stresses initially). The recommended order of the **BE** commands is:

BE GEN
BE MAT
BE HALF (if half plane solution is used)
BE FIX
BE STIFF

Note: Contacts at the boundary are assigned material properties according to the **BE MAT** command. Use the **CHANGE** command to set different material conditions at the boundary.

Block <Material n> <Constitutive m> x1 y1 x2 y2 ...
<Circular xc yc r n>

Create a rigid block of material number n and constitutive number m. Defaults are n = 1, m = 1, if m, n are omitted. Corner coordinates are: (x1,y1),(x2,y2), etc. in a clockwise direction. Continuation lines may be used but a pair of numbers defining a corner must not be separated. A circular block is created with the **CIRCULAR** keyword where (xc, yc) are the coordinates of the center of the circular block, r is the radius, and n is the number of edges along the outer boundary. Only one **BLOCK** command may be used per run. Further blocks may be created with a **CRACK, SPLIT, TUNNEL, ARC** or **JSET** command and unwanted ones deleted with the **DELETE** command. Any block may be changed to simply deformable (SDEF) with a **CHANGE** command or to fully deformable (FDEF) using the **GENERATE** command. Note: At present the code will not accept some commands if only one block is present.

BOundary <xl xu yl yu> keyword <keyword> . . .
<Corner ic1 ic2>

Boundary conditions are specified over a region $xl<x<xu$, $yl<y<yu$ or between boundary corners ic1 and ic2 (the affected boundary runs clockwise from block corner ic1 to ic2). [NOTE: corner addresses can be found from the **PRINT BOUND STATE** command after the first **BOUNDARY** command has been executed. Corner addresses are listed under the column heading COR.]. If the range is omitted, the boundary condition applies to the entire boundary. Boundary conditions are defined by a given boundary condition keyword. The keywords and associated parameters are:

<u>Load Boundary</u>

XLoad	fx	x-direction load (see note)
YLoad	fy	y-direction load (see note)

<u>Free Boundary</u>

XFRee		removes boundary condition in x-direction
YFRee		removes boundary condition in y-direction

Boundary (continued)

Stress Boundary

STress sxxo sxyo syyo

boundary stress parameters: xx stress, xy stress, yy stress (see note)

XGrad sxxx sxyx syyx
YGrad sxxy sxyy syyy

linearly-varying boundary stress where sxxo,sxyo and syyo are stresses at origin (0,0) and where

```
sxx = sxxo + (sxxx  •  x) + (sxxy  •  y)
sxy = sxyo + (sxyx  •  x) + (sxyy  •  y)
syy = syyo + (syyx  •  x) + (syyy  •  y)
```

(see note)

Note: All loads and stresses specified after the end of a **CYCLE** command are added into an incremental force to be applied during the next **CYCLE** command according to the type of history specified. At the end of this **CYCLE** command, the incremental vectors are added to the total vector and the incremental vector is set to zero. Therefore, for example, a change in a boundary stress condition must be input as a stress incremental change.

Caution: The **STRESS** boundary command affects all degrees of freedom. For example, if **BOUND STRESS** follows a **BOUND XVEL** or **YVEL** command affecting the same gridpoint(s), then the effect of the previously prescribed **XVEL** or **YVEL** will be lost.

Boundary (continued)

Velocity (Displacement) Boundary

XVEl	vx	x-direction velocity for FDEF blocks
YVEl	vy	y-direction velocity for FDEF blocks

Hydraulic Boundary

PP	fp	fluid pressure at boundary (must still use load or stress keywords to set mechanical boundary conditions)
PGRAD	fpg	fluid pressure gradient along y-axis (must still use load or stress keywords to set mechanical boundary conditions)
IMPERMEABLE		impermeable boundary

Non-Reflecting (Viscous) Boundary

Non-reflecting boundaries are available for dynamic analyses:

XVisc		non-reflecting boundary in x-direction
YVisc		non-reflecting boundary in y-direction
Mat	n	material number n assigned to far-field properties (required for non-reflecting boundaries)

Boundary (continued)

Free- Field Boundary

Free-field boundaries are available for dynamic analysis (see **FField** command for additional required commands).

Ffield specifies free-field boundary condition. Note: free-field boundaries are applied to both lateral boundaries of a problem (see Application Problem D for example use of this command).

Boundary Multipliers

Multipliers can be applied to load, stress, or velocity boundaries using the following keywords:

XMul xm x-direction load multiplier (default is 1.0)

YMul ym y-direction load multiplier (default is 1.0)

User-supplied multiplier histories may be stored by UDEC for use later with **XHISTORY, YHISTORY** or **HISTORY** keywords (described under History Types). The user-supplied histories may be unformated and must contain three logical records of the following form:

Line 1. heading of up to 20 full words, 4 characters per word;

Line 2. np, tdel (number of points and timestep, respectively, where np is an integer and tdel is real); and

Line 3 through Line np+2.
np real values of the history variable — i.e., (hist(i), i=1, np). These are assumed to be equally spaced at intervals of tdel.

Boundary (continued)

The first value of each input history is assumed to correspond to time = 0. If a history is supposed to start from its beginning, but the current problem time is not zero, a **RESET TIME** command should be given before cycling.

The keyword and associated parameters for storing the input history are:

HRead	n	filename
		read history n (n=1 or 2 only) from file filename

<u>History Types</u>

Boundary loads, stresses, or velocities can be applied as a constant, linear, sinusoidal, or user-supplied function which remain in effect for one **CYCLE** command. The keywords are:

XHistory	<type>	x-direction boundary parameters
YHistory	<type>	y-direction boundary parameters
History	<type>	both x- and y-direction boundary parameters

Boundary (continued)

where types are:

Constant	constant value (default)
Linear	load increment varies from 0 to 1 during the next **CYCLE** command
Sine	freq tload sine wave load with frequency freq (cycles/sec) and applied for a problem time period of tload
COsine	freq tload cosine wave load with frequency freq (cycles/sec) and applied for a problem time period of tload
Function	history is applied in function FHIST(T) where T is total time. Note: the option for the user to supply a function is available to UDEC source code owners only. Owners with executable codes should use the **HREAD** command.
n	history multiplier n (previously read into UDEC with **HREAD** keyword) is applied (n=1 or 2 only)

CABle x1 y1 x2 y2 npoin mats asteel matg <preten>

Execution of this command creates reinforcing elements which consider explicitly the shear behavior of grout annulus. These elements are generated between endpoints (x1,y1) and (x2,y2). Note: for excavation problems, if point (x1,y1) is inside the excavation periphery, the first nodal point will be on the excavation periphery. The point (x2,y2) should always be located in the rock mass. This reinforcing logic can only be used with fully-deformable blocks which have been discretized into zones. Reinforcement properties and grout properties are specified and stored using the **PROPERTY** commands. The following parameters are also required.

asteel — cross-sectional area of reinforcing (i.e., steel)

matg — material property for grout (The **PROPERTY** command should be used to specify JKS and JCOH for grout. Note: The units for JKS and JCOH are force/unit cable length/displacement and force/unit cable length, respectively.)

mats — material number for reinforcing (i.e., steel) (The **PROPERTY** command should be used to specify G, K, yield, ycomp, and density for the material.)

npoin — number of lumped mass nodal points (npoin ≥ 2)

Pretensioning of the reinforcing can be specified by providing a value (in units of force) for the optional parameter preten.

CAll filename

A remote input command file, filename, can be run with the **CALL** command. Any series of input instructions can be placed in this file to run in a remote or batch mode. The commands **RETURN** or **STOP** may be inserted in the remote file to return input to the local mode or stop execution. At present, only one nesting level of calls is permitted.

CHange <range> keyword <keyword>

Block and joint material characteristics are prescribed and changed with the **CHANGE** command. All blocks with centroids lying within the optional coordinate range <range> have block material characteristics changed according to the keyword. The range must first be on the input line following the command and can be given in one of three forms:

xl xu yl yu

upper and lower limits for x and y

Reg x1,y1 x2,y2 x3,y3 x4,y4

coordinate range given in a <u>clockwise</u> direction

Ann xc yc r1 r2

annular range specified by the radii r1 and r2 whose centroid is xc,yc

If the range is omitted, all blocks have characteristics changed.

Characteristics or joints (i.e., contacts between blocks) are also prescribed and are changed with the **CHANGE** command. Selected joints with contact coordinates within the optional coordinate range, as described above, and / or an orientation range, are changed. The orientation range is defined by either of the following keywords.

ANGle = a1,a2 — Joints with orientation angle within the range a1 to a2 have material property or constitutive number changed (must precede **JMAT** or **JCONS** keyword). If omitted, all joints in range are changed. NOTE: $|a1|$ and $|a2|$ must be less than 180.

CHange (continued)

INterface m1 m2

Joints between block materials m1 and m2 have joint material property or constitutive number changed (must precede **JMAT** or **JCONS** keyword)

The following keywords are used to change characteristics.

<u>Block Characteristics</u>

Cons = n	Constitutive number n is assigned to designated blocks (see note).
Mat = n	Material property number n is assigned to designated blocks. (All blocks initially default to Mat=1.)
SDEF	Block is changed to simply-deformable.

<u>Joint Characteristics</u>

JCons = n	Constitutive number n is assigned to designated contacts (see note).
JMat = n	Material property number n is assigned to designated contacts. (All contacts initially default to **JMAT**=1.)

Note: The present version of UDEC has the following constitutive models:

CHange (continued)

Joint Model

JCONS	Model Description
1	point contact elastic/plastic with Mohr-Coulomb failure (units are force/displacement for contact stiffnesses, and force for cohesion and tension)
2 (default)	joint area contact elastic/plastic with Mohr-Coulomb failure (units are stress/displacement for joint stiffnesses, and stress for cohesion and tension)
3	continuously-yielding joint model (see Section 3.3 and Application Example E for detailed explanation)
5	same as **JCONS** = 2, except that internal fracture flag is set for each joint segment when joint shear or tensile strength is exceeded. If fracture flag is set, then cohesion and tension are ignored in all subsequent calculations.

CHange (continued)

Intact Block Model

CONS	Model Description
0	null material (Null model is used to model excavated material. The stresses within the null block are automatically set to zero.)
1 (default)	linearly-elastic, isotropic
3	elastic / plastic, Mohr-Coulomb failure [This model should be used with caution since accurate solutions to plasticity problems can only be achieved if all triangular zones have a gridpoint at centroid of block. **GENERATE QUAD** zoning has been added to UDEC to improve plasticity analyses. However, no significant errors have been noted to date in problems where the above criteria has not been met.]
4	experimental elastic / brittle fracture by Griffith's theory (for SDEF blocks only).

CRack x1 y1 x2 y2 <JREG>

A crack is created between points (x1,y1) and (x2,y2). **CRACK** can be used to create a discontinuous fracture in part of a block. The **CHANGE** command should be used to assign material property and constitutive numbers to the contacts along the crack. (Note: This command is not applicable for FDEF blocks.)

If the optional keyword JREG is given, cracks will be limited to the region defined by the last **JREGION** command.

CYcle n

Execute n timesteps (cycle 0 is permitted as a check on data). If the <esc> key is touched during execution, UDEC will return control to the user after the current cycle is completed. Note that memory is allocated for histories when the **CYCLE** command is executed. Unused memory is not recovered if execution is interrupted.

DAmping Auto <fac mult1 mult2>
fcrit freq <Stiffness> <Mass> <Internal>

Viscous damping is specified for the blocks. For static or steady-state problems, the objective is to absorb vibrational energy as rapidly as possible. In this case, the first form of the command should be used (**AUTO** keyword); this causes energy to be absorbed in proportion to the rate of change of kinetic energy. The optional parameter, fac, is the ratio of damping dissipation to kinetic energy change. If fac is not given, the default value of 0.5 is taken and gives a fast convergence in most cases. The multipliers, mult1 and mult2, adjust the damping coefficient by a fractional amount as the ratio changes. The default value of 0.99 for mult1 and 1.05 for mult2 are optimum in most cases.

The second form of the command is normally used for dynamic calculations when a certain fraction of critical damping is required over a given frequency range. This type of damping is known as Rayleigh damping, where fcrit = the fraction of critical damping operating at the center frequency of freq. [NOTE: Input frequencies for the program are in cycles / sec — not radians / sec.] The optional modifiers **STIFFNESS** and **MASS** denote that the damping is to be restricted to stiffness- or mass-proportional, respectively. If they are omitted, normal Rayleigh damping is used. (Note: Care is needed with stiffness-proportional damping because it can cause numerical instability if it is too high at the high eigenfrequencies. If UDEC crashes with a "Contact Overlap Too Great" error message, you should suspect this and try the same run with only mass-proportional damping.)

The keyword **INTERNAL** causes the specific damping to be applied to the 3 internal degrees of freedom of simply-deformable (SDEF) blocks.

DElete xl xu yl yu <Area = armin>

All blocks with centroids in the range $xl<x<xu,yl<y<yu$ are deleted if their area is smaller than armin. If armin is not specified, all blocks are deleted in the range.

or

DElete Block n

Blocks are deleted individually by block address number n. Type **PLOT BLOCK BNUM** for block numbers.

Dump n m

Dump memory to screen from the main array from address n to address m. Internal pointers MFREE, JUNK, IBPNT, ICPNT and IDPNT are also printed. MFREE gives the highest memory location that is currently free.

EDge emin

The minimum block edge is set to emin. (Default is twice the rounding length.) emin must be greater than or equal to twice the rounding length (same as the **SET EDGE** command).

End This command was used in previous versions of UDEC to denote the last input command in DATA file. This command is no longer necessary, but it is kept for compatibility with earlier DATA files.

FField

A dynamic free field is generated. This dynamic free field consists of a 1-D finite difference calculation executed in parallel with the main calculation and provides the lateral boundary conditions for dynamic analysis in which a vertically-propagating plane wave is applied to the base of the model. The boundary nodes to be linked to the free field should be defined by using the **BOUNDARY FF** command. The free-field boundary must be applied to both lateral boundaries of the problem. Both lateral boundaries should be vertical and have the same height.

Gen	yl yu np	The free-field mesh is generated between y-coordinates yl and yu with np grid point.
<y1 y2>	Mat n	Material numbers are assigned to the free-field zones. <y1 y2> is an optional range of y-coordinates.
<y1 y2>	Cons n	Constitutive relations are assigned to the free-field zones. <y1 y2> is an optional range of y-coordinates.
OFF		switches off free-field calculation.
ON		switches on free-field calculation.
Initialize		saves the current stress level. This command is required when the **FFIELD** command is switched on after a different type of lateral boundary condition has been used.

FField (continued)

The following keywords describe conditions at the base of the free field (i.e., y = yl). Note: Histories defined in the **BOUNDARY** command are also used here. The following conditions may be defined.

XVel	v	x-direction velocity
YVel	v	y-direction velocity
SXY	s	shear stress
SXX	s	normal stress
XVIsc		viscous boundary in x-direction
YVIsc		viscous boundary in y-direction

See Application Problem D for example use of this command.

Fix xl xu yl yu

All blocks with centroids in the range $xl<x<xu,yl<y<yu$ have current velocities fixed. FDEF blocks cannot be fixed. Refer to **RSET** command to specify velocity other than the current value.

FLUID keyword v <keyword v . . . >

Fluid flow properties are set by using the following keywords.

		Units
Rhow	fluid density	[mass / volume]
Bulkw	bulk modulus	[stress]
Cohw	yield strength	[stress]

Rhow is used to calculate buoyancy forces. By default, the flow logic assumes the water table is above all blocks — the water table is at y = 1e20. To eliminate buoyancy forces, use the **SET YWTAB y** command.

Bulkw is used for flow calculations. If Bulkw is zero, then the fluid flow calculation is suppressed. (Default Bulkw is zero.)

Cohw is the yield strength for non-Newtonian fluids (e.g., Bingham flow, see Application Problem G in Appendix B). (Default Cohw is zero.)

FRAction fb <fz> <fw>

fb is taken as the fraction of critical timestep to be used for block timestep (default fb=0.1). fz is the fraction of critical timestep to be used for zone timestep (default fz = 1.0). fw is the fraction of critical timestep to be used for fluid flow analysis (default fw = 1.0).

FREe xl xu yl yu

All blocks with centroids in the range $xl<x<xu, yl<y<yu$ are set free. Note: by default, all blocks are free initially.

GEnerate <range> keyword v < . . . >

All blocks with centroids lying within the optional coordinate range <range> are discretized into fully-deformable triangular finite difference zones. The range must be first on the input line following the command and can be given in one of two forms:

xl xu yl yu — upper and lower bound limits for x and y

Reg x1,y1 x2,y2 x3,y3 x4,y4

coordinate range given in a clockwise direction

If no range is given, then all blocks will be discretized.

Zone generation can be performed automatically or manually. The following keywords apply:

Edge	edmax	automatic generation of zones for arbitrarily-shaped block. The parameter edmax defines the maximum edge length of the triangular zones.
Quad	xw <yw>	automatic generation of zones for quadrilateral blocks only. Diagonally-opposed triangular zones are created to improve plastic flow calculation. Parameters xw and yw are zone widths in x- and y-direction. If yw is not given, the parameter defaults to xw.

GENerate (continued)

Manual Gridpoint glist . . . Zone zlist . . .

For manual generation, a list of gridpoints, glist, and zones, zlist, must be given. The format for glist is: xl yl x2 y2 x3 y3 . . . , where each x,y pair is a coordinate of a gridpoint. If a given coordinate lies within a certain tolerance of a block corner, the gridpoint is placed on that corner. If the coordinate lies within the same tolerance of a block edge, a new corner is created in the edge. The tolerance is taken as 0.9 times the rounding length. The format for zlist is: l1 m1 n1 l2 m2 n2 . . . . Each triple corresponds to the three gridpoints that define the zone, where the numbering of the gridpoints refers to the order in glist, starting with the last (i.e., last gridpoint is number 1). Both glist and zlist may extend over an arbitrary number of continuation lines, but doubles and triples should not be split over two lines. Care must be taken, when using manual zoning, to ensure that zoning completely fills the block.

For example:

```
BLOCK 0,10 0,20 10,20 10,10
GEN MAN GRID 5,15 0,20 0,10 &
             10,10 10,20 &
        ZONE 1,2,5 2,3,5 &
             3,4,5 4,1,5
```

will divide a block into four zones with five gridpoints.

GRavity gx gy

Gravitational accelerations are set for the x- and y-directions (same as **SET GRAVITY** command).

HEading

The next input line is taken as a heading printed on subsequent restart files and plot copies.

HIstory keyword <keyword> . . .

A time history is kept of selected variables. Variables are accessed by their "history" number, which corresponds to the numerical position of the variable in the list of variables following the **HISTORY** command. The variable value is stored every Ncyc cycles. A time history of as many as 40 variables can be made in one run. A maximum of 20 **CYCLE** commands can follow a **HISTORY** command. For more cycling, the history must be reset with the **RESET** command followed by a new **HISTORY** command. The available keywords are:

Address	i j	any real variable with address i and offset j (see Section 6.1 for offsets)
BFM	n	block centroid moment (n is block number) for RIGID or SDEF blocks
BFX	n	block x-direction centroid force (n is block number) for RIGID or SDEF blocks
BFY	n	block y-direction centroid force (n is block number) for RIGID or SDEF blocks
Damp		auto damping parameter
Function	n	history of a user-supplied variable is followed in function FPLHIS(n), where n is variable number. Note: this history is available to users with source codes only.
List		prints lists of all history types and locations, except thermal histories. A list of thermal history types and locations can be obtained by typing **THIST LIST** (same as the **PRINT HIST** command).

HIstory (continued)

Ncyc	n	histories are sampled every n cycles (default is n = 10)
NDis	n	normal displacement (n is contact number)
NStr	n	normal stress (n is contact number)
SDis	n	shear displacement (n is contact number)
SStr	n	shear stress (n is contact number)
SXX	x y	xx-stress at location nearest to x,y (fully-deformable blocks only)
SXY	x y	xy-stress at location nearest to x,y (fully-deformable blocks only)
SYY	x y	yy-stress at location nearest to x,y (fully-deformable blocks only)
Type	n	Displays the value of the variable (with history number n) on the console screen during cycling. Only one variable can be selected using this command.
Unbal		maximum resultant gridpoint force
Write	n fname	write variable (with history number n) time-history to file fname.
XDis	x y	x-displacement at location nearest to x,y
XHIS		x-force history of applied boundary history

HIstory (continued)

XVel	x y	x-velocity at location nearest to x,y
YDis	x y	y-displacement at location nearest to x,y
YHIS		y-force history of applied boundary history
YVel	x y	y-velocity at location nearest to x,y

INsitu <xl xu yl yu> STress sxxo sxyo syyo <keyword> . . .

Initial zone stresses in all FDEF blocks and joints in the range xl<x<xu and yl<y<yu are set to sxxo, sxyo and syyo. An optional linearly-varying initial stress initialized at origin (0,0) can be set by the following keywords and variables:

XGrad	sxxx sxyx syyx
YGrad	sxxy sxyy syyy

where

sxx = sxxo + (sxxx • x) + (sxxy • y)
sxy = sxyo + (sxyy • x) + (sxyy • y)
syy = syyo + (syyx • x) + (syyy • y)

NODIS inhibits calculation of normal displacements.

If total pressure option has been selected (i.e., SET -EFF), then in-situ fluid pressures are also initialized (fluid density, gravity (if present) and water table location must be previously defined).

ISET ival ia ioff

The integer value ival is inserted in the main array at address ia, with offset ioff. See Section 6.1 for offsets.

JDelete

Joints which do not completely intersect a block are deleted.

JRegion x1 y1 x2 y2 x3 y3 x4 y4

A quadrilateral region is defined for generation of a joint set (or sets). The coordinates of the four corners (in clockwise direction) delimit the boundary of the region. A joint set defined by the **JSET** command will be generated in all blocks or portions of blocks within this region. If the region is not specified, it is assumed to be the entire problem domain. The **JREGION** command can also be used to limit the action of the **CRACK** command.

JSet a(m),a(d) t(m),t(d) g(m),g(d) s(m),s(d) <x0 y0> <ad0>

A joint set is generated in the region defined by **JREGION** with characteristics defined by the following parameters (see Fig. 4-1 for illustration of parameters):

a	angle of joint track to x-axis
t	trace length of joint segment
g	gap length between joint segments
s	spacing normal to joint tracks

For each pair of values, the first entry should be the mean value and the second should be the maximum random deviation from the mean (for uniform probability distribution).

The final three parameters are optional:

x0 y0	coordinates (global axis) of the start of one joint trace. A joint will be generated starting at (x0,y0); further joints will be generated to fill the region defined by **JREGION**.
ad0	deviation of angle of all joints from the direction given for the joint track

JSet (continued)

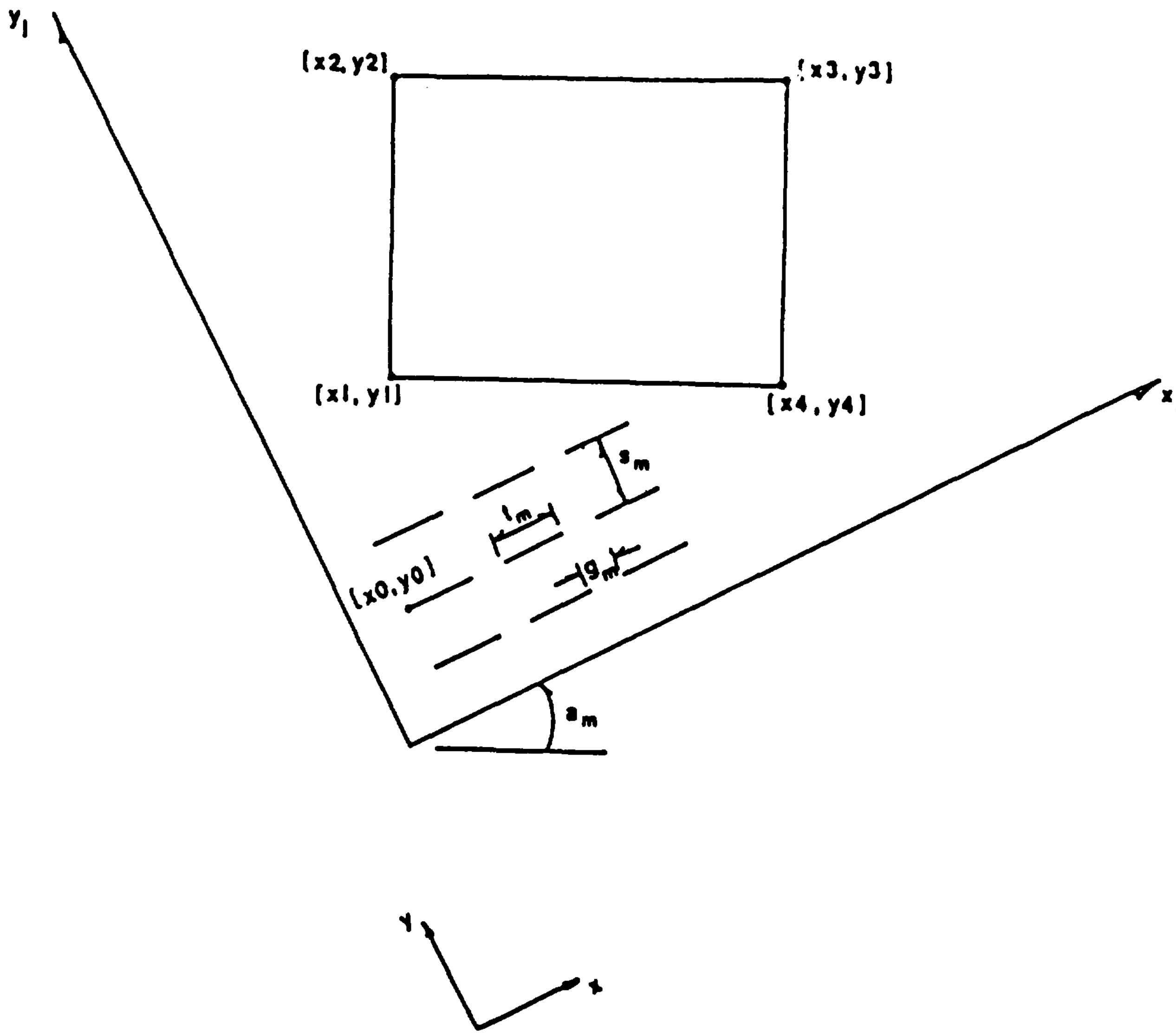


Fig. 4-1 Joint Set Parameters

LInk x1 y1 x2 y2

Mathematically links a "flying block" to the main data structure — (x1, y1) are the coordinates of any point inside the flying block and (x2, y2) are the coordinates of any point inside the block which will provide the link to the flying block. This block should be the one which is topologically closest to the flying block. (x1, y1) and (x2, y2) should be chosen close to the blocks' centroid locations to ensure correct linkage. See, also, discussion of limitations in Section 5.2.

Load xl xu yl yu xload yload

All blocks with centroids lying within the range $xl<x<xu$, $yl<y<yu$ are prescribed static loads applied at the block centroid. (This command is only applied to Rigid or SDEF blocks. Use **BOUNDARY** command for more general boundary conditions.)

Mscale ON <zcmass bcmass>
OFF

Mass scaling for all blocks is turned on to speed up convergence of static problems involving very non-uniform grids or very non-uniform elastic properties. For rigid and SDEF blocks, the user must supply a value for bcmass. (A dummy place for zcmass is therefore required.) For FDEF zcmass, bcmass need not be supplied. If no value is supplied, the code will perform density scaling automatically. If desired, the user may select to set all zone masses to zcmass and rigid block masses to bcmass by specifying those parameters. Experience has shown that average zone and block masses generally lead to satisfactory solutions.

Values for minimum, average and maximum zone and block masses can be found using the **PRINT MAX** command. Mass scaling can be turned off at the user's discretion.

Mscale PART tdel

Experimental partial density scaling for FDEF blocks. This command applies density scaling only to those zones necessary to achieve timestep tdel. This command is useful for dynamic analysis.

NEW This command allows the user to begin a new problem without leaving UDEC (same as **START**).

PFix <xl xu yl yu> keyword <keyword>
<Region x1,y1 x2,y2 x3,y3 x4,y4>

This command allows fluid pressure to be specified within domains (i.e., "open" spaces between blocks). It should be noted that the specified fluid pressures do not change unless a different **PFIX** or **PFREE** is executed, or new contacts are formed or deleted. To avoid forming or deleting contacts, use larger rounding lengths.

The fluid pressure is fixed in the domains lying within the optional range <range>. The range must be first on the input line following the command and can be given in one of the following forms.

xl xu yl yu		upper and lower limits for x and y
Reg	x1,y1	x2,y2 x3,y3 x4,y4 coordinate range given in a clockwise direction
Domain	n	domain with address n. Domain addresses can be found using the **PRINT DOMAIN** command.

If the range is omitted, the command is assumed to apply to all domains except the outer domain.

The following keywords are used to set the fluid pressure in a domain.

Pressure	p	fluid pressure
Xgrad	dx	fluid pressure gradient in x-direction
Ygrad	dy	fluid pressure gradient in y-direction

Total fluid pressure, PP, in a domain is determined as follows

$$PP = p + dx \bullet x + dy \bullet y$$

Note: Positive pore pressures cause the domain to expand.

PFix (continued)

Example **PFix** commands are:

> **PFIX** dom=521 pressure=10
> **PFIX** pressure=10
> **PFIX** 0 10 25 50 pressure=10 xgrad=1
> **PFIX** region 0,0 0,10 10,10 10,0 pressure=15

PFRee ia

The pressure is not controlled in the domain with address ia. If ia = 0, pressure is not controlled for all domains.

PRint <range> keyword <keyword . . . >

Printed output is produced for all data prescribed by keywords. The printed data for certain keywords (designated by *) can be restricted to an optional coordinate range <range>. The range must be first on the input line following the command and can be given in one of three forms:

xl xu yl yu — upper and lower limits for x and y

Reg x1,y1 x2,y2 x3,y3 x4,y4

coordinate range given in a clockwise direction

Ann xc yc r1 r2

annular range specified by the radii r1 and r2 whose centroid is xc,yc

MAt n — print block data for blocks with material number n. Use with **BLOCK**, **RIGID**, **SDEF**, **GRIDPOINT** or **ZONE** keywords.

If no range is given, then all data for the prescribed keyword will be printed Printing can be interrupted at any time by pressing the <esc> key. Data are printed for the following keywords:

BEBound — stress and displacements in boundary elements (Note: These are induced values — not total values.)

BEInt n x1 y1 x2 y2

stress and displacements at n points along line between x1,y1 and x2,y2 in boundary element domain (Note: These are induced values — not total values.)

PRint (continued)

Blocks*	general block data for all block types		
BOundary* <keyword>	boundary condition keywords:		
	BE	boundary element conditions (see offsets for boundary corner array in Section 6.1)	
	Disp	displacements at boundary	
	Fluid	fluid pressure and permeability condition	
		0	permeable
		1	impermeable
	Forces	forces at boundary	
	State	boundary condition state specified by boundary code ix,iy	
		0	free
		1	stress (or force)
		2	boundary element
		3	viscous
		4	velocity
		ff is free-field indicator	
		0	off
		1 or 2	on
	Thermal	temperature at boundary	

Note: If **PRINT BOUND** only is typed, all boundary condition information is printed.

PRint (continued)

CAble		data for experimental reinforcing elements (cables) which consider shear behavior of grout annulus
Contacts*	<keyword>	contact data keywords: Disp — contact displacements Force — contact force / stress data (Note: if joint length = 0.0, contact force is printed; otherwise, joint stress is printed.) State — contact state data (i.e., material and constitutive numbers, length, angle, contact type and adjacent block addresses). Note: If **PRINT CONTACT** only is typed, all contact data are printed.
CORners		corner data
DList		domain linked list
Domains*		domain data
File	filename	filename puts output to file "filename". Keyword "FILE" must precede keywords of data to be put in file "filename".
Flows*		fluid flow data

PRint (continued)

Gridpoint	keyword	gridpoint data for FDEF blocks:
		Disp — gridpoint displacements and velocities
		Force — gridpoint forces
		State — gridpoint state data (i.e., corner-link address and unscaled mass)
		Temp — gridpoint temperature (see Section 7.0)
History	<n1 . . . >	time history of variables n1, n2, ... If no history is specified, a list of all history types and locations is printed.
HMax	n1 . . .	maxima of time histories n1,n2, . . . set by **HISTORY** command
Joint	x1,y1 x2,y2 <tolc>	prints contact data for a single joint along line from (x1,y1) to (x2,y2) within a band ± tolc perpendicular to the line [(x1,y1), (x2,y2)]. If tolc is omitted, a band width twice the rounding length is used.
List		block linked list
LM		structural element lumped mass data

PRint (continued)

Maxima		maxima and minima of block and zone data (zone data will not print out if FDEF blocks have not been zoned)
Property	<keyword>	property keywords:

Block	block properties
BOundary	far-field material properties
CAble	cable properties
Contact	contact properties (JCON=1)
Fluid	fluid flow properties
Joint	joint properties (JCON=2 or 5)
Reinforcement	reinforcing element properties
Structural	structural element properties
Thermal	thermal properties

Note: If **PRINT PROP** only is typed, all properties are printed.

Reinforce*	local reinforcing data
RIgid	rigid block data

PRint (continued)

SDEF		SDEF block data
STAtus		present status of global variables
STC		structural element interface data with excavation periphery
STE		structural element data
STRess	xl xu yl yu nx ny	Principal stresses in the range $xl<x<xu$ and $yl<y<yu$ are printed on grid nx,ny. The data are also sent to a file named "STRESS".
Table	ntab	printout of values in table ntab. Set by **TABLE** command.
Zone	keyword	zone data keywords for FDEF blocks:

STAte	zone state data (i.e., zone address, gridpoint addresses, unscaled zone mass)
STRess	zone stresses and plasticity indicator:

0	elastic
1	at failure
2	failed in the past
3	tensile failure

PROperty Material n keyword v <keyword v . . .>
JMaterial n

The first parameter must be the specification of the solid material or joint material number. Material properties, v, are defined for material number n. All material properties must be specified as a positive value. Block and joint properties are not assigned to specific blocks or joints by the **PROPERTY** command. Assignment of particular properties to specific blocks or joints is done with the **CHANGE** command. Cable, reinforcing element and structural element properties are also assigned with the **PROPERTY** command to MATERIAL number n (see **CABLE**, **REINFORCING** and **STRUCTURE** commands.

CAUTION: Do not use material numbers for cable, reinforcing or structural element properties which are the same as those for joint and block material numbers. Always use the **PRINT PROPERTY** command to check property assignment.

PROPERTY keywords for the available constitutive (see **CHANGE** command) models are:

Joint Material Properties (JMATERIAL keyword)

JCONS = 1		Units
KN	contact normal stiffness	[force/displacement]
KS	contact shear stiffness	[force/displacement]
CCOhesion	contact cohesion	[force]
CPerm	contact permeability	[length^2/(stress • time)]
CDIlation	contact dilation coefficient	[tangent of dilation angle]
CFriction	contact friction coefficient	[tangent of friction angle]
CTensile	contact tensile strength	[force]

PROperty (continued)

JCONS = 2, 3 or 5		Units
AZero*	aperture for zero normal stress	[length]
ARes*	residual aperture at high stress	[length]
JKN	joint normal stiffness	[stress / displacement]
JKS	joint shear stiffness	[stress / displacement]
JCoh	joint cohesion	[stress]
JDil	joint dilation coefficient	[tangent of dilation angle]
JFric	joint friction coefficient	[tangent of friction angle]
JPerm	joint permeability constant	[1 / (stress • time)]
JTensile	joint tensile strength	[stress]

---

*CAUTION: The maximum hydraulic aperture in a model is limited to the product of CAPRATIO and the maximum value for ARES. In order to consider larger hydraulic apertures, increase CAPRATIO using the **SET** command (by default, CAPRATIO=5). Hint: Set CAPRATIO=1 to maintain constant apertures.

PROperty (continued)

Note: **JCONS** = 3 (supplemental joint properties for continuously-yielding model) uses the same joint property parameters as required for **JCONS** = 2 (i.e., JKN, JKS, etc.), plus the following:

JEN exponent of joint elastic normal stiffness. Specifying this parameter permits joint normal stiffness to be a function of current normal stress — i.e.,

$$\text{JKN} := \text{JKN}\,(\sigma_n{}^{\text{JEN}})$$

where ":=" means replaced by, and σ_n is the current normal stiffness.

JES exponent of joint elastic shear stiffness. Use of this parameter is identical to that discussed previously — i.e.,

$$\text{JKS} := \text{JKS}\,(\sigma_n{}^{\text{JES}})\ .$$

JIF joint initial friction angle [radians]

JR joint roughness parameter [length]

NOTE: **JCONS = 5** is similar to **JCONS=2** except that an internal fracture flag is set for each joint segment when the joint tensile or shear strength is exceeded. If a joint is fractured (i.e., fracture flag is set), then joint tensile strength and joint cohesion are ignored in all subsequent calculations.

PROperty (continued)

Block Material Properties (MATERIAL keyword)

CONS = 1 or 3		Units
Bulk (or K)	block bulk modulus	[stress]
SHear (or G)	block shear modulus	[stress]
Density	block density	[mass/volume]

NOTE: The equations for bulk and shear modulus are:

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

$$K = \frac{E}{3(1 - 2\nu)}$$

CONS = 3		Units
Cohesion	block cohesion	[stress]
Friction	block friction coefficient	[tangent of friction angle]
Tensile	block tensile strength	[stress]
DIlation	block dilation coefficient	[tangent of dilation angle]

PROperty (continued)

CONS = 4		Units
TF	block tensile strength factor based on Griffith's theory (for SDEF blocks only) where:	[stress]

$TF = sp1$ if $3sp1 + sp2 > 0$

and

$TF = -(sp1 - sp2)^2 / 8(sp1 + sp2)$

if $3sp1 + sp2 < 0$

sp1 is the maximum principal stress in the SDEF block, sp2 is the minimum principal stress in the SDEF block, and TF corresponds to the uniaxial tensile strength of the intact material.

Hoek-Brown Strength Criterion (MATERIAL keyword)

This is used for Hoek-Brown Strength/Stress contour plots only.

HBS	Hoek-Brown "s" parameter
HBM	Hoek-Brown "m" parameter
Ucs	unconfined compressive strength

PROperty (continued)

Cable Reinforcing Properties (MATERIAL Keyword and **CABLE** Command)

		Units
Shear (or G)	elastic shear modulus for cable reinforcing	[stress]
Bulk (or K)	elastic bulk modulus for cable reinforcing	[stress]
Density	mass density for cable reinforcing	[mass / volume]
YComp	compression yield force for cable reinforcing [use positive value]	[stress]
Yield	tensile yield force for cable reinforcing [use positive value]	[stress]

Cable Grout Properties (MATERIAL Keyword and **CABLE** Command)

		Units
JKS	grout shear stiffness	[force / unit cable length / displacement]
JCoh	grout shear strength	[force / unit cable length]

PROperty (continued)

Reinforcing Element Properties (MATERIAL Keyword and **REINFORCE** Command)

		Units
RAStiff	axial stiffness	[force / length]
RSStiff	shear stiffness	[force / length]
RLength	1/2 "active" length	[length]
RUAxial	ultimate axial capacity (must be > 0)	[force]
RUShear	ultimate shear capacity (must be > 0)	[force]
RRFac	reversal factor (usually 1)	[—]
RSExp	shear stiffness exponent (usually 1)	[—]
RAExp	axial stiffness exponent (usually 1)	[—]

Structural Element Properties (MATERIAL Keyword and **STRUCTURE** Command)

		Units
Elements		
Dens	mass density of element	[mass / structure]
Bulk (or K)	elastic bulk modulus for element	[stress]
Shear (or G)	elastic shear modulus for element	[stress]

PROperty (continued)

		Units
Interface Material		
KN	interface normal stiffness	[force / displacement]
KS	interface shear stiffness	[force / displacement]
CCohesion	interface cohesion	[force]
CFriction	interface friction coefficient	[tangent of friction angle]
CTensile	interface tensile strength	[force]

Quit The run stops (same as **STOP**).

REInforce mat x1 y1 x2 y2

Execution of this command causes reinforcing elements to cross joints which intersect the line with endpoints (x1,y1) and (x2,y2). If reinforcement is to be used with fully-deformable blocks, the FDEF blocks must be discretized before reinforcement locations are specified. Reinforcement properties are specified and stored using the **PROPERTY** command with material number mat.

RESET keyword <keyword . . . >

Certain variables are reset to zero according to the keyword:

Disp		All displacements of gridpoints of FDEF blocks are set to zero, but gridpoint coordinates are not affected. This command does not alter the physics of the problem being modeled because displacements are not used in any calculation.
Hist		The values of the current selection of output histories are set to zero. This keyword must be used if more than 20 **CYCLE** commands follow a **HISTORY** command.
INterface	m1 m2	Only joints between block materials m1 and m2 have joint stresses and displacements reset to zero (must precede **STRESS** keyword).
JDisp		All joint shear displacements are set to zero.
Mat	n	Only blocks with material n have stresses reset (must precede **STRESS** keyword).
Time		Problem (mechanical) time is set to zero. This has no effect on the problem being modeled but is useful for input histories and for convenience in presenting results. Thermal time is not affected.
Rota		All zone and block rotations are set to zero.
Stress		All stresses in blocks and joints are set to zero. Stresses in selected blocks can be reset using the **MAT** keyword preceding the **STRESS** keyword. Stresses in selected joints can be reset to zero using the **INTERFACE** keyword preceding the **STRESS** keyword.

Restart <filename>

The program is restarted using data from the named file (or else from the default file, if no file name is given).

RSet v ia ioff

The real value v is inserted in the main array at address ia, with offset ioff. See Section 6.1 for offsets.

RETurn returns the program control to the local mode.

ROund d

Each block corner is rounded with a circle that is tangential to the two corresponding edges at a distance d from the corner. The default value for d is 0.5. A round length equal to one percent of the typical block edge length is recommended. It should be noted that for fully-deformable blocks, corner gridpoints are located at the corner of the unrounded block. It is also recommended that once the round length is specified it not be changed, especially after cycling is initiated.

SAve <filename>

The current problem in memory is saved at its present state on the named file, <filename> (or on the default file, if no file name is given).

SEt keyword <keyword>

This command is used to set a variety of problem execution variables. The keywords for this command are:

CSCAn	n	update center coordinates for contacts every n cycles (default is n = 100)
CAPratio	r	maximum contact hydraulic aperture is limited to r times the residual aperture (default is 5).
CLEmin	cmin	minimum contact length for JCONS = 2,3 or 5 (default is 1/10 of average contact length)
DScan	n	scan domains for new contacts only every n cycles (default is n = 3). Note: **DSCAN** should only be used if the model has many contacts and joint displacements are small.
Edge	emin	The minimum block edge is set to emin (default is twice the rounding length). emin must be ≥ twice the rounding length (same as for the **EDGE** command).
EFF		selects old fluid flow logic, in which domain fluid pressures do not include the hydrostatic pressure. Buoyancy loads are applied to blocks with centroids below water table. This is the default option.

SEt (continued)

-EFF		selects new fluid flow logic, in which domain pressures correspond to the total fluid pressure. Water table location is only used by the **INSITU** command to calculate initial fluid pressure. Fluid pressures are given by the **BOUND** command (i.e., a gradient in the y-division is normally required). This option allows more general boundary conditions, for example, when the problem top surface is not plane.
FUP		performs boundary force update (default).
-FUP		suppresses boundary force update.
Gravity	gx gy	gravitational acccelerations are set for the x- and y-directions.
JCondf	n	default constitutive relation for new contacts (after cycle 1). If JCondf is not specified, it is assumed equal to 2.
JMatdf	n	default material number for new contacts (after cycle 1). If JMatdf is not specified, it is assumed equal to the block material number on one side of the joint.

SEt (continued)

Log	ON	opens a file named "UDEC.LOG"on the default disk drive. If a file "UDEC.LOG" already exists, it is overwritten. Any text which is printed to the screen from this point is also written to the log file. This is particularly useful for keeping a record of interactive sessions. The file may be edited to create batch data files.
	OFF	turns off the logging function. It does not close the log file "UDEC.LOG". If **SET LOG ON** is given at some later stage in the session, subsequent screen output will be appended to the file.
OVtol	ovtol	contact overlap tolerance set to ovtol (default = one-half rounding length)
Ptol	p	domain pressure variable tolerance for fluid flow calculation (default is 0.01).
SFlow		sets fluid flow algorithm which provides faster convergence than the standard calculation. This command is to be used in analyses where only the steady state condition (and not the transient response) is required.
-SFlow		suppresses steady-state flow condition (default)
UPcon	n	update contact locations only every n cycles (default is n = 1)
Write		writes "scratch" joint file to RJOINT.
-Write		does not write "scratch" joint file (default).

SEt (continued)

Xform	ON	permits writing a formatted save file for transfer between versions of UDEC installed on different computers.
	OFF	writes an unformatted binary save file which cannot be transferred between computers but is smaller in size. (Default is **XFORM=OFF**.)
YWtable	y	y-coordinate of water table If **SET EFF** is selected, blocks with centroids below this line are subject to buoyancy force if gravity is specified. Gravity is assumed to act in the y-direction only. The default condition assumes the water table is above all blocks — i.e., fully- saturated conditions. If **SET -EFF** is selected, water table information is used by the **INSITU** command only.
ZPress	x y rad pz	sets pressure, pz, in zones (of FDEF blocks) within a radius, rad, of location (x,y). The pressure will obey histories defined by the **BOUNDARY** command.

SPlit x1 y1 x2 y2

All blocks in the path of a line extending from point (x1, y1) to (x2, y2) are split into two. The **CHANGE** command should be used to assign material property and constitutive numbers to the contacts created by **SPLIT**.

Note: Unlike the **CRACK** command, this command neglects joints which do not completely separate a block in two. (Note: This command is not applicable for FDEF blocks.)

Start This allows the user to begin a new problem without leaving UDEC (same as **NEW**).

STOp The run stops (same as **QUIT**).

STRess xl xu yl yu nx ny

Principal stresses in the range xl<x<xu and yl<y<yu are printed on grid nx,ny (same as **PRINT STRESS** command).

STRUcture

xc yc fang rrange npoin mat thick <ang>

Structural elements are generated in interior region. Center of elements generation is at (xc,yc) with the first node at an angle fang measured from the positive x-axis. Search for nodal points is restricted to edges within a radius range from (xc,yc). The number of elements is given by npoin, the material number by mat, and the thickness of the structural elements by thick. The optional parameter ang may be specified as in the **ARC** command. Here, ang is a counterclockwise angle measured from fang.

NOTE: Both the structure material and interface material are assigned by this command. Required properties for the interface are identical to those for JCONS=1. Required properties for the structure are density, bulk modulus, and shear modulus. (See the **PROPERTY** command.)

TAble ntab x1,y1 x2,y2 x3,y3 . . .

Table of values can be input for use by various commands in UDEC. ntab is the table number and x,y are paris of values for table ntab. (Presently, this is only available for the **TADD** command in thermal analysis, Section 7.0).

Thermal This command must precede the **BLOCK** command if any thermal analysis is to be performed at any stage of the problem. (See Section 7 for a full description of thermomechanical analysis with UDEC.)

Tunnel xc yc r n

A circular joint or crack pattern is created where xc,yc is the center of the circle, r is the radius, and n is the number of segments or sides defining the circle. This command must be used before blocks are made fully-deformable.

Following execution of this command, future cracks or joints will not penetrate the tunnel periphery. If it is desirable for cracks to penetrate the periphery, specify a second tunnel with zero radius at some point outside the problem domain, or use a **SPLIT** command.

UPCON An update is performed for all coordinates of contacts. This command is useful when generating complex joint patterns.

UPDOM An update is performed for all domains. This command is useful when generating complex joint patterns.

4.6 Full Description of Input Commands for Plotting

COPY <filename>

Execution of this command causes hardcopy plot of previous screen plot to be made. If the optional filename is specified, output for this plot only will be directed to specified file. To change hardcopy routing globally, use the **SET OUT** command. If no file names are specified, the plot data will be sent to the default files. The default file for interactive operation is COM1. The default file for batch operation is PLT.

HEading The next input line is taken as a heading printed on subsequent restart files and plot copies.

JPLot START
STOP

Joints generated by **JREgion**, **JSet**, and **CRack** commands between **START** and **STOP** keywords are plotted to the screen (in interactive mode) to help the user formulate a joint structure. In batch mode, the plot files are written to the default file PLT or any file specified by the **SET OUT** command. Splitting of blocks is not performed for the joint generation commands between **START** and **STOP**.

PLot keyword <switch . . .> <keyword <switch . . .> . . .>

Requests a plot to be made to the screen. Several variables may be plotted at once by giving several keywords on one line. Certain keywords may be followed by <sc>. If the parameter sc is specified, vector magnitudes are multiplied by sc. Vectors are always plotted in window units. If sc is not specified, the multiplier defaults to 5% of the maximum window dimension divided by the maximum vector magnitude.

Certain joint parameters may also be scaled using <sc>. In this case, the value sc gives the reference line thickness. If sc is not specified, all line thicknesses are plotted relative to the maximum value. The optional parameter sc is useful in comparing results.

The keywords are as follows:

APerture	<sc>	joint aperture
Axial	<sc>	axial forces in reinforcement
BLocks		assemblages of blocks
BNum		block address numbers plotted at block centroids
BOundary		outer boundary (can only be plotted after execution of **BOUNDARY** command)
CAble		location of cable reinforcing elements. Note: if the **PLOT MATERIAL** command is executed, reinforcing elements will be plotted in the appropriate material color.
CForce	<sc>	plots the axial force in all cable bolt elements. The optional parameter <sc> can be used to scale the axial force vector.

PLot (continued)

CLosure	<sc>	joint closure and separation [Note: separation may not plot if compressive normal stresses have never been present.]
Contacts		normal to contacts
Disp	<sc>	gridpoint displacement vectors
DShe		directions of shear displacement
Flow	<sc>	plot fluid flow magnitudes in joints
FRactures		joints with **JCONS** = 5 which have fracture flag set
FVel	<sc>	plot fluid velocities in joints
Gpforces	<sc>	gridpoint force vectors
HIstory	n1 <n2 . . .>	
		time history of variables n1, n2 . . . assigned by **HISTORY** command
Hoek		contours of strength / stress ratio for the Hoek-Brown criterion with a tension cut-off

PLot (continued)

JLine x1 y1 x2 y2 keyword

plots a linear variation of the joint parameters, defined by the keyword, for all joints which are intersected by the line from (x1,y1) to (x2,y2) at the location where the line intersects the joint. The following keywords may be used:

Ape	aperture
Flow	flow rate
NStr	normal stress
PP	pore pressure
SStr	shear stress
Vel	fluid flow velocity

In place of the keyword, a joint (contact) offset number, koff, may also be used (see Section 6.1).

Joint x1,y1 x2,y2 keyword <tolc>

plots a linear variation of the joint parameter, defined by the keyword, for a single joint along the line from (x1,y1) to (x2,y2) within a band ±tolc perpendicular to the line [(x1,y1), (x2,y2)]. The following keywords are used:

Ape	aperture
Flow	flow rate
NStr	normal stress
PP	pore pressure
SStr	shear stress
Vel	fluid flow velocity

In place of the keyword, a joint (contact) offset number (koff, may also be used (see Section 6.1).

MAT material property number assigned to blocks and joints (designated by color)

PLot (continued)

MOhr		contours of strength / stress ratios for the Mohr-Coulomb criterion with a tension cut-off
Plasticity		the plastic state (i.e., elastic, plastic, unloaded, failed in tension) of fully-deformable (FDEF) zones with CONS=3
PP	<sc>	pore pressure
REinforce		reinforcement crossing joints
Rotate	Block Zone	rotation angle of blocks or zones (FDEF blocks only)
SDif		principal stress difference contours (FDEF blocks only)
SHear	<sc>	shear displacement of joints. The optional scale <sc> may be used to provide a reference shear displacement — for example, in comparing several similar problems. If sc is not specified, then all shear displacements are scaled to the maximum value.
SLip		joints which are near limiting friction or are separated
SMIN		minimum principal stress contours (FDEF blocks only)
SMAX		maximum principal stress contours (FDEF blocks only)

PLot (continued)

Stresses	<sc>	zone principal stresses of FDEF blocks or average principal stresses of SDEF blocks
STRUct		structural element geometry
SXX		xx-stress contours (FDEF blocks only)
SYY		yy-stress contours (FDEF blocks only)
SXY		xy-stress contours (FDEF blocks only)
Tension		a single contour separating regions of tension and compression
Vel	<sc>	block and / or gridpoint velocity vectors
Vflow	<sc>	flow vectors indicating fluid flow direction
Window		plotting window
XDis		gridpoint x-displacement contours
XVel		gridpoint x-velocity contours
YDis		gridpoint y-displacement contours
YVel		gridpoint y-velocity contours
ZOnes		zones in FDEF blocks

PLot (continued)

If no further parameters are given, certain plotting characteristics will be chosen automatically. However, certain keywords may be followed by any number of "switches" which set certain characteristics of the plot. Each switch operates on the keyword which precedes it. The following switches are available:

GRid	nx ny	partitions the window into nx by ny contouring points to be used to determine contours (default nx=ny=20).
Interval	val	contours will be plotted at interval of val
Line	x1 y1 x2 y2	labels the contours, placing labels along a line from x1,y1 to x2,y2.
Max	vmax vmin	maximum value of contour, vmax, and minimum value of contour, vmin, are set.
NC		corner rounding deleted for plotting blocks. Deleting rounded corners speeds plotting.
NOheading		suppresses captions on screen plot (Note: This must precede **HISTORY** command, if used.)
ZEro		suppresses zero contour line

PLot (continued)

The following supplemental keywords may be used to modify history plots to achieve more meaningful presentation. The following supplements may be used before the HIST keyword.

AB	n	plots history n on abscissa axis; (n assigned by **HISTORY** command) [Default for abscissa is problem time.].
XRev		reverses sign of history plotted to abscissa axis.
XWINDOW	xl xu	specifies range for abscissa axis.
YREV		reverses sign of history plotted to ordinate axis.
YWINDOW	yl yu	specifies range for ordinate axis.

SEt keyword <keyword>

This command is used to set a variety of plotting parameters. The keywords for this command are:

Plotting Keywords

CGA		sets graphics mode to color screen (CGA)
CHAR	i	used to select alternate character sets for the HP plotters or the Sperry laser printer. Consult the appropriate manuals for applicable values (Scandinavian = 30 for the HP and 6 for the Sperry). [default = 0 = standard ASCII]
EGA		sets graphics mode to color screen for use with enhanced color monitors (default).
Mono		sets graphics mode to monochrome, high resolution
Output	p	p may be any logical device or filename. COM1, COM2 or LPT1 will send plot data out to external devices. (CON will send plot data to the screen). P may be a disk filename; however, the file will be opened each time a **COPY** command is issued. Multiple **COPY** commands with the same filename designated will cause the file to be overwritten each time. [NOTE: If no filename is specified, plot data will be sent to COM1 (interactive mode) or PLT (batch mode).]

SEt (continued)

PLOT	<keyword>	This sets plotter type for pen plots. Current options are: HP — [for the HP-7470A pen plotter (default)], and SPerry — [for the Sperry Model 37 laser printer]. PS — [for the PostScript(R) laser printer] Hint: The route (**SET OUTPUT = p)** for the pen plots is not saved as part of the save file, nor are the plotter type and character set code. This was done intentionally so that save files may be transported between machines which are connected to various hardware output devices. "UDEC.INI" may then be used to set these options which are appropriate for the specific site.
Xywrite	ON	sets Xywrite output format on.
	OFF	turns off Xywrite output format.

Window <xl xu yl yu>
<AUTO>

This command creates an imaginary window on the screen or pen plotter for the purposes of plotting. The region of space xl to xu, yl to yu is mapped onto the square screen area. Hence, if the window region is not square, a distorted picture will be drawn in which the x and y scales are different. If a window command is not given, or if the keyword **AUTO** is used, the default scales a square window with dimensions slightly larger than the maximum and minimum grid dimensions. If the window is less than the grid dimensions, the screen image will be clipped at the window boundaries. The user may use this feature to obtain enlarged views of detail at points of interest.

5.0 PROBLEM SOLVING WITH UDEC

5.1 Introduction

To run a simulation with UDEC, the problem geometry, boundary conditions and material constitutive relations and properties must first be specified. These procedures are similar to those of nearly all stress analysis codes. The region to be modeled must be subdivided into blocks defined by discontinuities. Blocks may be further discretized into constant strain finite difference triangles to model deformation of individual blocks. Each zone is defined by its three corner nodes, or gridpoints. Generation of triangular zones is done automatically, with the size of zones specified by the user. Blocks which have been internally discretized are called fully-deformable blocks.

Once the problem geometry has been defined, boundary and initial conditions must be provided. The conditions can consist of:

(1) fixing block centroids, gridpoint displacements, or velocities in the x- and/or y-directions;

(2) applying pressure on any outer boundary;

(3) applying x- and/or y-direction forces at any block centroid or boundary gridpoint;

(4) initial stresses in the body; and

(5) gravity.

The actual solution of a problem using an explicit code such as UDEC differs from many of the commonplace implicit codes. Explicit codes are often termed "time-marching" in that the basic equations of motion are solved at successive timesteps until equilibrium or a steady-state condition is achieved. In practical terms, this means that, for the user, the problem is solved in a physically-meaningful manner.

The user is responsible for judging when equilibrium or a steady-state condition has been achieved. An effective way to do this is to monitor, using the **HISTORY** command, changes in problem variables (i.e., stress, velocity, displacement) during execution. The general problem-solving procedure and suggested command order are given in Section 4.2.

The general solution procedure involves problem set-up (defining geometry, material constitutive relations, parameter properties, boundary conditions and in-situ stresses) and then the successive changing of conditions of the problem, such as excavating material, changing boundary conditions, etc. Following each problem step (changes to one or more conditions), the problem is cycled to equilibrium or a steady-state condition. This procedure is convenient because it physically represents the processes which occur in the natural environment. Modeling of a few simple problems, such as those contained in the manual, is suggested to familiarize the user with the solution process.

5.2 Limitations

5.2.1 Cracks Which Partially Penetrate a Block — UDEC was not designed to consider problems involving blocks with partially penetrating cracks (Fig. 5-1). These cracks may exist during problem set-up, when block geometries are being specified. If no other cracks intersect a partially-penetrating crack, then the crack will be deleted, either prior to execution or at the time the block is internally discretized into finite difference triangles.

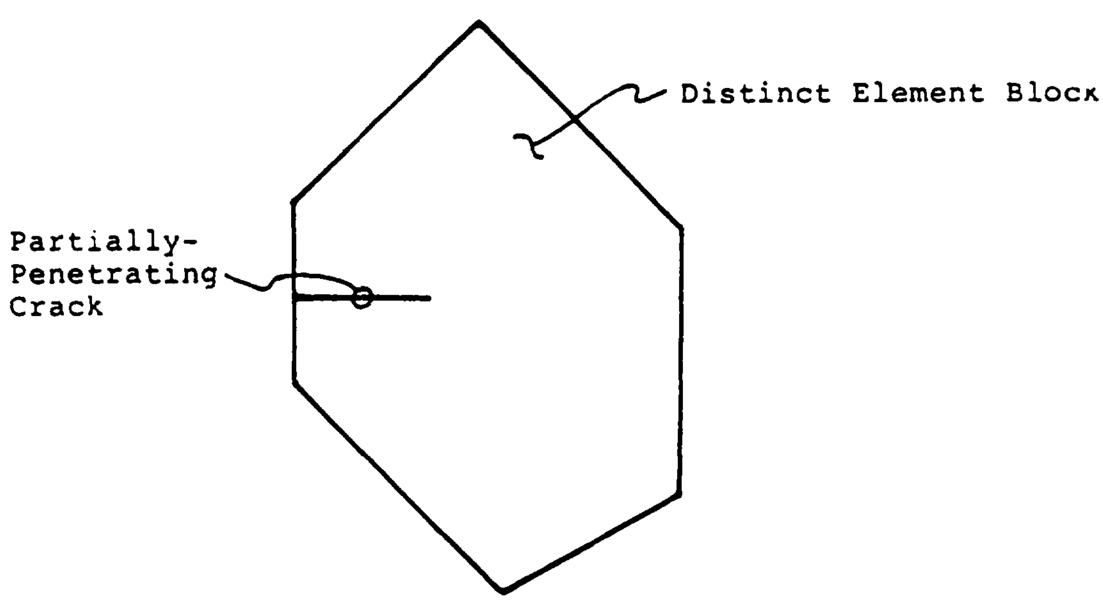


Fig. 5-1 Block with Partially Penetrating Crack

5.2.2 Disconnected Blocks — The data structure of UDEC was designed specifically to model compact rock masses (i.e., blocks tightly packed). The code can handle isolated cases where blocks lose contact with their neighbors by using "virtual contacts", which are fictitious links between particles. However, the code is not designed for multiple virtual contacts; in this case, it becomes inefficient. It is suggested that these disconnected blocks be deleted if they are of no further interest.

5.2.3 Problem Size — The numerical problem size primarily is a function of the number of blocks, contacts, and zones (if any). There is no unique way to specify the allowable problem size; however, it is possible to gain an idea of how large the problem is in relation to allowable size by using the **PRINT MAX** command. The allowable problem size (i.e., memory) is given by the parameter MTOP. The highest memory address used at the current state is given by the parameter MFREE. For example, if MTOP is 100,000 and MFREE is 99,000, then almost all of the allowable memory has been used and only a small amount is available for storage of additional information. As a rough guide, with MTOP=100000, the maximum size problem that can be run is approximately 400 rigid blocks or 225 fully-deformable blocks (2 zones per block). The procedure for increasing the problem size in UDEC is described in Section 6. It should be noted that histories are also stored in memory, so that frequent storage of numerous variables specified by the **HISTORY** command may cause memory capacity to be exceeded. One memory location is required for each **HISTORY** variable at each time it is recorded.

5.2.4 Simply-Deformable (SDEF) Blocks — As pointed out in Section 3.2, simply-deformable blocks use three modes of internal deformation, corresponding to the three strains in two dimensions — i.e., ε_{11}, ε_{22}, and $\varepsilon_{12} + \varepsilon_{21}$. These modes are shown in Fig. 5-2.

In addition to these modes, there remain the three rigid-body modes considered by the original block program (two translational and one rotational degrees of freedom), making six degrees of freedom per block in total.

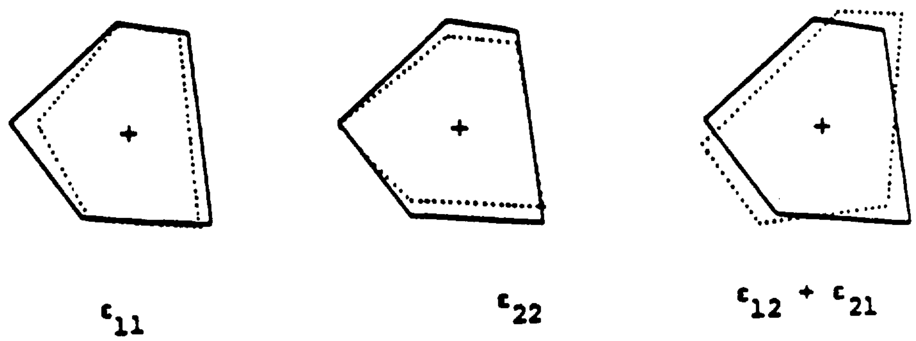


Fig. 5-2 Three Independent Modes of Deformation Used in UDEC

However, some deformation modes, such as those shown in Fig. 5-3, cannot be represented by simply-deformable blocks. A more detailed description of the limitations of SDEF blocks is given by Williams and Mustoe (1987).

To overcome inherent limitations of SDEF blocks, it is often possible to substitute fully-deformable blocks with a minimum number of internal finite difference zones. For quadrilateral blocks with two zones per block, the increased storage and computing time is minimal when compared to simply-deformable blocks.

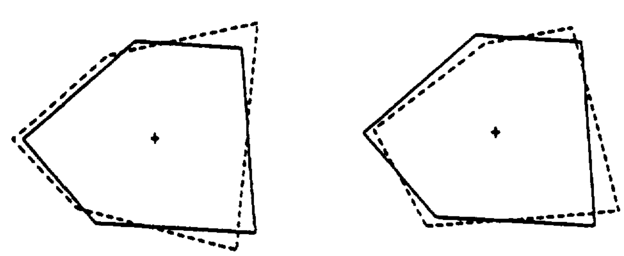


Fig. 5-3 Deformation Modes Not Possible with SDEF Blocks

5.3 Contact Overlap

The error message "Contact Overlap Too Great" can occur during cycling if one block penetrates too far into another. The maximum amount of overlap allowed by the code is one-half the rounding length. If such an error occurs, it is usually necessary to restart the problem from an earlier state. However, before restarting, it is important to identify the cause of the error and correct it. Useful information concerning the contact location(s) where overlap occurred is given preceding the contact overlap message. The following possible causes for contact overlap can be identified.

Joint Normal Stiffness Too Low

If the joint normal stiffness is unrealistically low for the loads applied, the blocks will penetrate too far into each other. This cause can often be identified by (a) plotting a close-up of the affected area, or (b) printing contacts in the same area. The remedy for this cause is simply to increase the normal stiffness. The currently specified problem properties can be seen by typing **PRINT PROPERTIES**.

Numerical Instability

Numerical instabilities, characterized by increasing amplitude of oscillations, result from timesteps which are too large. History plots which show wild fluctuations are indicative of a numerical instability. The only way to correct a numerical instability without changing other problem parameters is to reduce the timestep by using the **FRAC** command. Increasing mass damping parameters can often hide instability but will not likely eliminate it. UDEC automatically determines a timestep which is stable for most cases; however, situations may arise when this timestep is too large. Situations which have been identified as causing numerical instability are:

(1) use of stiffness proportional damping which is too great at high frequencies;

(2) use of high values of joint dilation;

(3) use of problem geometries in which one block contacts many (more than 3) other blocks on one side; and

(4) use of non-reflecting (i.e., viscous) boundaries in which the bounding material is significantly stiffer than the material in the problem domain.

If the cause of the contact overlap cannot be identified, it may be necessary to use the **SET CSCAN** command. This command causes the location of the center of rounded corners to be updated more frequently, resulting in more accurate calculation of contacts and overlap.

Another procedure for eliminating this error is to increase the rounding length at the start of the problem. This procedure is useful if the original rounding length was very small and/or the problem geometry involved blocks with very acute angles.

5.4 Saving/Restoring Runs

UDEC allows the user to save a state and then restore it at any point in the simulation process. A save file is created by typing the command:

> save filename

where "filename" is the name you wish to give the save file. To restore a saved file, simply type the command

> restart filename

where, again, "filename" is the name of the previously-saved state. It is always a good idea to make liberal use of save files for purposes such as parameter studies, plotting, and printing.

5.5 Suggestions and Advice

— UDEC uses constant-strain zones in fully-deformable blocks. If the strain gradient is high, many zones may be needed to represent the non-uniform strain distribution. Try running the same problem with more zones, to check. Constant-strain zones are used because, for plastic flow, it is better to use many low-order elements than a few high-order elements.

— For a new problem, always do a trial run with a few blocks (zones) to get a quick feel for the response and possible difficulties. When you understand the trial results, increase the number of blocks (zones) to obtain better accuracy.

— When generating complex joint patterns, the **CYCLE=0** command can be used to view contact locations which will exist at the time execution begins. A plot of contact locations can be obtained using **PLOT CONTACTS**.

— A very stiff loading plate often can be replaced by a series of gridpoints which are given constant velocity or by a fixed rigid block. (Recall that the **FIX** command fixes velocities, not displacements.)

— In order to determine a collapse load, it often is better to do it under "strain-controlled" conditions rather than "stress-controlled" conditions — i.e., apply a constant velocity and measure the reaction forces rather than apply a constant force and measure displacements. A system that collapses becomes difficult to control as the applied load approaches the collapse load. (This is true of a real system as well as a model system.)

— Use symmetry conditions, whenever possible, to save computer memory and run time. For example, if a system is symmetrical about a vertical axis, you can represent the symmetry line as a vertical boundary with the gridpoints fixed in the x-direction (but free in the y-direction).

— Make frequent use of save files. For example, save intermediate states when doing parameter studies.

— Treat a UDEC model just like a physical model. Try to reproduce in a UDEC run the stages that actually would occur in nature. Keep in mind that there is no unique equilibrium state for an inelastic system. There may be many possible states that satisfy equilibrium; the one you get depends on the load path.

— UDEC shows how a system behaves. Make frequent, simple tests to check that you are doing what you think you are doing. For example, if a loading condition and geometry is symmetrical, check that the response is symmetrical or, after making a loading change or other change, execute a few cycles initially (e.g., 5) to verify that the initial response is of the correct sign and in the correct location. You might also do back-of-the-envelope estimates of the expected order of magnitude of stress or displacements and compare them to UDEC output.

— If you apply a violent shock to a system, you will get a violent response. If you do non-physical things to the system, you must expect strange results.

— Critically examine the output before proceeding with the simulations. If, for example, everything is ok except for large velocities in one corner zone, do not go on until you understand the reason. In this case, you might have left a "fixed" grid point free.

— UDEC does not give a "Factor of Safety" directly. If you need a factor of safety, it can be defined for any parameter that you consider important by taking the ratio of the actual value to the value which causes failure. For example,

$$F_{\phi} = \frac{\text{actual friction angle}}{\text{friction angle to cause failure}}$$

$$F_L = \frac{\text{load to cause failure}}{\text{design load}}$$

Note that the larger value is always divided by the smaller value (assuming that the system does not fail under the actual conditions).

— When performing dynamic analyses by applying time-varying velocities or stresses to a boundary which should also be a non-reflective boundary (to represent infinite far-field boundary), it is possible to apply stress and viscous boundary conditions to the same boundary segment (see Verification Problem C). However, it is necessary to double the applied stress to overcome the effect of the viscous boundary. A velocity history may be converted to a stress boundary condition for similar purposes using the formula

$$\sigma = 2(\rho\ C_p)V$$

where σ = applied normal stress,

ρ = intact mass density,

C_p = speed of wave propagation of medium, and

V = input velocity.

Recall that c_p is given by

$$c_p = \left[[K + (4/3)G]/\rho \right]^{1/2}$$

5.6 References

Williams, John R., and Graham G.W. Mustoe. "Modal Methods for the Analysis of Discrete Systems," Computers and Geotechnics, 4, 1-19 (1987).

6.0 PROGRAM GUIDE

The program guide contains the complete contents of all the groups in the data structure. Figures 6-1 through 6-4 schematically show the linkage of these various groups and should assist the user in following through the parameters and data groups. Figure 6-1 shows the "linked list" arrangement of the main data arrays. Figures 6-2, 6-3 and 6-4 illustrate the conventions for pointers and links in the block data, domain data, and contact data arrays, respectively, and Fig. 6-5 shows the structural arrangement of redundant memory groups. The program guide will assist the user in making any code modifications.

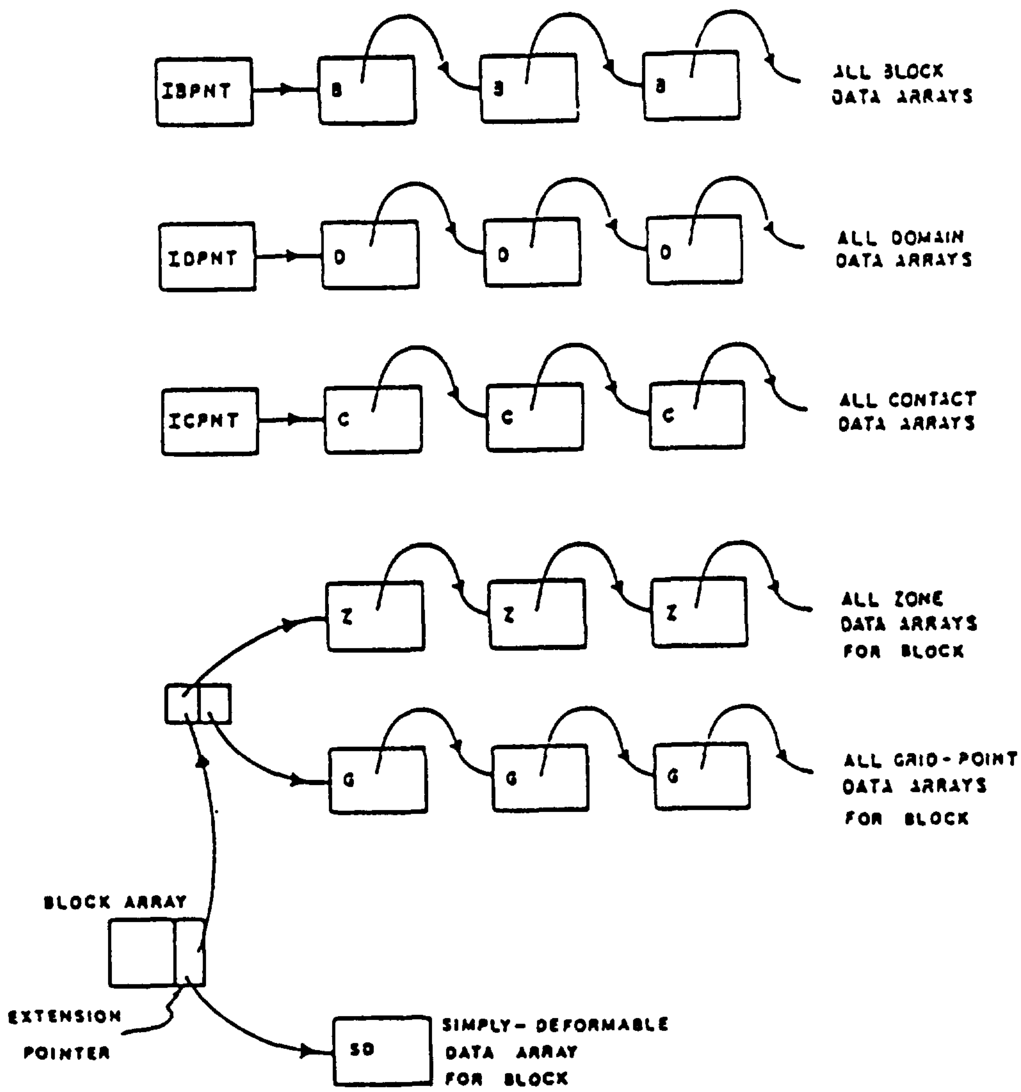


Fig. 6-1 Linked Lists for Main Data Arrays

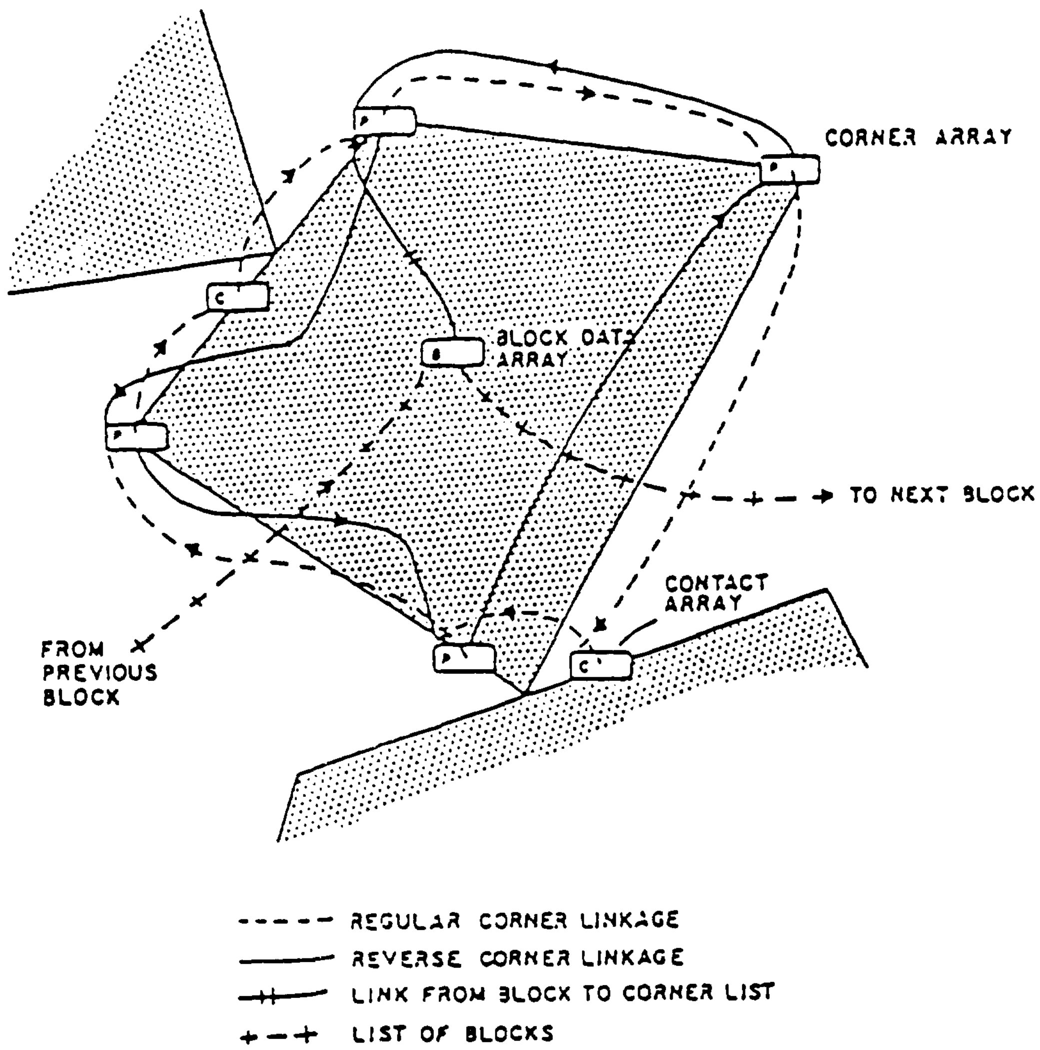


Fig. 6-2 Block Pointers and Reverse Corner Links

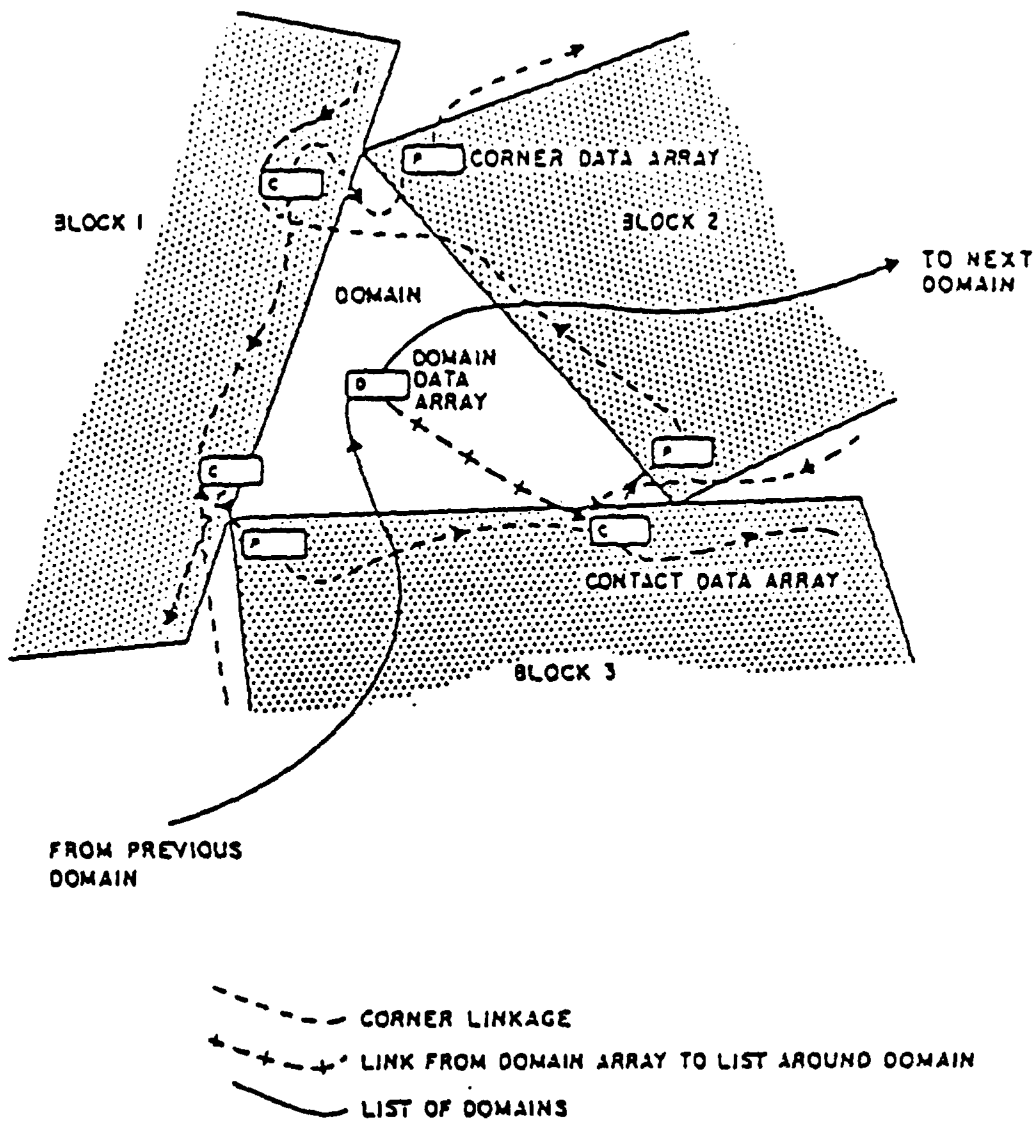
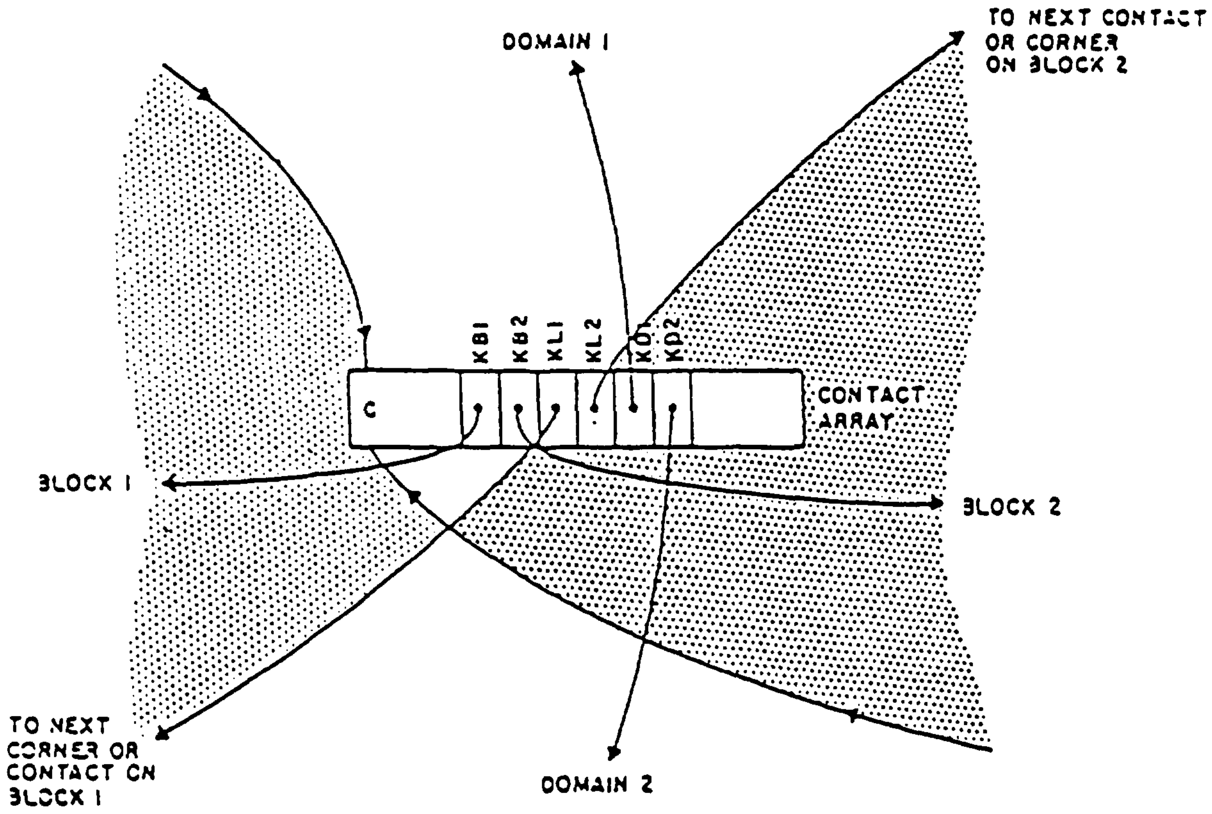


Fig. 6-3 Domain Linkages



NOTE: KB1, KB2 — KD2 REFER TO THE OFFSETS

Fig. 6-4 Convention Used for Pointers Within a Contact Array

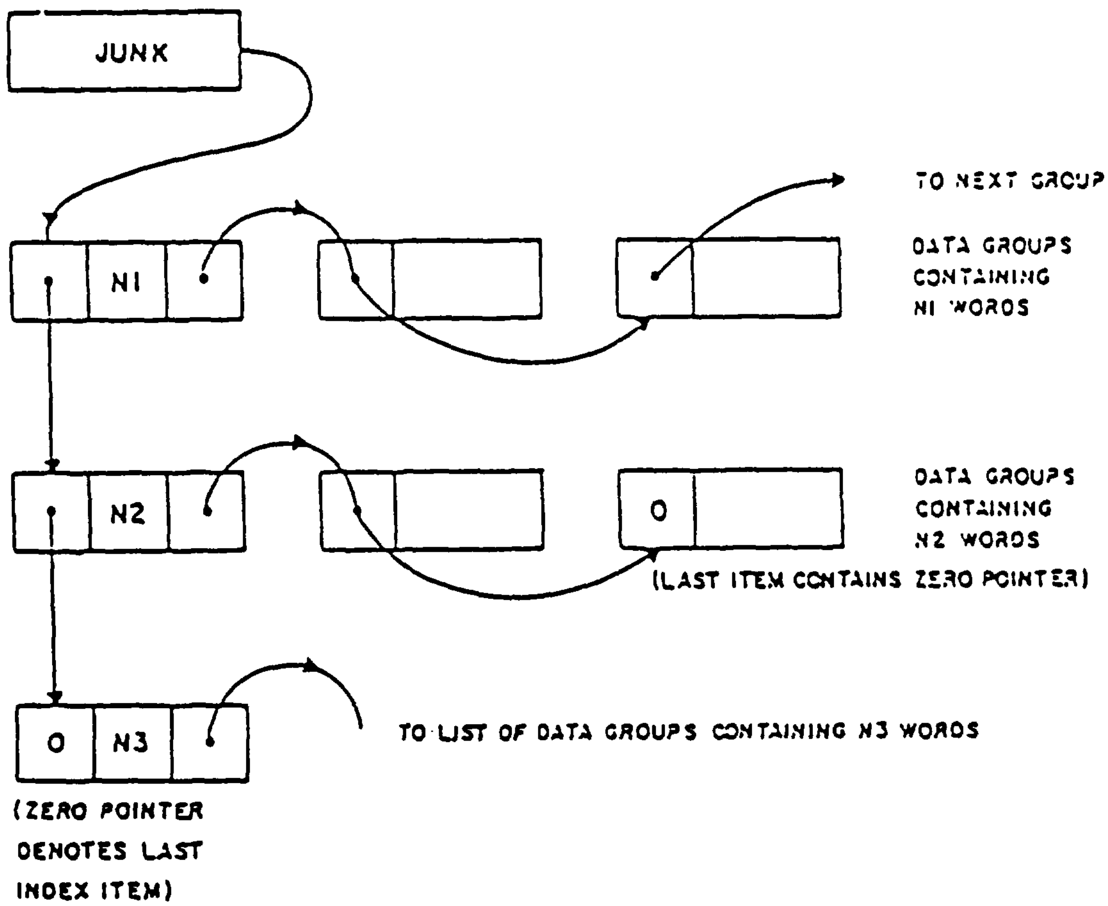


Fig. 6-5 Structure of "Junk List" Holding Redundant Groups of Memory

6.1 Addresses and Offsets

Variables in UDEC are stored in "Linked Lists", which consist of addresses (the first memory location for a particular item — i.e., block or contact) and offsets, which prescribe the memory location relative to the address. Addresses can be found using the **PRINT** command. In most cases, the first integer given is the address. Commands in UDEC allow commonly used real variables to be tracked (using the **HISTORY** command) or changed (using the **CHANGE** or **RSET** commands). However, it is sometimes desirable to track or change less commonly used variables. The commands **HISTORY ADDRESS** and **RSET** are general purpose commands that allow any problem variable to be changed if its address and offset are known. The following sections list the offsets for various data.

Offsets for Block Data Array

Each block data array consists of 23 words (NVBL=23). This array is accessed via the IBPNT pointer.

Offset	Parameter	Description
0	--	block type number
		MRIG=1 rigid block
		MSDEF=2 simply-deformable
		MFDEF=3 fully-deformable
1	KB	pointer to next block in block list
2	KP	pointer on one corner in block's corner list
3	KMAT	material number
4	KCONS	constitutive number
5	KBCOD	code number:
		0 free block
		1 fixed block
7	KX	x-coordinate of centroid
8	KY	y-coordinate of centroid
9	KXD	x-velocity
10	KYD	y-velocity
11	KTD	angular velocity (counterclockwise positive)
12	KAREA	block area
13	KBM	block mass
14	KBI	moment of inertia
15	KBDSF	density scaling factor
16	KBFX	x-centroid force-sum
17	KBFY	y-centroid force-sum
18	KBFT	centroid moment sum
19	KXL	x-load applied to block centroid
20	KYL	y-load applied to block centroid
20	KBEX	extension pointer (to SDEF or FDEF data)
22	KBUMAX	maximum block displacement (not used)
22	KBPOR	code number
		0 non-porous block
		1 porous block

Offsets for Corner Data Array

Each corner data array consists of 13 words (NVCR=13). This array is accessed from the block pointer KP.

Offset	Parameter	Description
1	KL	pointer to next corner or contact on block, in clockwise direction
2	KR	pointer to next corner in counterclockwise direction
3	KNB	pointer to host block
5	KXP	x-coordinate of corner
6	KYP	y-coordinate of corner
7	KXCP	x-coordinate of local circle center
8	KYCP	y-coordinate of local circle center
9	KRAD	radius of local circle
10	KXDP	x-velocity of corner
11	KYDP	y-velocity of corner
11	KGP	pointer to corresponding grid point if block is fully-deformable
12	KBDEX	extension pointer (to boundary corner array)

Offsets for Contact Data Array

Each contact data array consists of 32 words (NVCN=32). This array is accessed via the ICPNT pointer.

Offset	Parameter	Description
0	MCON	=5, denotes contact
1	KC	pointer to next contact in contact list
2	KB1	address of first block involved in contact
3	KB2	address of second block involved in contact
4	KL1	pointer to next item in clockwise list of block corresponding to KB1
5	KL2	same as KL1 but for block KB2
6	KD1	address of domain to left of contact going from block KB1 to KB2
7	KD2	address of domain to right of contact going from block KB2 to KB1
8	KCM	material type number
9	KCC	constitutive number
11	KXC	x-contact coordinate
12	KYC	y-contact coordinate
13	KXDC	relative x-velocity (of block KB2 relative to block KB1)
14	KYDC	relative y-velocity
15	KCS	relative shear displacement
16	KCN	relative normal displacement
17	KCFS	shear force
18	KCFN	normal force (compression positive)
18	KCCOD	code number: 1 corner/corner contact 2 corner/edge contact (KB1...corner, KB2...edge) 3 edge/corner contact (KB1...edge, KB2...corner)
20	KCAP	mean aperture for joint
21	KCQ	flow-rate across joint or contact
22	KCL	length associated with joint
23	KGAM	bounding friction angle (JCONS=3)

Offsets for Contact Data Array (continued)

Offset	Parameter	Description
24	KCNX	contact normal cosines
25	KCNY	contact normal cosines
26	KCRAT1	interpolation factor
27	KCRAT2	interpolation factor
27	KC1A	pointer to corner in block 1
28	KC2A	pointer to corner in block 2
30	KCJREV	reversal factor (JCONS=3)
31	KCJDS	old displacement increment (JCONS=3)
31	KHEX	extension pointer for other constitutive relations

Offsets for Domain Data Array

Each domain data array consists of 8 words (NVDO=8). This array is accessed via the IDPNT pointer.

Offset	Parameter	Description
0	MDOM	=6, denotes domain
1	KD	pointer to next domain in domain list
3	KDAR	domain area
4	KPP	pore pressure for domain
5	KUMAX	fictitious domain displacement
5	KDLOOP	pointer to one contact in counterclockwise list around domain
6	KDCOD	code number: 0 domain pressure not controlled 1 domain pressure controlled
8	KPPO	old pore pressure

Offsets for Simply-Deformable Block Extension Array

Each simply-deformable block extension array consists of 13 words (NVSD=13). This array is accessed from the block printer KBEX.

Offset	Parameter	Description
1	KED11	strain-rate tensor
2	KED12	strain-rate tensor
3	KED21	strain-rate tensor
4	KED22	strain-rate tensor
5	KSI11	internal stress tensor
6	KSI12	internal stress tensor
7	KSI21	internal stress tensor
8	KSI22	internal stress tensor
9	KSA11	applied stress tensor (multiplied by block area)
10	KSA12	applied stress tensor (multiplied by block area)
11	KSA21	applied stress tensor (multiplied by block area)
12	KSA22	applied stress tensor (multiplied by block area)
12	KBPLAS	plasticity indicator 0 elastic 1 at yield 2 elastic but previously at yield 4 surpassed tension cut-off

Offsets for Gridpoint Data Array

Each gridpoint data array consists of 12 words (NVGP=12). This array is accessed indirectly from the block pointer KBEX (see Fig. 6-1).

Offset	Parameter	Description
0	KG	pointer to next gridpoint in gridpoint alist
1	KCOR	pointer to corresponding block corner
3	KXG	x-coordinate
4	KYG	y-coordinate
5	KXDG	x-velocity
6	KYDG	y-velocity
7	KGFX	x-force sum
8	KGFY	y-force sum
9	KGPM	gridpoint mass
10	KGDSF	density scaling factor
11	KGUX	x-displacement
12	KGUY	y-displacement

Offsets for Zone Data Array

Each zone data array consists of 12 words (NVZO=12). This array is accessed indirectly from the block pointer KBEX (see Fig. 6-1).

Offset	Parameter	Description
0	KZ	pointer to next zone in zone list
1	KZG	start of triple pointer to 3 surrounding grid-points
5	KZS11	stress tensor
6	KZS12	stress tensor
7	KZS22	stress tensor
8	KZM	zone mass
9	KZROT	zone rotation
9	KZLL	pointer to neighboring zone for mixed-discretization calculation
10	KZPLAS	plasticity indicator 0 elastic 1 at yield 2 elastic but previously at yield 4 surpassed tension cut-off
11	KZEX	extension pointer to other constitutive relations

Offsets for Boundary Corner Array

Each boundary corner array consists of 16 words (NVBD=16). This array is accessed via the IBDPNT pointer.

Offset	Parameter	Description
0	KBDC	pointer to next boundary corner
1	KBDCOR	pointer to block corner
2	KBDX	type of boundary condition in the x-direction
3	KBDY	type of boundary condition in the y-direction

Boundary Conditions

MLOAD=1	load boundary
MBEL=2	b.e. boundary
MVISC=3	viscous boundary
MVEL=4	velocity boundary

Offset	Parameter	Description
5	KBDFX	total x-force
6	KBDFY	total y-force
7	KBDFXI	applied x-force increment
8	KBDFYI	applied y-force increment
7	KBDVX	applied x-velocity
8	KBDVY	applied y-velocity
9	KBDXM	x-load multiplier
10	KBDYM	y-load multiplier
10	KBEN	pointer to b.e. node
11	KBE2	pointer to second b.e. node
13	KBEF	b.e load distribution factor
14	KBDT	not used
14	KBDFF	free-field indicator
16	KBDPP	pore pressure
16	KBDPER	permeability condition

0	permeable
1	impermeable

Offsets for Free-Field Data Array

Each free-field data array consists of 17 words (NVFF=17). This array is accessed via the IFFPNT pointer.

Offset	Parameter	Description
0	KBDC	pointer to next boundary corner
1	KFFX	x-coordinate
2	KFFY	y-coordinate
3	KFFXD	x-velocity
4	KFFYD	y-velocity
5	KFFUX	x-displacement
6	KFFUY	y-displacement
7	KFFFX	x-force
8	KFFFY	y-force
9	KFFXX	xx-stress
10	KFFXY	xy-stress
11	KFFYY	yy-stress
12	KFFM	mass
12	KFFMAT	material number
13	KFFCON	constitutive number
14	KFFPL	plasticity indicator
16	KFFXXI	initial xx-stress
17	KFFXYI	initial xy-stress

Offsets for Lumped Mass in Cable Reinforcing

Data for the lumped mass in cable reinforcing is stored in an array identical to the block data array. This array is accessed via the ICMPNT pointer.

Offset	Parameter	Description
0		MRIG=1
1	KB	pointer to next lumped mass point
2	KP	zone in which lumped mass point is located
3	KMAT	grout material number
4		not used
5	KBCOD	steel material number
7	KX	x-coordinate of mass
8	KY	y-coordinate of mass
9	KXD	x-component of mass velocity
10	KYD	y-component of mass velocity
11	KTD	weighting factor 1
12	KAREA	steel volume
13	KBM	mass
14	KBI	weighting factor 2
15	KBDSF	density scaling factor = 1
16	KBFX	x-component of force sum
17	KBFY	y-component of force sum
18	KBFT	weighting factor 3
19	KXL	applied load is x-direction
20	KYL	applied load in y-direction
20	KBEX	extension pointer (to grout annulus shear force). Shear force is stored in an array identical to zone array with shear forces stored in offsets KZS11=5 and KZS22=7.

Offsets for Cable Reinforcing Axial Element

Data for the cable reinforcing axial element is stored in an array identical to the contact data array. This array is accessed via the ICEPNT pointer.

Offset	Parameter	Description
1	KC	pointer to next element
2	KB1	pointer to mass 1
3	KB2	pointer to mass 2
8	FCM	cable material number
11	KXC	original x-coordinate of point 1
12	KYC	original y-coordinate of point 1
13	KXDC	original x-coordinate of point 2
14	KYDC	original y-coordinate of point 2
15	KCS	x-component of displacement end 1
16	KCN	y-component of displacement end 1
17	KCFS	x-component of displacement end 2
18	KCFN	axial force
19	KCAP	y-component of displacement end 2
21	KCL	original element length

Offsets for Local Reinforcement Array

Data for the local reinforcement array is stored in an array identical to the contact data array. The array is accessed via the IREPNT pointer.

Offset	Parameter	Description
0	MCON	=6, denotes local reinforcement
1	KC	pointer to next contact in contact list
2	KB1	address of first block involved in contact
3	KB2	address of second block involved in contact
4	KL1	pointer to next item in clockwise list of block corresponding to KB1
5	KL2	same as KL1 but for block KB2
6	KD1	address of domain to left of contact going from block KB1 to KB2
7	KD2	address of domain to right of contact going from block KB2 to KB1
8	KCM	material type number
9	KCC	constitutive number
11	KXC	x-coordinate
12	KYC	y-coordinate
13	KXDC	relative x-velocity (of block KB2 relative to block KB1)
14	KYDC	relative y-velocity
15	KCS	relative shear displacement on joint
16	KCN	relative normal displacement on joint
17	KCFS	shear force at joint
18	KCFN	normal force at joint
20	KCAP	orientation of bolt relative to joint
21	KCQ	total axial displacement of bolt
22	KCL	axial force on bolt
23	KGAM	shear force on bolt

Offsets for Structural Element Lumped Mass Array

Data for the lumped mass array for structural elements is an array identical to the block data array. This array is accessed via the ILMPNT pointer.

Offset	Parameter	Description
0	--	MRIG=1
1	KB	point to next lumped mass in list
3	KMAT	material number
7	KX	x-coordinate of lumped mass
8	KY	y-coordinate of lumped mass
9	KXD	x-velocity
10	KYD	y-velocity
11	KTD	angular velocity (counterclockwise positive)
12	KAREA	lining thickness
13	KBM	mass
14	KBI	moment of inertia
15	KBDSF	density scaling factor (not used)
16	KBFX	x-force sum
17	KBFY	y-force sum
18	KBFT	moment sum
19	KXL	x-load applied to lumped mass
20	KYL	y-load applied to lumped mass

Offsets for Structural Elements Array

Data for structural elements are stored in an array identical to the contact data array. This array is accessed via the ISTPNT pointer.

Offset	Parameter	Description
1	KC	pointer to next element
2	KB1	address of lumped mass 1
3	KB2	address of lumped mass 2
13	KXDC	total moment end 1
14	KYDC	total moment end 2
15	KCS	total shear displacement
16	KCN	total axial displacement
17	KCFS	total shear force
18	KCFN	total axial force
20	KCAP	Young's modulus
21	KCQ	second moment of area
22	KCL	length
23	KGAM	area

Offsets for Structural Element Interface Array

Data for structural element interfaces are stored in an array identical to the contact data array. This array is accessed via the INTPNT pointer.

Offset	Parameter	Description
1	KC	pointer to next interface
2	KB1	address of lumped mass
3	KB2	address of block
5	KL2	pointer to next item in clockwise list of block
8	KCM	material number
9	KCC	constitutive number (=1)
11	KXC	x-coordinate of contact
12	KYC	y-coordinate of contact
13	KXDC	relative x,y velocity (of block relative to lumped mass)
14	KYDC	relative x,y velocity (of block relative to lumped mass)
15	KCS	relative shear displacement
16	KCN	relative normal displacement
17	KCFS	shear force
18	KCFN	normal force

6.2 Main Common Block (UDECOM) Variables

BAFLAG	.TRUE, if batch mode
PPFLAG	.TRUE, if pore-pressure calculation requested
ERFLAG	.TRUE, if an error has occurred
STFLAG	.TRUE, if the first input line has been processed
DCFLAG	.TRUE, if the domain pressure is controlled
ZSFLAG	.TRUE, if zone density scaling is requested
ADFLAG	.TRUE, if auto damping is requested
CRFLAG	.TRUE, if block-splitting calculation is requested
JMPSAV	index of last computed GOTO in MON
NERR	error number
JUNK	pointer to list of spare memory groups
MFREE	first unused memory address
IBLOCK	current block number
IDOM	current domain number
ISTACK	stack pointer
NCYC	currently requested number of cycles
NCTOT	total number of cycles
TDEL	timestep
FRAC	requested fraction of critical timestep for blocks
IROUTE	routing number, used in main routine
NLINE	output line count
NPAGE	output page count
JMPGEN	routing number for continuation line in GEN
ALPHA	mass damping coefficient
BETA	stiffness damping coefficient
CON1	damping factor (1.0-(ALPHA*TDEL/2.0))
CON2	damping factor (1.0/(1.0+ALPHA*TDEL/2.0))
BDT	BETA/TDEL
ALPB	internal mass damping coefficient for simply-deformable blocks
C1B	damping factor (1.0-(ALPB*TDEL/2.0))
C2B	damping factor (1.0/(1.0+ALPB*TDEL/2.0))
DEGRAD	PI/180
PI	3.14159
DAMIN	minimum domain area allowed for fluid calculation
ZCMASS	zone mass for density scaling
BCMASS	block mass for density scaling
ATOL	distance between particles at which a contact is first formed
BTOL	distance between particles at which a contact is broken
CTOL	maximum (negative) overlap allowed when forming contacts

Main Common Block (UDECOM) Variables (continued)

DTOL	rounding length
DTOL2	DTOL/2.0 (maximum contact overlap)
ETOL	limit on maximum domain displacement to trigger contact update
FTOL	total area of blocks for setting plotting scale factor
GTOL	not used
HTOL	half of minimum edge length
IBPNT	pointer to list of blocks
ICPNT	pointer to list of contacts
IDPNT	pointer to list of domains
IODPNT	pointer to outer domain
AKN(I)	normal contact stiffness, material I
AKS(I)	shear contact stiffness, material I
AMU(I)	contact friction coefficient, material I
COH(I)	contact cohesion, material I
DENS(I)	density, material I
AKNJ(I)	joint normal stiffness, material I
AKSJ(I)	joint shear stiffness, material I
AMUJ(I)	joint friction coefficient, material I
COHJ(I)	joint cohesion, material I
PERMJ(I)	joint permeability constant, material I
PERMC(I)	contact permeability constant, material I
AZERO(I)	initial aperture, material I
ARES(I)	residual aperture, material I
BULK(I)	bulk modulus, material I
SHEAR(I)	shear modulus, material I
TFAC(I)	tensile strength factor, material I
ALAM1(I)	Lame constant, material I
ALAM2(I)	Lame constant, material I
GRAVX	x-component of gravitational acceleration
GRAVY	y-component of gravitational acceleration
RHOW	fluid density
BULKW	fluid bulk modulus
IX1	window viewport coordinate
IX2	window viewport coordinate
IY1	window viewport coordinate
IY2	windcw viewport coordinate
RX1	problem window coordinate
RX2	problem window coordinate

Main Common Block (UDECOM) Variables (continued)

RY1	problem window coordinate
RY2	problem window coordinate
VRX1	plotter viewport coordinate
VRX2	plotter viewport coordinate
VRY1	plotter viewport coordinate
VRY2	plotter viewport coordinate
ADFAC	auto damping ratio
GAM	(ALPHA*TDEL)/2
ADMUL	auto damping downward multiplier
ADMUP	auto damping upward multiplier
EKE	total kinetic energy in model
DILAT(I)	contact dilatancy coefficient, material I
DILATJ(I)	joint dilatancy coefficient, material I
TENS(I)	contact tensile strength, material I
TENSJ(I)	joint tensile strength, material I
OTOL	tolerance for contact closure
UJCOS(I)	not available
UJSIN(I)	not available
UJSP(I)	not available
MATUJ(I,J)	not available
ANEL(I,3,3)	not available
IBDPNT	pointer to list of boundary corners
NCINC	number of cycles since last "Cycle" command
TTOT	total time
MATFF	material number of far field
PWVEL	P-wave velocity of far field
SWVEL	S-wave velocity of far field
NBEN	number of b.e. nodes
IKPNT	pointer to b.e. stiffness matrix
IXHIS	type of x-load history
IYHIS	type of y-load history
FXHIS	current value of x-load history
FYHIS	current value of y-load history
IHPNT(I)	pointer to input history array
IHNP(I)	number of points in input history
SHT1(I)	time for first point of input history
SHDT(I)	time increment for input history
PFLOAD	frequency for sine and cosine history
PTLOAD	direction for sine and cosine history

Main Common Block (UDECOM) Variables (continued)

FUFLAG	not available
NUPCON	cycle interval for contact update
NDSCN	cycle interval for domain scan
FRACZ	requested fraction of critical timestep for zones
ZPFLAG	partial mass density scaling flag
ZPSDT	partial mass density scaling timestep
NPLHIS	number of time histories
NPLHCY	increment for time histories
NPLHBL	number of storage blocks for time histories
NPLHTY	history to be typed on the screen
IPLHAD(I)	address of history I
PLHDIS(I)	total displacement (for displacement histories in RIGID or SDEF blocks)
IPLHF(I)	address of history function supplied by user
IPLHBL(I)	address of history data for block I
IPLHNH(I)	number of histories in block I
IPLHNP(I)	number of points in block I
PLHT1(I)	initial time for block I
PLHDT(I)	timestep for block I
XJREG(4)	x-coordinates in JREGION command
YJREG(4)	y-coordinates in JREGION command
RJFLAG	RJOINT flag (.True, if JSET or CRACK is used)
ISJPNT	pointer to stored joints
PPRES	pore pressure for use by POREP command
CRTOL	cycle reversal tolerance for BB model
PPSIOP	pore pressure gradient for use by POREP command
ALNSET(I)	for use with Barton-Bandis Joint Model
AJRCO(I)	for use with Barton-Bandis Joint Model
AJSCO(I)	for use with Barton-Bandis Joint Model
PHIR(I)	for use with Barton-Bandis Joint Model
ALO(I)	for use with Barton-Bandis Joint Model
AKNI(I)	for use with Barton-Bandis Joint Model
UNMAX(I)	for use with Barton-Bandis Joint Model
AINIT(I)	for use with Barton-Bandis Joint Model
SIGMAC(I)	for use with Barton-Bandis Joint Model
UINIT(I)	for use with Barton-Bandis Joint Model
JHFLAG	for use with Barton-Bandis Joint Model

Main Common Block (UDECOM) Variables (continued)

FFFLAG	free-field flag
IFFPNT	free-field pointer
YLFF	free-field lower y-coordinate
YUFF	free-field higher y-coordinate
NPFF	no pointer in ff
IXFF	boundary condition type at ff base
IYFF	boundary condition type at ff base
XDFF	velocity at ff base
YDFF	velocity at ff base
SXYFF	stresses at ff base
SYYFF	stresses at ff base
XFFF	total force at base
YFFF	
KLFFXX	offsets for stresses in lower ff zone
KLFFXY	offsets for stresses in lower ff zone
KLFFYY	offsets for stresses in lower ff zone
KUFFX	offsets for force in upper ff zone
KUFFY	offsets for force in upper ff zone
KUFFXD	offsets for velocity in upper ff zone
KUFFYD	offsets for velocity in upper ff zone
KUFFFX	offsets for force in upper ff zone
KUFFFY	offsets for force in upper ff zone
KUFFM	offset for mass in upper ff zone
SBDLFX	not available
SBDLFY	not available
SBDVFX	not available
SBDVFY	not available
SBDFFX	not available
SBDFFY	not available
YWTAB	y-coordinate of water table

Main Common Block (UDECOM) Variables (continued)

EFFLAG	effective stress flag
PZPRES	zone pressure applied to FDEF blocks
NZPRES	number of zones with fixed pressure
JZPRES(10)	list of zones with fixed pressure
SURFLG	not available
YSURF	not available
HSURF	not available
ROSURF	not available
ANPHI(I)	parameter for CL3 plasticity model
ANPSI(I)	parameter for CL3 plasticity model
CN2(I)	parameter for CL3 plasticity model
PC1(I)	parameter for CL3 plasticity model
PC2(I)	parameter for CL3 plasticity model
AKNJEX(I)	normal stiffness exponent
AKSJEX(I)	shear stiffness exponent
AJR(I)	joint roughness parameter
AJGAM(I)	joint boundary friction angle
NGPEQ	not available
IGPEQ(20,2)	not available
JVFLAG	joint volume flag
ENFLAG	not available
WEKIN	not available
WESTR	not available
WDAMP	not available
WJOINT	not available
WVISC	not available
WLOAD	not available
DWEKIN	not available
DWESTR	not available
DWDAMP	not available
DWJOIN	not available
DWVISC	not available
DWLOAD	not available
DWREL1	not available
DWREL2	not available
FRACW	fraction of fluid flow timestep
JMATDF	default contact material number

JCONDF	default contact constitutive number
CLEMIN	minimum contact length
CLETYP	typical contact length
CAPRAT	maximum contact aperture ratio
EVFLAG	equal volume flag
CDFLAG	contact deletion flag
TUNXC	centerpoint coordinates for circular tunnel
TUNYC	centerpoint coordinates for circular tunnel
TUNRAD	radius for circular tunnel
NTUNPT	number of points on tunnel periphery
IREPNT	pointer to reinforcing elements
ISTPNT	pointer to structural elements
ILMPNT	pointer to lumped mass for structural elements
INTPNT	pointer to interface between structural elements and distinct elements
ICMPNT	pointer to cable bolt lumped mass
ICEPNT	pointer to cable bolt element
NPOH	number of points in output histories
WRFLAG	auxiliary output flag
SFFLAG	steady flow calculation flag
PTOL	tolerance for steady flow calculations
NSFDT	number of fluid flow calculation cycles between mechanical cycles (SFFLAG=.TRUE. only)
FOBMAX	maximum out-of-balance force (FDEF gridpoints only)
FOBTYP	average out-of-balance force
VEGMAX	maximum gridpoint velocity
VEGTYP	average gridpoint velocity
COHW	fluid yield stress (for Bingham fluid flow)
OVTOL	contact overlap tolerance (see SET OVTOL)
HBM(I)	Hoek-Brown parameters used to contour strength stress ratios
HBS(I)	Hoek-Brown parameters used to contour strength stress ratios
UCS(I)	Hoek-Brown parameters used to contour strength stress ratios
NUPC	number of cycles between updates of corner radius centerpoint
THFLAG	.TRUE, if thermal analysis requested
NVBL	number of variables in block data array
NNCR	number of variables in corner data arrag
NVGP	number of variables in gridpoint array
YIELD(I)	tensile yield force in cable bolts
YIELDC(I)	compression yield force in cable bolts
CAPMAX	maximum contact hydraulic aperture
EMPTY( )	empty array
IA(MTOP)	main array

6.3 Boundary Element Common Block (BECOM) Variables

NPOIN	number of boundary element nodes
NDOFN	number of boundary element degrees of freedom
NBELM	number of boundary elements
NGAUS	number of interpretation points
IHALF	half-space solution indicator
IBEFIX	fixed point indicator
XFIX1	fixed points coordinates
YFIX1	fixed points coordinates
XFIX2	fixed points coordinates
YFIX2	fixed points coordinates
AXFIX	fixed point translation
AYFIX	fixed point translation
EMOD	Young's modulus
GMOD	shear modulus
NU	Poisson's ratio
K1	elastic parameters
K2	elastic parameters
NU12	elastic parameters
NU34	elastic parameters
PXX	initial stresses
PYY	initial stresses
PXY	initial stresses
DPXX	initial stresses
DPYY	initial stresses
DPXY	initial stresses
GPF(5)	Gauss interpolation parameters
RWF(5)	Gauss interpolation parameters
SGPF(5)	Gauss interpolation parameters
SHF(5)	Gauss interpolation parameters

Boundary Element Common Block (BECOM Variables) (continued)

XS(150)	nodal coordinates
YS(150)	nodal coordinates
UMUL(150)	load multiplier
TRACS(300)	nodal tractions
DISPS(300)	nodal displacements
UFACX(300)	fixed point multipliers
UFACY(300)	fixed point multipliers

6.4 Logical Unit Numbers (LUN)

LUN	Parameter	Description
0	LUNIF	input file — interactive
0	LUNOF	output file — interactive
2	LUNOF	output print file
3	LUNG	save/restart file
9	LUNP	plotted output
10	LUNF	input file
4		temporary boundary element files
7		temporary boundary element files
8		temporary boundary element files
12		auxiliary output file with information on joint generation and updates
14		formatted history input file
15		unformatted history input file
21	LUNL	log file

6.5 Array Limits

In the present version of UDEC, parameters which are important in controlling the size of problem which can be run are the following:

Parameter	Present Value	Description
MTOP	750,000	size of main array — IA (for DSI-780 with 4 mb)
MBECOM	1,695	size of boundary element common block (dimensioned for 150 nodes)
MCOM	1,180	size of main common block given by MCOM = 350 + 63*NMAT + 3*NPLMAX + 5*NPBLMX
NMAT	10	number of material property groups
NPLMAX	40	number of histories
NPBLMX	20	number of cycle commands which can be issued before histories need to be reset
MCOMC	214	size of common block for thermal analysis given by MCOMC = 34 + 4*NMAT + NTMAX + 5*NTBLMX
NTMAX	20	number of thermal histories
NTBLMX	20	number of **RUN** commands which can be issued before thermal histories need to be reset

In general, if the above parameters, the common blocks, and version number are unchanged, then all "SAVE" files should be able to be "restarted". "SAVE" file sizes are controlled entirely by the variable MFREE, which denotes the highest address used in array IA and the number and length of histories recorded.
Other arrays and parameters of interest include:

Array	Parameter	Present Size	Description
HED		80	problem heading
	NTYP	3	number of block types (rigid, SDEF, FDEF)
	NCONS	10	number of constitutive relations
HISSTR	NPLMAX	40	history titles
THISTR	NPLMAX	40	thermal history titles

7.0 THERMAL-MECHANICAL MODELING

7.1 Introduction

This version of UDEC simulates the transient flow of heat in materials and the subsequent development of thermally-induced stresses. It has the following specific features.

1. Heat flow is modeled as conduction — either isotropic or anisotropic, depending on the user's choice of material properties.

2. Several different thermal boundary conditions may be imposed.

3. Heat sources may be inserted into the material either as line sources or as volume sources. The sources may be made to decay exponentially with time.

4. Both implicit and explicit calculations schemes are available, and the user can switch from one to the other at any time during a run.

7.2 Theoretical Background

7.2.1 Basic Equation — The basic equation of conductive heat transfer is Fourier's law, which can be written in one dimension as

$$Q_x = - k_x \frac{\partial T}{\partial x} \qquad (7\text{-}1)$$

where Q_x = flux in the x-direction (W/m^2), and

k_x = thermal conductivity in the x-direction (W/m °C).

A similar equation can be written for Q_y. Also, for any mass, the change in temperature can be written as

$$\frac{\partial T}{\partial t} = \frac{Q_{net}}{C_p M} \tag{7-2}$$

where Q_{net} = net heat flow into mass (W),

C_p = specific heat (J/kg °C), and

M = mass (kg).

These two equations are the basis of the thermal version of UDEC.

Equation (7-2) can be written as

$$\frac{\partial T}{\partial t} = \frac{1}{C_p \rho} \left[\frac{\partial Q_x}{\partial x} + \frac{\partial Q_y}{\partial y} \right]$$

where ρ is the mass density.

Combining this with Eq. (7-1),

$$\frac{\partial T}{\partial t} = \frac{1}{C_p \rho} \frac{\partial}{\partial x} \left[k_x \frac{\partial T}{\partial x} \right] + \frac{\partial}{\partial y} \left[k_y \frac{\partial T}{\partial y} \right]$$

$$= \frac{1}{\rho C_p} \left[k_x \frac{\partial^2 T}{\partial x^2} + k_y \frac{\partial^2 T}{\partial y^2} \right]$$

if k_x and k_y are constant. This is called the Diffusion Equation.

Temperature changes cause stress changes according to the equation

$$\Delta\sigma_{ij} = -\ \delta_{ij}\ K\ \beta\ \Delta T \tag{T-3}$$

where $\Delta\sigma_{ij}$ = change in stress ij,

δ_{ij} = Kronecker delta function,

$$\begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix}$$

K = bulk modulus,

β = volumetric thermal expansion coefficient, and

ΔT = temperature change.

Note that $\beta = 3\alpha$, where α= linear thermal expansion coefficient.

The mechanical changes can also cause temperature changes as energy is dissipated in the system. This effect is neglected because it is usually negligible.

7.2.2 Implementation in UDEC — UDEC discretizes fully-deformable blocks into triangular zones which are also used for the thermal analysis. Blocks which are not fully deformable are also discretized for the thermal analysis, but better results are usually obtained for fully-deformable blocks.

At each timestep, Eqs. (7-1) and (7-2) are solved numerically, using the following scheme.

1. In each triangle, $(\partial T/\partial x)$ and $(\partial T/\partial y)$ are approximated using the equation

$$\frac{\partial T}{\partial x_i} = \frac{1}{A} \int T n_i \, ds$$

$$\cong \frac{1}{A} \sum_{m=1}^{3} \bar{T}^{m} \varepsilon_{ij} \Delta x_j^{m}$$

where A = area of the triangle,

n_i = i^{th} component of outward normal,

$\bar{T}^{m}$ = average temperature on side m,

Δx_j^{m} = difference in x_j between ends of side m, and

ε_{ij} = two-dimensional permutation tensor,

$$\varepsilon_{ij} = \begin{bmatrix} 0 & 1 \\ -1 & 0 \end{bmatrix}$$

The heat flow into each gridpoint of the triangle is calculated from

$$F_i = A_j Q_i$$

where A_j is the width of the line perpendicular to the component Q_i, as shown in Fig. 7-1:

$$F_{total} = F_x + F_y$$

$$= A_y Q_x + A_x Q_y$$

2. For each gridpoint,

$$\Delta T = \frac{Q_{net}}{C_p M} \Delta t$$

where Q_{net} is the sum of F_{totals} from all zones affecting gridpoint i.

This scheme is explicit, so Δt is limited by numerical stability considerations. An implicit scheme is also available, and is described below.

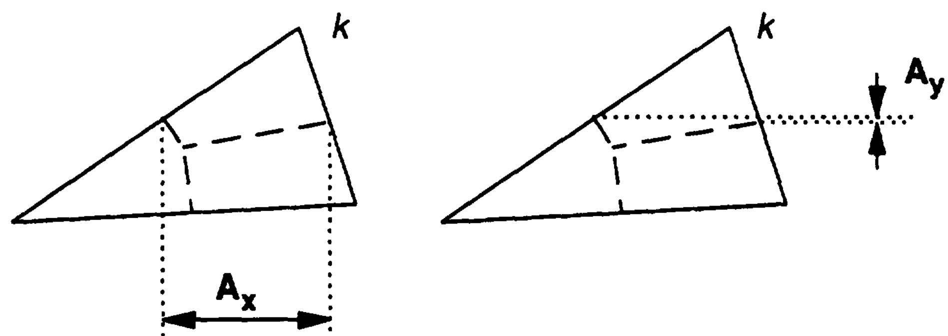


Fig. 7-1 Heat Flow Into Gridpoint k

7.3 Implicit Thermal Logic in UDEC

7.3.1 Introduction — The implicit thermal logic in UDEC uses the Crank-Nicholson method, and the set of equations is solved by an iterative scheme known as the Jacobi method. An implicit method is advantageous for solving linear problems such as heat conduction with constant conductivity, because it allows the use of much larger timesteps than those permitted by an explicit method, particularly at later times in a problem, when temperatures are changing slowly.

7.3.2 Theory — The usual one-dimensional explicit finite difference scheme for heat conduction can be written

$$\frac{\rho C_p}{k} \bullet \frac{T_i(t + \Delta t) - T_i(t)}{\Delta t} = \frac{T_{i+1}(t) - 2T_i(t) + T_{i-1}(t)}{(\Delta x)^2} \qquad (7\text{-}4)$$

An implicit method can be derived by replacing the right-hand side of Eq. (7-4) by the expression:

$$\frac{1}{2}\left[\frac{T_{i+1}(t+\Delta t) - 2T_i(t + \Delta t) + T_{i-1}(t + \Delta t)}{(\Delta x)^2} + \frac{T_{i-1}(t) - 2T_i(t) + T_{i-1}(t)}{(\Delta x)^2}\right]$$

This method, known as the Crank-Nicholson method, has the advantage that it is stable for all values of Δt, but has the disadvantage of being implicit. This means that the temperature change at any point depends on the temperature change at other points. (This can be seen by rewriting the implicit scheme as

$$\frac{\rho c_p}{k\Delta t}\Delta T_i = \left[\frac{T_{i+1} + \frac{1}{2}\Delta T_{i+1} - 2(T_i + \frac{1}{2}\Delta T_i) + T_{i-1} + \frac{1}{2}\Delta T_{i-1}}{(\Delta x)^2}\right]$$

since $T_k(t + \Delta t) = T_k(t) + \Delta T_k$.

The implicit method requires that a set of equations be solved at each timestep for the values of ΔT_i.

In matrix notation, the <u>explicit</u> method can be written as

$$\underline{\Delta T} = \underline{\underline{C}}\ \underline{T}$$

where $\underline{\underline{C}}$ is a coefficient matrix,

$\underline{T}$ is a vector of the temperatures, and

$\underline{\Delta T}$ is a vector of the temperature change.

The <u>implicit scheme</u> can be written as

$$\underline{\Delta T} = \underline{\underline{C}}(\underline{T} + \tfrac{1}{2}\underline{\Delta T})$$

which can be rewritten as

$$(\underline{\underline{I}} - \tfrac{1}{2}\underline{\underline{C}})\underline{\Delta T} = \underline{\underline{C}}\ \underline{T}$$

where we need to solve for $\underline{\Delta T}$ at each timestep.

The matrix

$$(\underline{\underline{I}} - \tfrac{1}{2}\underline{\underline{C}})$$

is diagonally dominant and sparse, because only neighboring points contribute non-zero values to $\underline{\underline{C}}$.

This set of equations is thus efficiently solved by an iterative scheme. For ease of implementation as a simple extension of the explicit method, the Jacobi method is used. For the NxN system $\underline{Ax} = \underline{b}$, this can be written for the n^{th} iteration as

$$x_i(n+1) = \frac{b_i}{a_{ii}} - \sum_{\substack{j=1 \\ j\neq i}}^{N} \left[\frac{a_{ij}}{a_{ii}} x_j(n) \right] \qquad i = 1,2, \ldots N$$

that is,

$$x_i(n+1) = \frac{1}{a_{ii}} \left[b_i - \sum_{j=1}^{N} a_{ij}\, x_j(n) \right] + x_i(n)$$

In our case, this becomes,

$$\Delta T_i(n+1) = \frac{1}{(1-\frac{1}{2}C_{ii})} \left[\sum_{j=1}^{N} C_{ij}\, T_j - \sum_{j=1}^{N} (\delta_{ij} - \tfrac{1}{2}C_{ij})\Delta T_j(n) \right] + \Delta T_i(n)$$

$$= \frac{1}{(1-\frac{1}{2}C_{ii})} \left[\sum_{j=1}^{N} C_{ij}\, T_j + \frac{1}{2} \sum_{j=1}^{N} C_{ij}\, \Delta T_j(n) - \Delta T_i(n) \right] + \Delta T_i(n)$$

This equation shows the analogy between the implicit scheme and the explicit scheme which can be written as

$$\Delta T_i = \sum_{j=1}^{N} C_{ij} \; T_j$$

The amount of calculation required for each timestep is approximately n + 1 times that required for one timestep in the explicit scheme, where n is the number of iterations per timestep. This extra calculation can be more than offset by the much larger timestep permitted by the implicit method. However, the implicit scheme can give poor accuracy because it assumes that the temperature change is a linear function of time in a single timestep, which may not be accurate, especially when temperatures are changing fast, as they generally do near the beginning of a run.

7.4 Thermal-Mechanical Input Instructions and Command Descriptions

The format of the input commands is the same as that of other commands in UDEC. Table 7-1 illustrates examples of consistent sets of units. No conversions are performed by the program.

The rest of this section is divided into two parts — the first describing commands specific to thermal problems and the second describing additions to commands already in UDEC.

Table 7-1

SYSTEM OF UNITS FOR THERMAL PROBLEMS

	METRIC				BRITISH	
Length	m	m	m	cm	ft	in
Density	kg/m^3	$10^3 kg/m^3$	$10^6 kg/m^3$	$10^6 g/cm^3$	$slugs/ft^3$	$snails/in^3$
Stress	Pa	kPa	MPa	bar	lb_f/ft^2	psi
Temperature	K	K	K	K	R	R
Time	s	s	s	s	hr	hr
Specific Heat	J/(kgK)	10^{-3}J/(kgK)	10^{-6}J/(kgK)	10^{-6}cal/(gK)	$(32.17)^{-1}$Btu/(lbR)	$(32.17)^{-1}$Btu/(lbR)
Thermal Conductivity	W/(mK)	W/(mK)	W/(mK)	(cal/s)/(cmK)	(Btu/hr)/(ftR)	(Btu/hr)/(inR)
Thermal Modulus	Pa/K	KPa/K	MPa/K	bar/K	$(lb_f/ft^2)/R$	psi/R
Convective Heat Transfer Coefficient	$W/(m^2K)$	$W/(m^2K)$	$W/(m^2K)$	$(cal/s)/(cm^2K)$	$(Btu/hr)/(ft^2R)$	$(Btu/hr)/(in^2R)$
Radiative Heat Transfer Coefficient	$W/(m^2K^4)$	$W/(m^2K^4)$	$W/(m^2K^4)$	$(cal/s)/cm^2K^4$	$(Btu/hr)/(ft^2R^4)$	$(Btu/hr)/(in^2R^4)$
Flux Strength	W/m	W/m	W/m	(cal/s)/cm	(Btu/hr)/ft	(Btu/hr)/in
Source Strength	W/m^2	W/m^2	W/m^2	$(cal/s)/cm^2$	$(Btu/hr)/ft^2$	$(Btu/hr)/in^2$
Decay Constant	s^{-1}	s^{-1}	s^{-1}	s^{-1}	hr^{-1}	hr^{-1}
Stefan-Boltzmann Constant	5.67	x	$10^{-8} W/m^2K^4$	1.356×10^{-12} $cal/(cm^2sK^4)$	1.713×10^{-9} $Btu/(ft^2hrR^4)$	1.19×10^{-11} $Btu/(in^2hrR^4)$

where 1K = 1.8R

1J = 0.239 cal = 9.48×10^{-4} Btu

1J/kgK = 2.39×10^{-4} Btu/lbR

1W = 1 J/s = 0.239 cal/s = 3.412 Btu/hr

1W/mK = 0.578 Btu/(ft/hrR)

$1W/m^2K = 0.176\ Btu/ft^2hrR$

Note that unless radiation is being used, temperatures may be quoted in the more common units of °C (instead of K) or °F (instead of R).

where $Temp(°C) = \frac{5}{9} * (Temp(°F) - 32)$

Temp(°F) = (1.8 Temp(°C)) + 32

Temp(°C) = Temp(K) - 273

Temp(°F) = Temp(R) - 460

7.4.1 List of Thermal Input Commands

INITEM value xl xu yl yu

The temperature is set to value at all corners and gridpoints in the range xl $\leq$ x $\leq$ xu, yl $\leq$ y $\leq$ yu. Thermal stresses are not induced by this method of setting the temperature.

RUN <<keyword=value>. . .>

This command executes thermal timesteps. Calculation is performed until some limiting condition is reached. The limiting conditions may be the temperature increase at any point, the number of steps, or the simulated age. The limits are changed by the optional keywords listed below. Once a particular limit is specified, it is used for future run commands.

Age = A — problem time (in consistent units)

Delt = Δt — thermal timestep

The thermal timestep is calculated automatically by the program. This parameter allows the user to change the timestep. If the program determines that this value is too large when the explicit scheme is used, it will automatically reduce the timestep to a suitable value when it begins the analysis. The value determined by the program is usually one half the critical value for numerical stability. If the program selects a value which causes instability, this option can be used to reduce the timestep further.

Noage — turns off the previously requested test for exceeding age A. The default for the age parameter is that the age is not tested until an age has been explicitly requested via an "age = value" following a **RUN** command.

Temp = T — maximum total temperature change since the previous mechanical cycles (default x=20)

Step = s — timesteps (default = 500)

RUN (continued)

Two other keywords are available — i.e.,

Implicit	uses the implicit scheme rather than the default explicit scheme.
TOL = tol	points in this tolerance merged for thermal calculations (default = 0.1).

Old limits apply when set or on restart. When a **RUN** command has been completed, the program will indicate which parameter has caused it to terminate. To ensure that it stops for the correct one, the values of the others should be set very high.

The explicit scheme is always used unless the keyword IMPLICIT follows the **RUN** command.

TADD ntab xc,yc ang1,ang2

ntab	table number (between 1 and 10)
xc,yc	co-ordinates of center of arc
ang1,ang2	beginning and ending angles of arc (between -180° and 180°)

Temperatures can be incremented in an angular region using this command. The temperatures are taken from table ntab (see **TABLE** command). The angular region is centered at (xc,yc), and the arc is defined by the angles ang1 and ang2. If a complete circular region is required, the angles should be given as 180 and 180 degrees. The temperatures are taken from table ntab. The x,y pairs in this table represent pairs of radii and temperature increments. The radii represent the distance from (xc,yc) and the code interpolates between these x-values to add to the y-values in the table to the current temperatures. The thermal stresses are also applied, based on these temperature changes.

TFIX value xl xu yl yu

The temperature at all corners and gridpoints in the range $xl<x<xu, yl<y<yu$ is held fixed at value during the simulation. If value is not the current temperature, stresses are induced by the difference between value and the current temperature. (NOTE: by default, all temperatures are free to change initially.)

TFREE xl xu yl yu

The temperature at all corners and gridpoints in the range xl<x<xu,yl<y<yu is allowed to change during the simulation. Note: by default all temperatures are free to change initially.

THAPP <xl xu yl yu> keyword parameter1 parameter2 <region x1,y1 x2,y2 x3,y3 x4,y4>

This command is used to apply thermal boundary conditions and sources in the range xl<x<xu,yl<y<yu or in the region defined by (x1,y1), (x2,y2), (x3,y3), and (x4,y4). The keywords which can be used and the properties associated with parameters 1 and 2 are listed below.

Keyword	Parameter 1	Parameter 2
Convection	convective heat transfer coefficient	temperature of medium to which heat transfer occurs
Flux*	initial strength	decay constant
Radiation	radiative heat transfer coefficient (For black bodies, this is the Stefan-Boltzmann constant)	temperature of medium to which radiation occurs
Source	initial strength	decay constant

*CAUTION: If flux is applied between two blocks, the specified flux is applied to both blocks.

THAPP (continued)

The convection, radiation, and flux commands are used to apply the stated boundary condition between <u>corners</u> in the range xl<x<xu,yl<y<yu. Convection, radiation and flux conditions are meaningful only along lines. The program ensures that this condition is achieved by only applying the boundary condition between two corners in the range (xl<x<xu,yl<y<yu) if they are consecutive corners on the block. This condition is only checked during cycling — so, some corners listed when the boundary conditions are printed out may not be subject to the boundary condition if the next corner either clockwise or anticlockwise on the block is not also listed.

The **SOURCE** keyword results in a volume source of the stated strength in all blocks with <u>centroids</u> in the range xl<x<xu,yl<y<yu.

Consistent units for the various quantities listed above are summarized in Table T-1.

The decay constant in the SOURCE and FLUX options is defined by the equation

$$s_{curr} = s_{ini} * \exp[c_d(t_{curr} - t_{ini})]$$

where s_{curr} = current strength,

s_{ini} = initial strength,

c_d = decay constant,

t_{curr} = current time, and

t_{ini} = initial time (when source was added).

To remove a CONVECTION or RADIATION boundary condition, the same condition should be applied with the heat transfer coefficient of opposite sign.

THAPP (continued)

CAUTION: It is not physically realistic to use negative heat transfer coefficients in any other circumstances.

To remove a FLUX or SOURCE condition, the condition should be applied with the strength replaced by S_{rep}, where

$$S_{rep} = -S_{ini} * \exp[c_d(t_{curr}-t_{ini})]$$

Note that, unless otherwise specified by the **THAPP** command, all boundaries are adiabatic (i.e., insulated).

THIST keyword <keyword> . . .

This command is similar to the **HIST** command. A history may be kept of the temperature at positions in the grid. Temperatures are accessed by their "thermal-history" number, which corresponds to the order in which the histories were requested by the user. The temperature is stored every NTcyc thermal steps. A time history of the temperature may be made at up to twenty points. A maximum of twenty **RUN** command may follow a **THIST** command. For more thermal steps, the history must be reset with the **RESET** command followed by new **THIST** commands. The keywords available are:

LIST prints a list of all thermal history locations.

NTCYC n thermal histories are sampled every n steps. (default is n=10)

TYPE n displays the value of temperature history n on the screen during thermal stepping. Only one point can be selected with this command.

WRITE n fname

write temperature history number n to file fname

Temperature x y

temperature at location nearest to x,y

7.4.2 List of Changes to UDEC Input Commands

PLOT	keyword	
	Keywords which have been added are:	
	Temperature	temperature contours
	THist	n1 <n2. . .>
		temperature histories n1,n2 . . . assigned by **THIST** command.

PRINT keyword

PRop — The new properties relevant to the thermal model (and described under the **PROP** command below) are printed out in addition to the properties printed out by the standard version of UDEC.

THERMAL — Thermal boundary conditions and sources are printed.

THIST — <n1 . . .>

Temperature histories n1,n2, . . ., are printed. If no temperature history number is specified, then a list of all temperature history locations is printed.

PROP Material n keyword v <keyword v>

Keywords which have been added are:

CONd	thermal conductivity
SPecheat	specific heat
THEXP	linear thermal expansion coefficient
XCOnd	thermal conductivity in x-direction
YCOnd	thermal conductivity in y-direction

The actual properties used by the program are the thermal conductivities in the x- and y- directions. The COND keyword simply sets the conductivities in both directions equal to the set value.

RESET keyword

The new keyword THIST has been added. The effect of this command is to set the current values of the thermal histories to zero. This command must be used if more than 20 **RUN** commands follow a **THISTORY** command.

SET keyword = value

Several new keywords have been added.

NMECH — maximum number of mechanical steps executed between thermal steps, when NTHER is non-zero (see below). NOTE: The number of steps executed may be less than NMECH if the out-of-balance force gets low enough or the clock-time is exceeded (default value = 500).

NTHER — number of thermal steps to do before switching to mechanical steps.

NOTE: The default value of NTHER is zero, in which case no interlinking occurs. If NTHER is not zero, the program will switch to mechanical steps every NTHER steps, or when the temperature change parameter (THSOLVE TEM = value) is violated. If the temperature change parameter is violated when NTHER=0, thermal cycling stops, and further thermal or mechanical cycling is user-controlled.

Caution: Geometry changes are ignored by the thermal model until a **RUN** command is given. This means that when the mechanical models are accessed automatically, the geometry changes are ignored on return to thermal steps. If large geometry changes occur, it is better to divide the run into several **RUN** commands rather than having only one.

SET (continued)

THDT

The thermal timestep is set to value.

NOTE: The program calculates the thermal timestep automatically. This keyword allows the user to choose a different timestep. For the explicit method, if the program determines that the chosen step is too small, it will automatically reduce it to a suitable value when thermal steps are taken. It will not revert to a user-selected value until another **SET THDT** command is issued. The program selects a value which is usually one half the critical value for numerical stability. This command has the same effect as a **RUN DELT =** . . . command.

When using the implicit scheme, the thermal timestep is not limited by stability considerations. If the program appears to show numerical instability when solving thermal problems, it may be necessary to reduce the timestep below that selected by the program.

7.5 Solving Thermal-Mechanical Problems

UDEC can be used in the usual way to model the excavation of material, change material properties, and change boundary conditions. The mechanical logic (the standard UDEC program) is also used in the thermomechanical program to take "snapshots" of the mechanical state at appropriate intervals in the development of the transient thermal stresses. This logic is best explained by Fig 7-2.

A difficulty associated with implementing this scheme lies in determining the meaning of "large" in Step 5(b). An advantage of explicit schemes is that the solution is reached in a physically meaningful manner, which is important for non-linear constitutive laws. In order to accomplish this in thermal analyses, the out-of-balance force caused by the temperature changes should not be allowed to adversely affect the accuracy of the solution. If the analysis being performed is linear, no temperature increase will be too great, and UDEC need only equilibrate when the simulation time is such that a solution is required. For non-linear problems, experiment to obtain a feel for what "large" means in the particular problem being solved, by trying different allowable temperature increases on the **RUN** command as follows.

1. Save the mechanical equilibrium state reached by UDEC (so that you can come back and try again).

2. Plot the stresses and shear displacements. If the stresses are near yield, the thermal stresses caused by the temperature changes should not be large. If the stresses are far from yield, larger stresses can be tolerated.

3. Run thermal steps until a particular temperature increase is reported by the program (using a **RUN TEM** =value command).

4. Cycle mechanically to attain equilibrium.

5. Again, plot the stresses and shear displacements. If the area where the stresses are at or near yield is not much larger than at step 2 and the shear displacements are not very different, the temperature increase allowed was acceptable. If the changes are judged to be too great, the run must be repeated with a smaller allowed temperature change.

1. **SETUP**

- define problem geometry
- define material properties
- define thermal properties
- set boundary conditions (thermal and mechanical)
- set initial conditions (thermal and mechanical)
- set any internal conditions, such as heat sources

2. **CYCLE TO EQUILIBRATE MECHANICALLY**

3. **PERFORM ANY DESIRED ALTERATIONS** such as excavations

4. **CYCLE TO EQUILIBRATE MECHANICALLY**

REPEAT steps 3 and 4 until "initial" mechanical state is reached for thermal analysis.

5. **TAKE THERMAL TIMESTEPS** until

   (a) desired time is reached; or

   (b) temperature increases cause "large" out-of-balance forces in blocks.

6. **CYCLE TO EQUILIBRATE MECHANICALLY**

REPEAT steps 5 and 6 until sufficient time has been simulated.

REPEAT steps 3 to 6 as necessary.

Fig. 7-2 General Solution Procedure for Thermal-Mechanical Analysis

An important point to note is that the same temperature increase is not necessarily acceptable for all times in a problem. While the system is far from yield, large temperature increases will be acceptable, but near yield only relatively small increases can be tolerated.

7.5.1 Interlinking of Thermal and Mechanical Steps

For a problem in which the number of thermal steps before mechanical cycling is needed is large, analyses can be accomplished by executing a series of **RUN** and **CYCLE** commands. If the number of thermal steps is small, however, this is impractical. It has therefore been made possible to automatically switch to mechanical steps during a series of thermal steps using the **SET NTHER** and **SET NMECH** commands. If NTHER is set to a non-zero value, mechanical steps will be taken every NTHER steps, or whenever the temperature-change parameter (**RUN TEM** = value) is exceeded.

7.5.2 Modeling Hints

7.5.2.1 Zone Dimensions — UDEC divides fully-deformable blocks into triangular zones for mechanical calculations. For thermal calculations, the same zoning is used, with the exception that the triangles are further subdivided where the block is in contact with a corner on another block (Fig. 7-3).

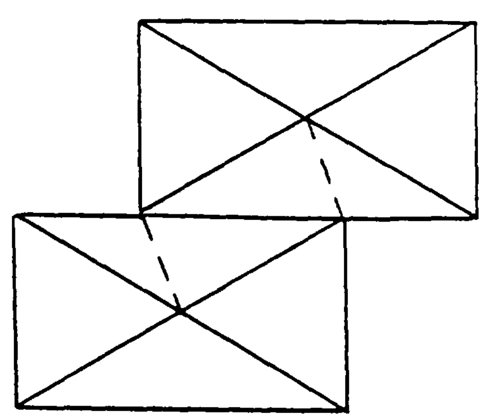


Fig. 7-3 Subdivision of Zones at Contacts

Rigid and simply-deformable blocks are divided into triangles using the centroid as a common vertex of all the triangles, with the other vertices at the corner and at the contact with corners on other blocks (Fig. 7-4).

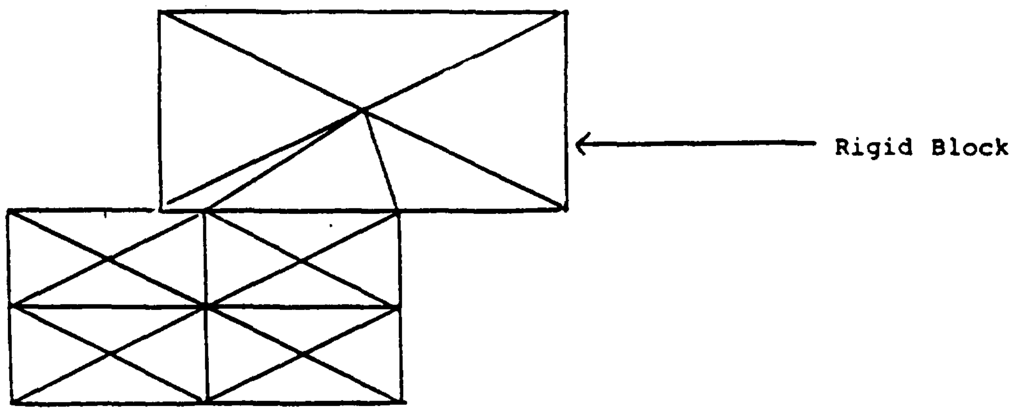


Fig. 7-4 Typical Zoning of Rigid Block

If the schemes outlined above were used without modification, it would be possible for very narrow triangles (such as those shown in Fig. 7-5) to be formed.

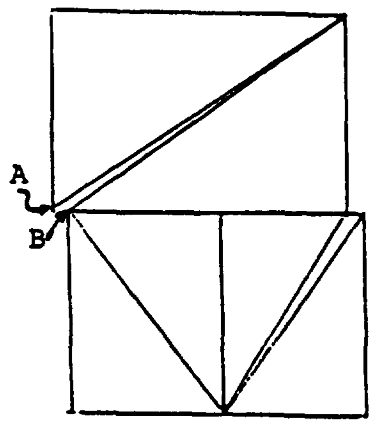


Fig. 7-5 Zones Which May Cause Inaccuracy

This causes inaccuracy and may also lead to extremely small thermal timesteps. To avoid this, it is important to not have blocks with small zones neighboring on blocks with large zones or large rigid blocks. The "thermal tolerance" option on the **RUN** command should also be used to force points such as A and B, in Fig. 7-5, to be treated as one for thermal calculations.

7.5.2.2 Boundary Locations — When modeling an infinite region, it is necessary to truncate the UDEC grid far enough away from the region of interest that the boundaries do not affect the solution. To determine whether the boundaries are far enough away, the following steps should be followed.

1. Let the boundary representing infinity be insulated (the default boundary condition).

2. Solve the problem.

3. Examine the temperature changes on the boundary.

4. If the temperature changes are small, it is safe to assume the boundary has a negligible effect. If the temperature changes are not small, the boundary is probably too close. To confirm this, or disprove it, rerun the problem with the boundary temperatures fixed at their initial values. If the results are significantly different, the boundary was too close.

7.5.2.3 Use of the Implicit Scheme — The advantage of an implicit method is that the timestep is not restricted by numerical stability. The disadvantages are that:

(1) extra memory is required to use this method;

(2) a set of simultaneous equations must be solved at each timestep; and

(3) larger timesteps may introduce inaccuracy.

These disadvantages must be kept in mind when deciding which method to use. They are discussed below.

Memory Requirement — If an attempt is made to use the implicit method for a large problem, an error message will be generated. The only way to avoid this is to run a smaller problem or use the explicit method.

Solving a Set of Equations — The set of equations to be solved at each timestep is solved iteratively. Each iteration of the solution takes about the same length of time as a single step of the explicit method. The number of iterations depends on the timestep chosen and the particular problem being solved, but it is always at least 3. Thus, the implicit scheme offers an advantage over the explicit scheme only if the timestep is much larger than that which the explicit scheme would use. On the other hand, the iterative scheme does introduce some restriction on the timestep. In general, a timestep between 100 and 10,000 times that used by the explicit scheme is satisfactory.

The program displays the iteration counter and a measure of convergence (the residual) to the left of the timestep counter while the implicit scheme is running. The user should check that the number of iterations being taken is such that the implicit scheme is indeed more efficient than the explicit scheme. If not, switch to the explicit scheme or change the timestep. This counter will also indicate if the method is not converging. If the residual is increasing with successive iterations, the method is not converging, and a smaller timestep must be used.

Inaccuracy Due to Large Timesteps — In the initial period of a solution, temperatures generally change much faster than they do later. It therefore is appropriate to use a smaller timestep or, more likely, the explicit method, initially, and then switch to the implicit method with larger timestep later in the solution. Convergence of the solution generally occurs in fewer iterations at later timesteps.

Selecting the Implicit Method — From the above discussion, it can be seen that the implicit method works best when used at late times in the solution period, and only if the timestep can be increased significantly over the one used by the explicit scheme.

7.6 Offsets for Thermal Logic

Offsets for Block Data Array

23	kbtem	block centroid temperature
24	kbdtl	block centroid temperature change in one cycle
25	kbthm	block centroid "thermal mass"
25	kbfix	flag for fixed centroid temperature
27	kbdtem	accumulated block centroid temperature change

Offsets for Gridpoint Data Array

12	kthgf	flag for fixed gridpoint temperature
14	kgtemp	gridpoint temperature
15	kgdteml	gridpoint temperature change in one cycle
16	kthmg	gridpoint "thermal mass"
17	kgdtem	accumulated gridpoint temperature change

Offsets for Corner Data Array

Offset	Parameter	Description
13	kthpf	flag for fixed corner temperature
15	ktemp	corner temperature
16	kdteml	corner temperature change in one cycle
17	kthma	corner "thermal mass"
18	kdtem	accumulated corner temperature change

7.7 Main Thermal Common Block Variables

agete	.TRUE, if age tested on **RUN** command
scetes	.TRUE, if heat source could not be calculated
flxtes	.TRUE, if heat flux could not be calculated
thflg	.TRUE, if mechanical cycles called from thermal cycles
delte	.TRUE, if user sets timestep
ntmax	maximum number of thermal histories
ntblmx	maximum number of storage blocks for thermal histories
spec(i)	specific heat
xcond(i)	conductivity in x-direction
ycond(i)	conductivity in y-direction
thexp(i)	linear thermal expansion coefficient
delt	thermal timestep
eqth1	temperature change limit on **RUN** command
neqth2	number of steps limit on **RUN** command
eqth3	clock time limit on **RUN** command
ntth	total number of thermal steps
temmax	maximum temperature change since last mechanical cycles
timth	thermal time
ithput	pointer to thermal boundary condition list
age	problem time to be tested by **RUN** command
thtol	tolerance within which points are combined
nthis	number of thermal histories
nthcy	increment for thermal histories
nthbl	number of storage blocks for thermal histories
nthty	thermal history to be typed on screen
ithad(i)	address of thermal history i
ithbl(i)	address of thermal history data for block i
ithnh(i)	number of thermal histories in block i
ithnp(i)	number of points in block i
thtl(i)	initial thermal time for block i
thdt(i)	timestep for block i
iccpnt	pointer to temporary list of thermal interactions
icgput	pointer to temporary list of thermal interactions
nther	number of thermal cycles executed before switching to mechanical cycles
nmech	number of mechanical cycles executed when called from thermal cycles

7.8 Thermal Verification Tests

7.8.1 Introduction — This section presents a set of thermal calculations performed with UDEC. The numerical solutions for both steady-state and transient problems are compared with analytical solutions. The problems presented demonstrate the accuracy of the thermal capability in UDEC; the results compare well with the analytical solutions. These tests were run on a DSI-780 coprocessor board; small differences between results may be attributed to differences in machine accuracy. The input files and output files are supplied on a diskette (included with this manual). It is recommended that users repeat these problems on their computers to check computational results.

7.8.2 Case 1: Steady-State Temperature Distribution Along a Tapered Fin

Problem Description:

A very long tapered rectangular steel fin dissipates heat to air, from a wall. The wall temperature is 1100° C, the surrounding air is at 100° C, and the convective coefficient is 15 W/m$^{2\circ}$ C. The fin has a thermal conductivity of 15 W/m° C and tapers from a width of 8.33 cm to 0 cm over a distance of 33.33 cm. Determine the surface temperature distribution along the fin (shown in Fig. 7-6).

The analytical solution for this problem is given by Carslaw and Jaeger (p. 142). The temperatures obtained from UDEC are compared with this solution in Table 7-4, below. The results from UDEC agree well with the analytical solution.

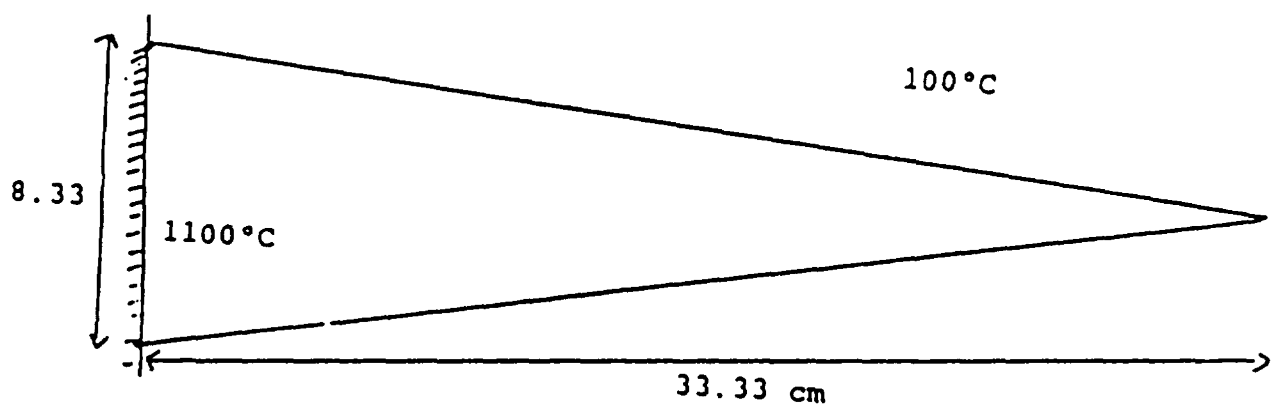


Fig. 7-6 Temperature Distribution Along a Tapered Fin

```
* Case 1 - Tapered Fin - Explicit Solution
*
start
thermal
rou 0.001
block 0,0 -.1,0 -.1,.08333 0,0.0833 0.3333,0.041667
split 0,0 0,1
gen -1 1 -1 1 auto 0.033
prop mat 1 dens 1000 spec 300 cond 15
initem 1100 -1 1 -1 1
tfix 1100 -0.001 0.001 -1 1
thapp reg -.001,-.001 -.001,.001 .3335,.0417 .3335,.0415 convec 15 100
thapp reg -.001,.0832 -.001,.0835 .3335,.0417 .3335,.0415 convec 15 100
initem 700 0.06 0.14 -1 1
initem 500 0.14 0.24 -1 1
initem 400 0.24 0.34 -1 1
run ste 80000 tem 1500 tol 0.001
pr bl
sav case1e.sav
```

```
* Case 1 - Tapered Fin - Implicit Solution
*
start
thermal
rou 0.001
block 0,0 -.1,0 -.1,.08333 0,0.0833 0.3333,0.041667
split 0,0 0,1
gen -1 1 -1 1 auto 0.033
prop mat 1 dens 1000 spec 300 cond 15
initem 1100 -1 1 -1 1
tfix 1100 -0.001 0.001 -1 1
thapp reg -.001,-.001 -.001,.001 .3335,.0417 .3335,.0415 convec 15 100
thapp reg -.001,.0832 -.001,.0835 .3335,.0417 .3335,.0415 convec 15 100
initem 700 0.06 0.14 -1 1
initem 500 0.14 0.24 -1 1
initem 400 0.24 0.34 -1 1
set thdt=10
run ste 100 tem 1500 tol 0.001 impl
pr bl
sav case1i.sav
```

Table 7-2

COMPARISON OF UDEC WITH ANALYTICAL SOLUTION

x	Analytical temperature	Explicit temperature	Explicit Error (%)	Implicit temperature	Implicit Error (%)
0	1100	1100	0.00	1100	0.00
0.04166	940.5	936.5	0.43	936.7	0.40
0.08332	801	797.4	0.45	797.7	0.41
0.125	677.5	675	0.37	675.4	0.31
0.1666	569.8	567.8	0.35	568.2	0.28
0.2083	475.8	474.3	0.32	474.7	0.23
0.25	394.5	393.4	0.28	393.7	0.20
0.2916	324.7	323.6	0.34	323.8	0.28
0.3333	264.8	265	0.08	265.2	0.15

7.8.3 Case 2: One-Dimensional Steady-State Heat Conduction and Convection through a Composite Wall

Problem Description:

A furnace wall consists of two layers, firebrick and insulating brick. The temperature inside the furnace is 3000° C, and the outside temperature is 25° C. The thermal conductivity of the firebrick is 1.6 W/m° C; that of the insulating brick is 0.2 W/m ° C. The convective heat transfer coefficients are 100 W/m^{2}° C in the furnace and 15 W/m^{2}° C on the outside wall. The firebrick is 25 cm thick; the insulating brick is 15 cm thick. Calculate the temperature at both sides of the insulating brick as well as at the inner surface of the firebrick (see Fig. 7-7). The results from UDEC agree well with the analytical solution as shown in Table 7-5.

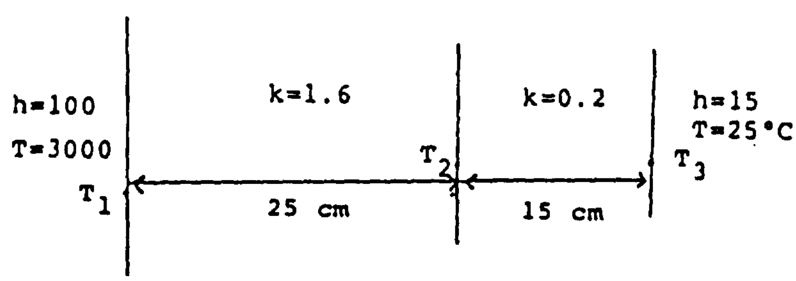


Fig. 7-7 Composite Wall

```
* Case 2 - Furnace Wall - Explicit Solution
*
start
thermal
round 0.001
bl 0,0 0,.025 0.4,.025 .4,0
split .25,-1 .25,1
gen 0,.25 0,.05 auto .05
gen .25,.4 0,.05 auto .025
prop mat 1 dens 10000 spec 300 cond 1.6
prop mat 2 dens 1250 spec 300 cond 0.2
change -1 0.25 -1 1 mat 1
change .25 .4 -1 1 mat 2
thapp -1 0.01 -1 1 conv 100 3000
thapp .39 .41 -1 1 conv 15 25
initem 2500 -.01 .251 -1 1
initem 1000 .249 .41 -1 1
run ste 40000 tem 1e5 tol 1e-4
pr bl
save case2e.dat
```

```
* Case 2 - Furnace Wall - Implicit Solution
*
start
thermal
round 0.001
bl 0,0 0,.025 0.4,.025 .4,0
split .25,-1 .25,1
gen 0,.25 0,.05 auto .05
gen .25,.4 0,.05 auto .025
prop mat 1 dens 10000 spec 300 cond 1.6
prop mat 2 dens 1250 spec 300 cond 0.2
change -1 0.25 -1 1 mat 1
change .25 .4 -1 1 mat 2
thapp -1 0.01 -1 1 conv 100 3000
thapp .39 .41 -1 1 conv 15 25
initem 2500 0 .25 -1 1
initem 1000 .25 .4 -1 1
set thdt = 6000
run ste 100 tem 1e5 tol 1e-4 impl
pr bl
sav case2i.sav
```

Table 7-3

COMPARISON OF UDEC WITH ANALYTICAL SOLUTION

	Analytical	Explicit		Implicit	
	temperature	temperature	error (%)	temperature	error (%)
T1	2969.7	2970	0.01	2970	0.01
T2	2496.8	2496	0.03	2497	0.01
T3	226.8	226.7	0.04	226.6	0.09

7.8.4 Case 3: Thermal Response of a Heat-Generating Slab

Problem Description:

An infinite plate of thickness 1m, initially at a temperature of 60° C, is subjected to a sudden heat generation rate of 40 kW/m^3 and a surface temperature of 32° C. Determine the temperature distribution in the plate after 0.2 sec. The physical properties of the plate are:

Density	500 kg/m^3
Specific Heat	0.2 kJ/kg ° C
Thermal Conductivity	20 W/mdeg ° C

The analytical solution for this problem is given graphically in Schneider (p. 309). The results from UDEC agree well with this solution, as shown in Table 7-5, below.

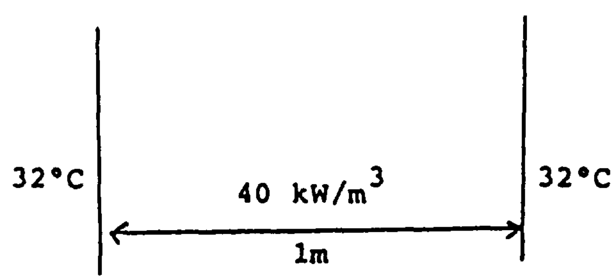


Fig. 7-8 Heat-Generating Slab

```
* Case 3 - Heat Generating Slab - Explicit Solution
start
thermal
rou 0.001
bl 0,0 0,1 1,1 1,0
split 0.5 0 0.5 1
gen -1 2 -1 2 auto 0.15
prop mat=1 dens=500 cond=20 spec=0.2
initem 60 -1 2 -1 2
thapp -1 2 -1 2 source 20000 0
tfix 32 -1 0.001 -1 2
tfix 32 0.999 1.1 -1 2
run ste 250 tem 200 tol 0.0001 delt 8e-4
pr bl
sav case3e.sav
```

```
* Case 3 Heat Generating Slab - Implicit Solution
start
thermal
rou 0.001
bl 0,0 0,1 1,1 1,0
split 0.5 0 0.5 1
gen -1 2 -1 2 auto 0.15
prop mat=1 dens=500 cond=20 spec=0.2
initem 60 -1 2 -1 2
thapp -1 2 -1 2 source 20000 0
tfix 32 -1 0.001 -1 2
tfix 32 0.999 1.1 -1 2
run ste 2 tem 200 tol 0.0001 delt 1e-1 impl
sav case3i.sav
```

Table 7-4

COMPARISON OF UDEC WITH ANALYTICAL SOLUTION

x	Analytical temperature	Explicit temperature	Explicit error (%)	Implicit temperature	Implicit error (%)
0	32	32	0.00	32	0.00
0.125	85	83.97	1.21	82.36	3.11
0.25	113	113.4	0.35	111.7	1.15
0.375	127.8	127.6	0.16	126.2	1.25
0.5	131	131.8	0.61	130.1	0.69

7.8.5 CASE 4: Infinite Slab with Applied Heat Flux

Problem Description:

A heat flux of $1 W/m^2$ is applied to one edge of a semi-infinite slab, initially at 0 degrees. The slab has a conductivity of 1 W/m° C and a diffusivity of 1 m^2/s. The material has a Young's modulus of 1 kPa, a Poisson's ratio of 0.25 and a linear thermal expansion coefficient of 10^{-3} ° C (i.e., a volumetric thermal expansion coefficient of $3x10^{-3}$/° C).

```
K = E/(3*(1-2u))  = 0.667

G = E/(2(1+u))    = 0.4
```

The analytical solutions are given for the temperature and stresses by Carslaw and Jaeger (p. 75) and Timoshenko and Goodier (p. 435), respectively. As shown in Table 7-6, the results from UDEC are in good agreement with these solutions, especially considering the number of significant figures available in the solutions.

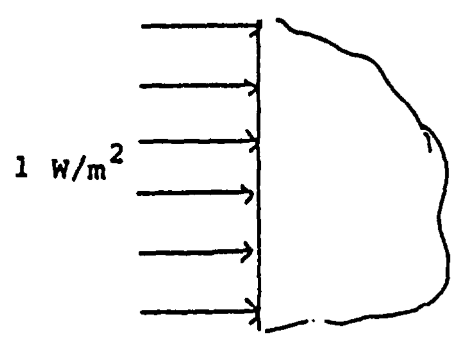


Fig. 7-9 Infinite Slab with Applied Flux

```
* Case 4 - Semi-Infinite Slab - Explicit Solution
start
thermal
round 0.001
bl 0 0 0 0.2 6 0.2 6 0
prop mat=1 bu=0.667 g=0.4 dens 1 cond 1
prop mat=1 spec=1 thexp=1e-3 jkn=1 jks=1
prop mat=1 coh 1e12  ten 1e12
prop mat=1 jcoh 1e12  jten 1e12
split 5 -1 5 1
gen -1 5 -1 1 auto 0.2
thapp -1 0.01 -1 1 flux 1 0
bou -1 6 -1 0.01 yvel 0
bou -1 6 0.19 0.21 yvel 0
fix 5 6 -1 1
his xvel 0 0.1
his xvel 2 0.1
his xvel 5 0.1
his syy 0.1098 0
his syy 0.214 0
his syy 0.411 0
his syy 0.5385 0.1
his syy 0.839 0
his syy 1.036 0
damp auto
run clo 1e10 tem 1e10 ste 4000 delt 5e-5 tol 0.001
save case4_1e.sav
run ste 16000
save case4_2e.sav
res case4_1e.sav
pr bl
c 2000
pr bl
sav case4_1be.sav
res case4_2e.sav
pr bl
c 2000
sav case4_2be.sav
pr bl
```

```
* Case 4 Semi-Infinite Slab - Implicit Solution
start
thermal
round 0.001
bl 0 0 0 0.2 6 0.2 6 0
prop mat=1 bu=0.667 g=0.4 dens 1 cond 1
prop mat=1 spec=1 thexp=1e-3 jkn=1 jks=1
prop mat=1 coh 1e12  ten 1e12
prop mat=1 jcoh 1e12  jten 1e12
split 5 -1 5 1
gen -1 5 -1 1 auto 0.2
thapp -1 0.01 -1 1 flux 1 0
bou -1 6 -1 0.01 yvel 0
bou -1 6 0.19 0.21 yvel 0
fix 5 6 -1 1
his xvel 0 0.1
his xvel 2 0.1
his xvel 5 0.1
his syy 0.1098 0
his syy 0.214 0
his syy 0.411 0
his syy 0.5385 0.1
his syy 0.839 0
his syy 1.036 0
damp auto
run clo 1e10 tem 1e10 ste 40 delt 5e-3 tol 0.001 impl
save case4_1i.sav
run ste 160 impl
save case4_2i.sav
res case4_1i.sav
pr bl
c 2000
pr bl
sav case4_1bi.sav
res case4_2i.sav
pr bl
c 2000
pr bl
sav case4_2bi.sav
```

Table 7-5

COMPARISON OF UDEC WITH ANALYTICAL SOLUTION

TEMPERATURE

t=0.2 sec

x	Analytical temperature	Explicit temperature	Explicit error(%)	Implicit temperature	Implicit error(%)
0	0.5046	0.5014	0.63	0.4999	0.93
0.1563	0.364	0.3616	0.66	0.3601	1.07
0.3125	0.2525	0.2499	1.03	0.2486	1.54
0.4688	0.1687	0.1671	0.95	0.1659	1.66
0.625	0.1078	0.1063	1.39	0.1054	2.23
0.7813	0.0661	0.06554	0.85	0.06483	1.92
1.094	0.0217	0.02154	0.74	0.02118	2.40

t=1.0 sec

x	Analytical temperature	Explicit temperature	Explicit error(%)	Implicit temperature	Implicit error(%)
0	1.1284	1.127	0.12	1.124	0.39
0.1563	0.9795	0.978	0.15	0.9753	0.43
0.3125	0.8436	0.842	0.19	0.8393	0.51
0.4688	0.7216	0.7201	0.21	0.7175	0.57
0.625	0.6123	0.6105	0.29	0.6081	0.69
0.7813	0.5154	0.5143	0.21	0.5119	0.68
1.094	0.3562	0.3556	0.17	0.3536	0.73

Table 7-5

(continued)

YY-STRESS COMPONENT (Syy)

t=0.2 sec

x	Analytical Syy	Explicit Syy	Explicit error(%)	Implicit Syy	Implicit error(%)
0.1098	-0.5369	-0.5312	1.06	-0.5293	a 1.42
0.214	-0.4259	-0.4386	2.98	-0.4367	2.54
0.411	-0.2624	-0.2538	3.28	-0.2522	3.89
0.5385	-0.185	-0.1846	0.22	-0.1833	0.92
0.839	-0.0727	-0.07649	5.21	-0.07562	4.02
1.036	-0.0363	-0.03486	3.97	-0.03433	5.43

YY-STRESS COMPONENT (Syy)

t=1.0 sec

x	Analytical Syy	Explicit Syy	Explicit error(%)	Implicit Syy	Implicit error(%)
0.1098	-1.3629	-1.357	0.43	-1.353	0.73
0.214	-1.2368	-1.252	1.23	-1.248	0.91
0.411	-1.02	-1.007	1.27	-1.003	1.67
0.5385	-0.8951	-0.892	0.35	-0.8884	0.75
0.839	-0.6439	-0.653	1.41	-0.6499	0.93
1.036	-0.5104	-0.502	1.65	-0.4993	2.17

7.8.6 Case 5: Transient Temperature Distribution in an Orthotropic Bar

Problem Description:

A long bar of rectangular cross-section is initially at 500° C. It is suddenly exposed to fluid at 100° C. The thermal conductivity is 20 W/m° C in the x-direction and 3.6036 W/m° C in the y-direction. The bar is 0.3333 m in the x-direction and 0.16666 m in the y-direction. The specific heat is 9.009e-3 J/kg° C, and the density is 400 kg/m^3. The convective heat transfer coefficient is 240 W/m° C. Determine the temperature at points A, B, C, D (see Fig. 7-10) at t=8.3333x10^{-4} secs.

Table 7-7compares the results from UDEC with the solution by Schneider (p. 262). The UDEC results compare very well with the analytical solutions.

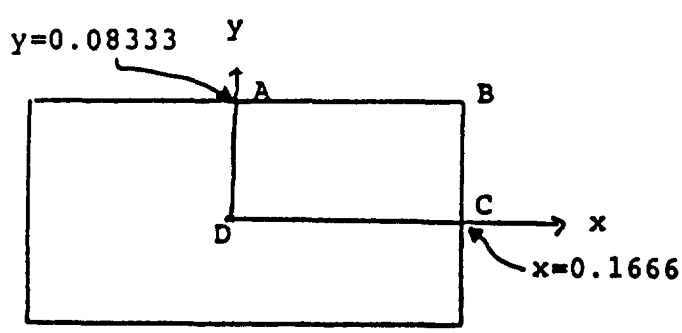


Fig. 7-10 Orthotropic Bar

```
* Case 5 Rectangular Bar - Explicit Solution
start
thermal
round 0.0001
bl 0,0 0,0.083333 0.166667,0.083333 0.166667,0
split 0,0.06 1,0.06
gen -1 1 -1 1 auto 0.02
prop mat=1 dens 400 spe 9.009e-3 xcon 20 ycon 3.6036
initem 500 -1 1 -1 1
thapp -1 1 0.0833 0.0834 convec 240 100
thapp 0.166 0.167 -1 1   convec 240 100
this tem 0 0
this tem 0 1
this tem 1 0
this tem 1 1
run ste 2193 delt 3.8e-7 tem 10000 tol 0.00001
pr bl
sav case5e.sav
```

```
* Case 5 Rectangular Bar - Implicit Solution
start
thermal
round 0.0001
bl 0,0 0,0.083333 0.166667,0.083333 0.166667,0
split 0,0.06 1,0.06
gen -1 1 -1 1 auto 0.02
prop mat=1 dens 400 spe 9.009e-3 xcon 20 ycon 3.6036
initem 500 -1 1 -1 1
thapp -1 1 0.0833 0.0834 convec 240 100
thapp 0.166 0.167 -1 1   convec 240 100
this tem 0 0
this tem 0 1
this tem 1 0
this tem 1 1
run ste 20 delt 4.166667e-5 tem 10000 tol 0.00001 impl
pr bl
sav case5i.sav
```

Table 7-6

COMPARISON OF UDEC WITH ANALYTICAL SOLUTION

x	y	Analytical T (°C)	UDEC T (°C)	Error (%)
0	0	458.7	462.4	0.81
0.1666	0	280.5	284.9	1.57
0	0.083333	198.2	199.7	0.76
0.1666	0.083333	149.4	152.7	2.21

7.9 References

Carslaw, H. S., and J. C. Jaeger. Conduction of Heat in Solids (2nd Ed.). London: Oxford University Press, 1959.

Schneider, P. J. Conduction Heat Transfer. Cambridge, Mass.: Addison-Wesley, 1955.

Timoshenko, S. P., and J. N. Goodier. Theory of Elasticity (3rd Ed.). New York: McGraw-Hill, 1970.

APPENDIX A

VERIFICATION TESTS

Several problems are presented which demonstrate the accuracy of UDEC for a variety of test conditions. These problems verify the different algorithms used in the code to simulate various physical phenomena. The input data files are given in this appendix. The data files and output files are also supplied on a diskette (included with this manual). It is recommended that users repeat these problems on their computers to check computational results. These tests were run on a DSI-780 coprocessor board; small differences between results may be attributed to differences in machine accuracy.

UDEC VERIFICATION TEST A

SLIP INDUCED BY HARMONIC SHEAR WAVE

Two homogeneous, isotropic, semi-infinite elastic media are separated by a plane discontinuity with a limited shear strength. A normally incident plane harmonic shear wave causing slip at the interface originates reflected and transmitted waves.

Figure A-1 displays the numerical model. Two fully-deformable blocks, discretized into finite-difference zones, were used. The interface EF was simulated by a joint with high stiffness.

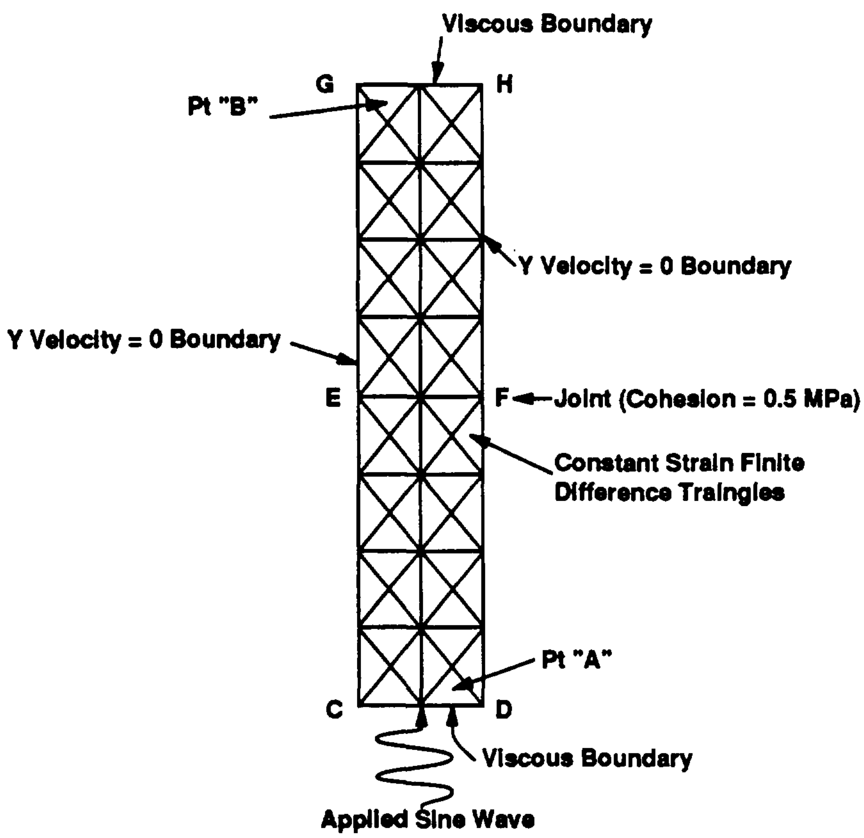


Fig. A-1 Problem Geometry and Boundary Conditions for Problem of Slip Induced by Harmonic Shear Wave

Boundary Conditions:

non-reflecting viscous boundaries at GH and CD

along lateral boundaries GC and DH; vertical motion prevented

Loading Conditions:

shear stresses corresponding to the incident wave applied along CD

Material Conditions:

Blocks — $\rho = 2.65 \times 10^3$ kg/m^3

$K = 16,667$ MPa

$G = 10,000$ MPa

Joints — $K_n = K_s = 10,000$ MPa/m

Cohesion only, zero friction

Figure A-2 displays the time variation of shear stress near points A and B for the case where:

joint cohesion	2.5 MPa
maximum stress of incident wave	1.0 MPa
frequency of incident wave	1 Hz

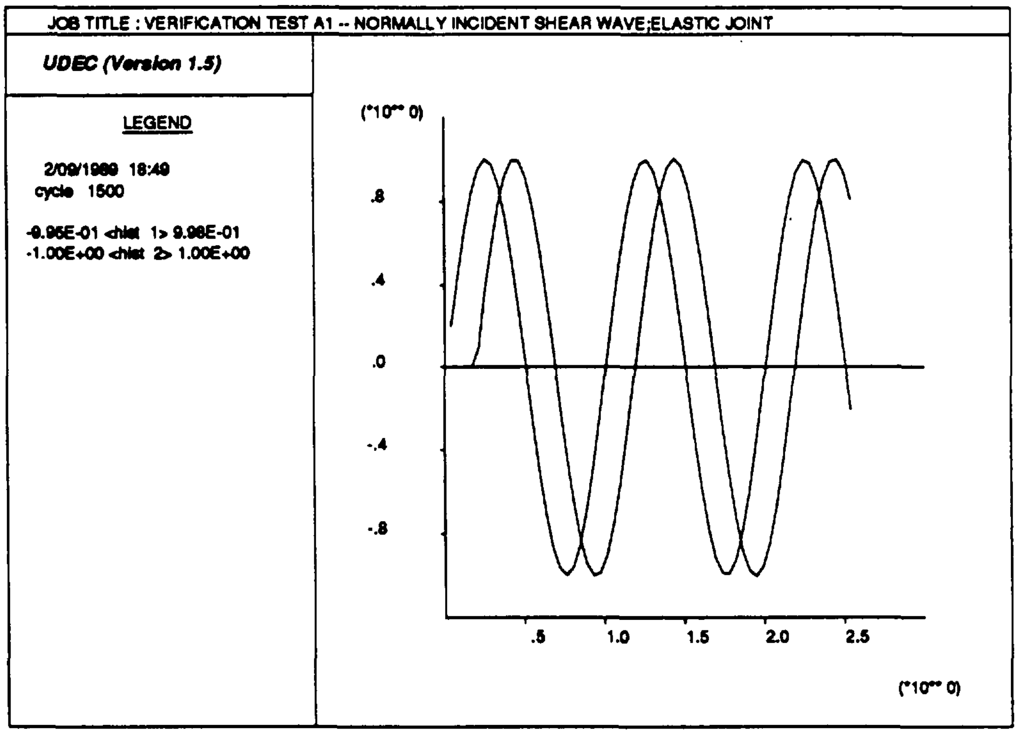


Fig. A-2 Time Variation of Shear Stress at Points A and B for Elastic Joint (i.e., cohesion > 1.0 MPa)

Fig. A-3 displays the time variation of shear stress near points A and B for the same case but with joint cohesion = 0.5 MPa. (Note that shear stress at A is the superposition of incident and reflected wave.)

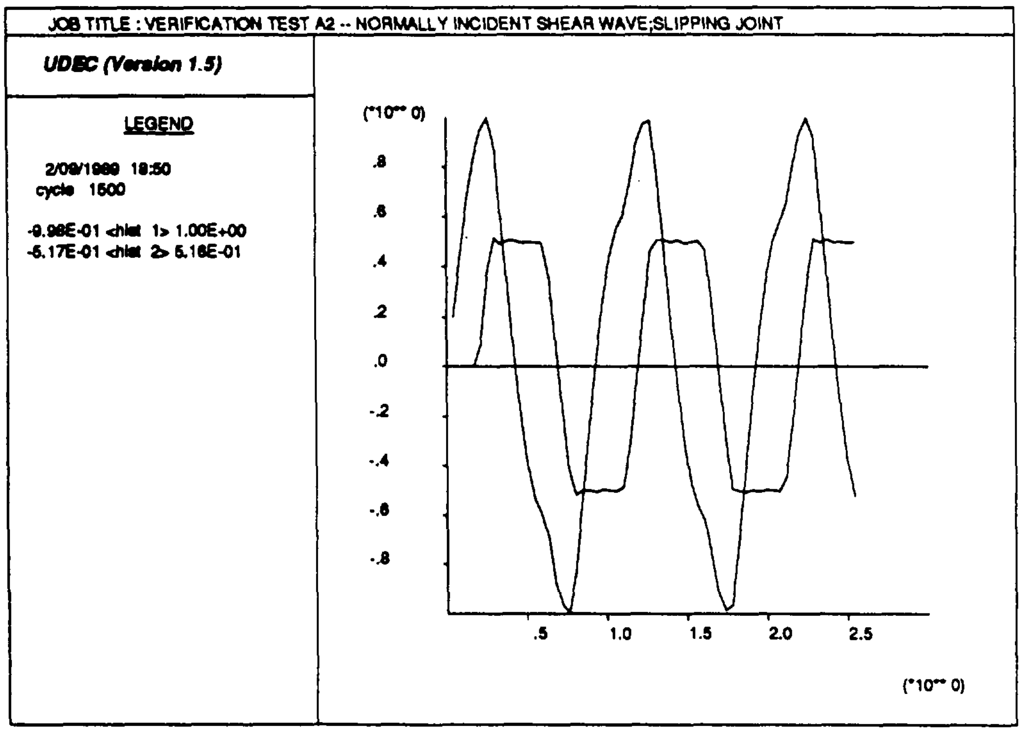


Fig. A-3 Time Variation of Shear Stress at Points A and B for Slipping Joint (i.e., cohesion = 0.5 MPa)

Miller (1978) gives the exact values for the coefficients:

$$R = \left[\frac{E_R}{E_I} \right]^{1/2} \quad \text{and} \quad T = \left[\frac{E_T}{E_I} \right]^{1/2}$$

where E_I, E_T and E_R represent the energy flux per unit area per cycle of oscillation associated with the incident, transmitted, and reflected waves, respectively.

The coefficient

$$A = (1 - R^2 - T^2)^{1/2}$$

is a measure of the energy absorbed at the interface.

The energy flux E_I is given by

$$E_I = \int_{t_1}^{t_1 + T} \sigma_s v_s \, dt$$

where $T = 2\pi/\omega$ is the period of the increment wave,

σ_s = shear stress, and

v_s = shear velocity.

Because the blocks are elastic,

$$\sigma_S = \rho c \; v_S$$

and

$$E_I = \rho c \int_{t_1}^{t_1 + T} v_S^2 \, dt$$

This integral was calculated from the x-velocity history at location (-160, -200) [history no. 3] for the case of no slip. (The integral of integration was the wave period between cycles 999 and 1379; velocities were sampled every 25 cycles.)

E_T was calculated in an analogous way from the x-velocity at (-160, 200) for the case of slip (cohesion = 0.5). E_R was calculated from the x-velocity at (-160, -200) of the reflected wave. This velocity was calculated as the difference of the velocities in the two runs (slip and no slip).

Figure A-4 compares the numerical results with the exact solution for these coefficients for three values of the parameter

$$\omega\gamma U/\tau_S$$

where $\gamma = (\rho G)^{1/2}$,

τ_S = joint cohesion,

U = displacement amplitude of incident wave, and

ω = frequency of incident wave.

The data shown in Fig. A-4 for $\omega\gamma U/\tau_S = 2$ were obtained from the input file listed herein. Data for other values of $\omega\gamma U/\tau_S$ were obtained by changing ω.

Reference

Miller, R. K. "The Effects of Boundary Friction on the Propagation of Elastic Waves," Bull. Seismic. Assoc. America, 68, 987-998 (1978).

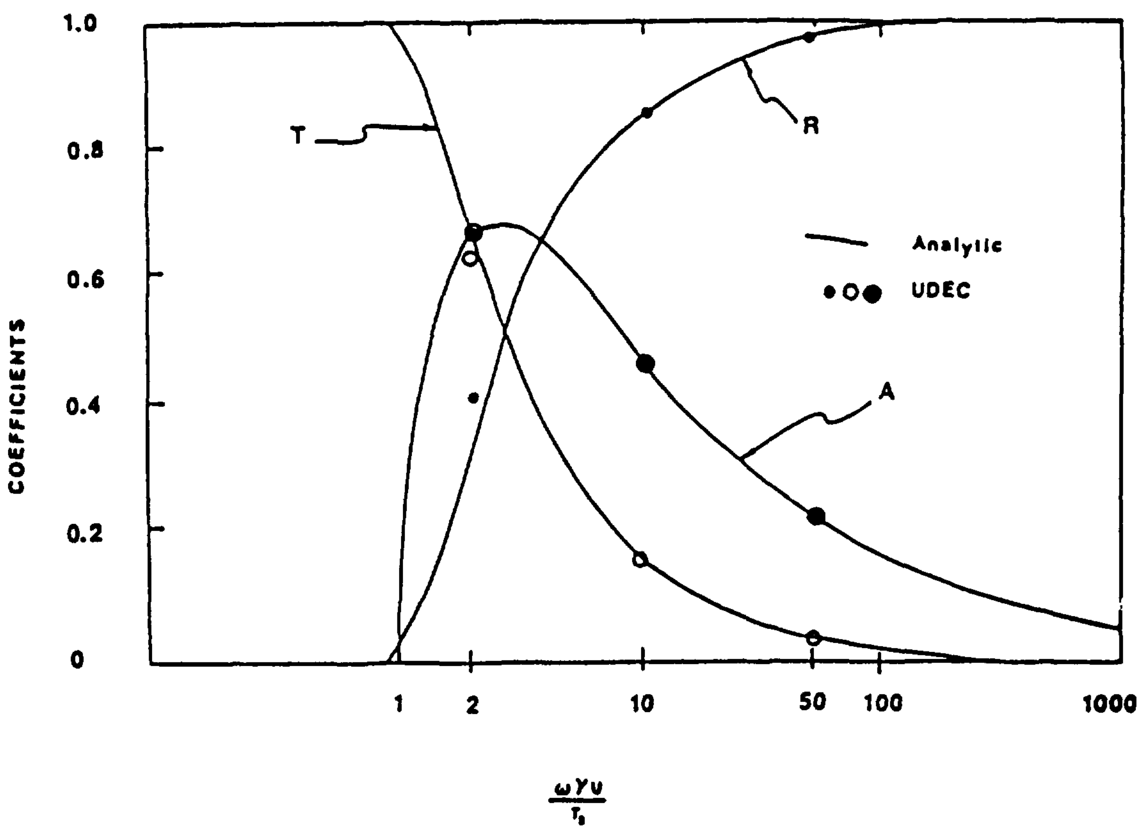


Fig. A-4 Comparison of Transmission, Reflection and Absorption Coefficients

INPUT COMMANDS FOR VERIFICATION TEST A

```
HEAD
TEST A1 -- NORMALLY INCIDENT SHEAR WAVE;ELASTIC JOINT
WIND -400 0 -200 200
PROP JMAT=1 KN=5000 JKN=10000 JKS=10000 JCOH=2.5 JTENS=1E6
PROP MAT=5 D=0.00265 K=16667 G=10000
ROUND 0.1
EDGE=10.0
BLOCK -200 -200 -200 200  -120 200 -120 -200
SPLIT -210,0 201 0
CHANGE MAT=5 CONS=1
GEN -200 200  -200 200 EDGE 60
CHANGE JMAT=1 JCONS=2
BOUND MAT=5
BOUND  -201,201 -201,-199 XVISC YVISC
BOUND -201 201 199 201 XVISC YVISC
BOUND -201 201 -201 -199 STRESS 0,2,0
BOUND -201,-199 -201,201 YVEL=0
BOUND -121,-119 -201 201 YVEL=0
BOUND HIST SINE (1.0,5.0)
FRAC=0.1
HIST N=25 SXY -160,-200 SXY -160,200  TYPE=1
HIST     XVEL(-160,-200) XVEL(-160,200)
INSITU STRESS 0 0 -1E-6
CYC 1500
PR MAX
PR HIST 1 2 3 4
SAVE TESTA1
STOP
HEAD
TEST A2 -- NORMALLY INCIDENT SHEAR WAVE;SLIPPING JOINT
WIND -400 0 -200 200
PROP JMAT=1 KN=5000 JKN=10000 JKS=10000 JCOH=0.5 JTENS=1E6
PROP MAT=5 D=0.00265 K=16667 G=10000
ROUND 0.1
EDGE=10.0
BLOCK -200 -200 -200 200  -120 200 -120 -200
SPLIT -210,0 201 0
CHANGE MAT=5 CONS=1
GEN -200 200  -200 200 EDGE 60
CHANGE JMAT=1 JCONS=2
BOUND MAT=5
BOUND  -201,201 -201,-199 XVISC YVISC
BOUND -201 201 199 201 XVISC YVISC
```

```
BOUND -201 201 -201 -199 STRESS 0,2,0
BOUND -201,-199 -201,201 YVEL=0
BOUND -121,-119 -201 201 YVEL=0
BOUND HIST SINE (1.0,5.0)
FRAC=0.1
HIST N=25 SXY -160,-200 SXY -160,200  TYPE=1
HIST     XVEL(-160,-200) XVEL(-160,200)
INSITU STRESS 0 0 -1E-6
CYC 1500
PR MAX
PR HIST 1 2 3 4
SAVE TESTA2
STOP
```

UDEC VERIFICATION TEST B

ELASTIC BEHAVIOR OF A JOINTED MEDIUM

This verification analysis demonstrates the capability of UDEC to simulate the elastic response of a jointed rock mass. The analytical solution, given by Singh (1973), relates a transversely isotropic continuum characterization of the medium to the properties of the joints and the intact material in the medium. The representation of this characterization is given in Fig. B-1. Transversely isotropic elastic rock mass moduli can be derived in terms of joint normal and shear stiffness and intact rock moduli for two orthogonal joint sets with staggered joints. The relationships for rock mass moduli defined with respect to the axes of anisotropy, n and s, shown in Fig. B-1, are:

$$E_n = E' = \frac{E_i}{1 + \dfrac{b_{nn} E_i}{k_{nh} s_n}} \tag{B-1}$$

$$E_s = \frac{E_i}{1 + \dfrac{E_i}{k_{nv} s_s}} \tag{B-2}$$

$$G_{ns} = \frac{G_i s_s s_n k_{sh} k_{sv}}{s_s s_n k_{sh} k_{sv} + G_i b_{sn} s_s k_{sv} + G_i s_n k_{sh}} \tag{B-3}$$

$$\nu_{ns} = \nu_i \frac{k_{nh}\ s_n}{k_{nh}\ s_n + b_{nn}\ E_i} \tag{B-4}$$

where E_n is the modulus of elasticity in the n-direction (i.e., normal to the plane of isotropy),

E_s is the modulus of elasticity in the s-direction (i.e., in the plane of isotropy),

G_{ns} is the shear modulus,

ν_{ns} is the Poisson's ratio which relates strain in the plane of isotropy to strain in the n-direction for loading the in the n-direction,

E_i is the intact rock elastic modulus,

ν_i is the intact rock Poisson's ratio,

G_i is the intact rock shear modulus,

k_{nv}, k_{sv} are the normal and shear stiffnesses, respectively, of the subvertical joints,

k_{nh}, k_{sh} are the normal and shear stiffnesses, respectively, of the subhorizontal joints,

s_s is the average spacing of the subvertical joints in the s-direction,

s_n is the average spacing of the subhorizontal joints in the n-direction,

s is the joint offset of the staggered subhorizontal joints,

and

$$b_{nn} = \left[1 + \frac{k_{sv}\ s}{k_{nh}\ s_s} \left(1 - \frac{s}{s_n}\right) \right]^{-1} \tag{B-5}$$

$$b_{sn} = \left[1 + \frac{k_{nv}\ s}{k_{sh}\ s_s} \left(1 - \frac{s}{s_n}\right) \right]^{-1} \tag{B-6}$$

which are stress concentration factors for staggered joints.

This factor is defined by Singh as the ratio of average normal and shear stresses along the joint (σ and τ in Fig. B-1) to the corresponding overall stresses on a plane parallel to that joint within the rock mass. Equations (B-5) and (B-6) are derived for rigid rock blocks. Approximations for b_{nn} and b_{sn}, which include elasticity of the rock, are also given by Singh.

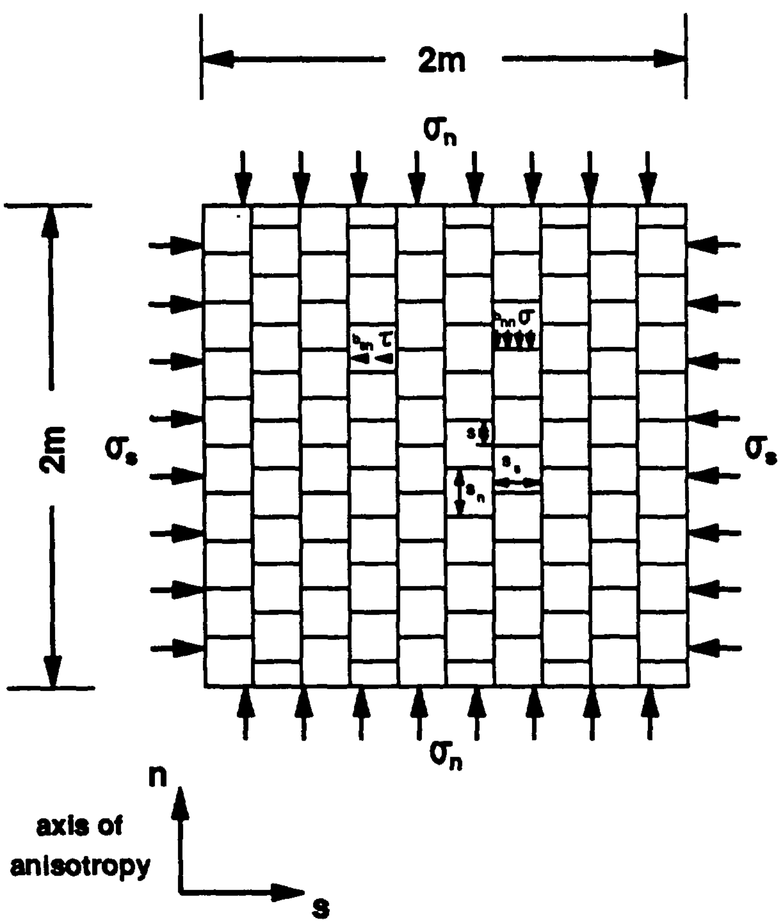


Fig. B-1 Rock Mass Containing Staggered Joints

Properties used in this analysis represent the behavior of jointed basalt. The average spacing of the joints are assumed to be 20 cm (i.e., $s_n = s_s = 20$ cm, $s = 10$ cm). Joint properties, estimated by Hart et al. (1985), are:

$$k_{nv} = 395.9 \text{ GPa/m}$$

$$k_{sv} = 438.8 \text{ GPa/m}$$

$$k_{nh} = 531.9 \text{ GPa/m}$$

$$k_{sh} = 438.8 \text{ GPa/m}$$

It is assumed that the intact rock properties are:

$$E_i = 89 \text{ GPa}$$

$$\nu_i = 0.26$$

The analytical solution using Eqs. (B-1) through (B-6) are compared to the calculated results using UDEC for the problem setting described in Fig. (B-1). Two tests are performed: the first, a load cycling in the n-direction while keeping the load in the s-direction constant; and the second, load cycling in the s-direction while keeping the n-direction load constant. In both tests, the cycled load is changed from 2 MPa to 10 MPa and back to 2 MPa, while the other load is fixed at 5 MPa.

For the problem conditions shown in Fig. B-1, the joint spacings must be adjusted to account for the effect of the model boundaries. Therefore, the spacings used in the analytical solution are

$$s_n = 2\text{m}/9.5 \text{ joints} = 0.21\text{m/joint}$$

and

$$s_s = 2\text{m}/9 \text{ joints} = 0.22\text{m/joint}$$

Using the above-noted joint properties and spacings, the transversely isotropic constants calculated from Eqs. (B-1) through (B-6) are:

$$E_n = 53.3 \text{ GPa}$$

$$E_s = 44.0 \text{ GPa}$$

$$G_{ns} = 21.0 \text{ GPa}$$

$$\nu_{ns} = 0.16$$

$$\nu = 0.26$$

These constants are used in stress-strain relations for transversely isotropic material behavior [e.g., see Lekhnitskii (1981)] to calculate strain in the n-direction and s-direction for the two load cycle tests. Two-dimensional plane strain conditions are imposed on these relations to make the calculations compatible with UDEC model conditions.

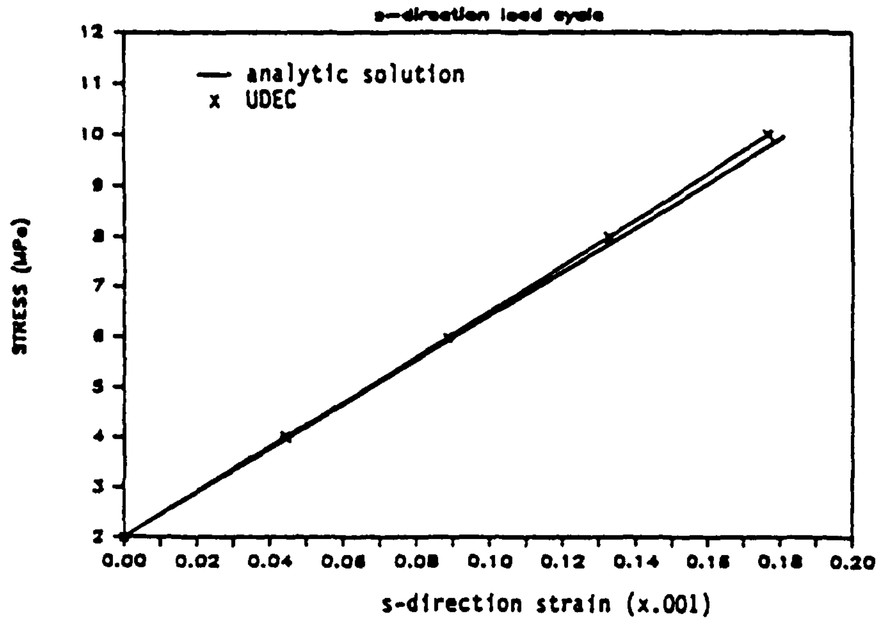
The results from the analytical solutions and those from UDEC are compared in Fig. B-3. The agreement is considered quite good for the two validation tests. The stress-strain slopes calculated by UDEC show a variation of less than 4% from those calculated from the analytical solutions.

REFERENCES

1. Singh, B. "Continuum Characterization of Jointed Rock Masses: Part I - The Constitutive Equations," Int. J. Rock Mech. Min. Sci. & Geomech. Abst., 10, 337-345 (1973).

2. Hart, R. D., P. A. Cundall, and M. L. Cramer. "Analysis of a Loading Test on a Large Basalt Block," Research and Engineering Applications in Rock Masses (Proceedings of the 26th U.S. Symposium on Rock Mechanics), Vol. 2, pp. 759-768. Eileen Ashworth, Ed. Boston: A. A. Balkema, 1985.

3. Lekhnitskii, S. G. Theory of Elasticity of an Anisotropic Body. Moscow: Mir Publishers, 1981.



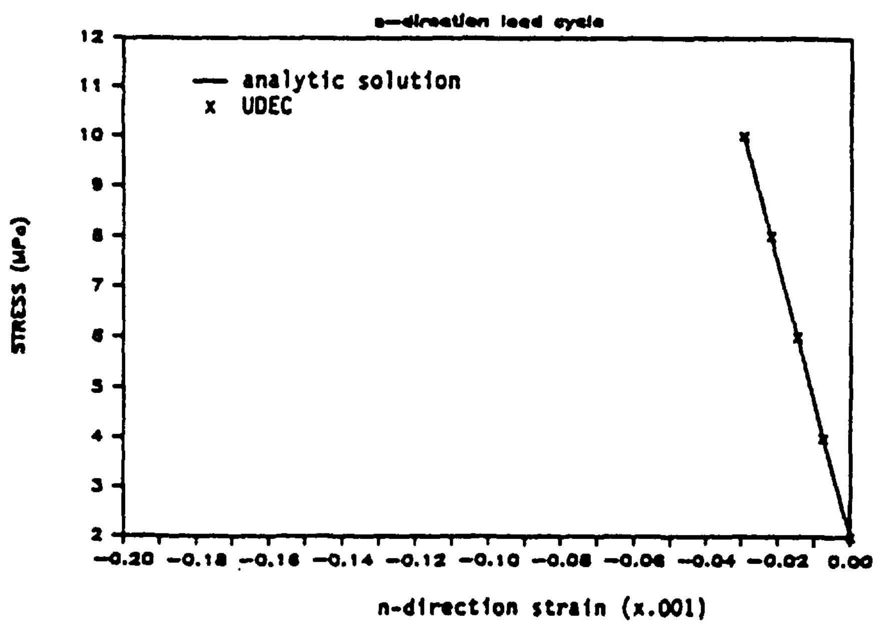


Fig. B2 Verification Test for s-Direction Load Cycling

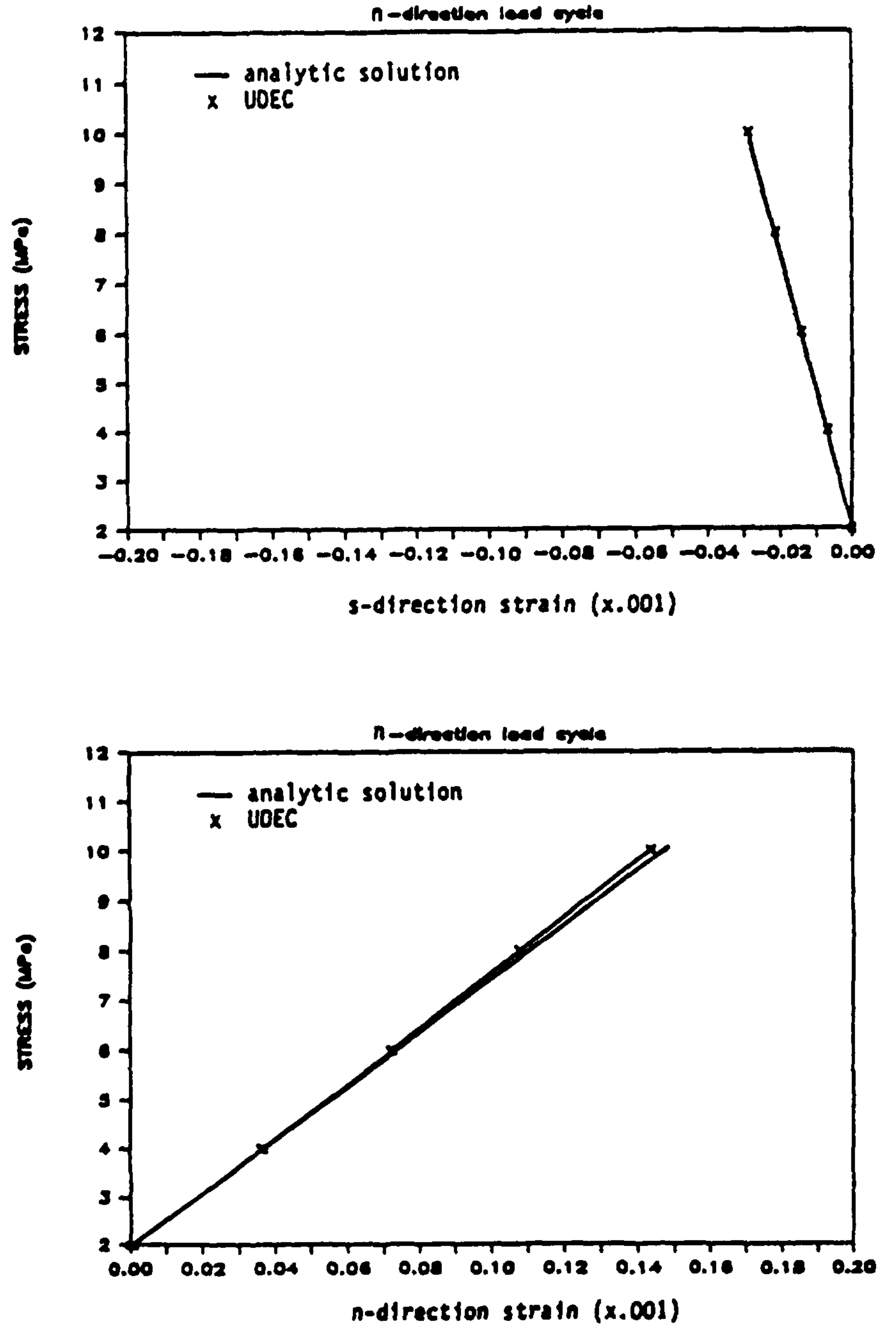


Fig. B-3 Verification Test for n-Direction Load Cycling

INPUT COMMANDS FOR VERIFICATION TEST B

```
START
*
* VERIFICATION TEST B
* ELASTIC BEHAVIOR OF A JOINTED MEDIUM
* (HORIZONTAL LOAD CYCLE)
*
ROUND 0.001
WINDOW -0.5,2.5 -0.5,2.5
BLOCK  0,0  0,2.0  2.0,2.0  2.0,0
JREG   0,0  0,2  2,2  2,0
JSET  90.0,0   5.0,0   0.0,0   0.2,0    ( 0.0,0.0, 0.0)
JSET   0.0,0  0.21,0  0.19,0   0.2,0    (-0.20,0.0, 0.0)
JSET   0.0,0  0.21,0  0.19,0   0.2,0    ( 0.0,0.1, 0.0)
JDEL
CHANGE  FDEF
GEN 0,2 0,2  AUTO 0.28
CHANGE MAT=1
* HORIZONTAL JOINT PROPERTIES
PROP M=1 D=2850 K=61.8E9 G=35.3E9 JKN=531.9E9 JKS=438.8E9 JF=1.0724
+      (KN=15.0E9 KS=15.0E9)
* VERTICAL JOINT PROPERTIES
PROP M=2 D=2850 K=61.8E9 G=35.3E9 JKN=395.9E9 JKS=438.8E9 JF=1.0724
+      (KN=15.0E9 KS=15.0E9)
CHANGE ANGLE -1,1   JMAT=1 JCON=2
CHANGE ANGLE 89,91  JMAT=2 JCON=2
DAMP 2.0 2500 MASS
BOUND   STRESS  -2.0E6 0.0 -5.0E6
INSITU  STRESS  -2.0E6 0.0 -5.0E6
HIST N=50  YDIS=1,0 YDIS=1,2 XDIS=0,1 XDIS=2,1
HIST TYPE 1
*
* X-LOAD INCREMENT (LOAD STEP 1)
BOUND -.1,.1  -.1,2.1  STRESS -2.0E6  0.0  0.0
BOUND 1.9,2.1  -.1,2.1  STRESS -2.0E6  0.0  0.0
CYCLE 1500
PRINT MAX
```

```
*
* X-LOAD INCREMENT (LOAD STEP 2)
BOUND -.1,.1  -.1,2.1  STRESS -2.0E6  0.0  0.0
BOUND 1.9,2.1  -.1,2.1  STRESS -2.0E6  0.0  0.0
CYCLE 1500
PRINT MAX
*
* X-LOAD INCREMENT (LOAD STEP 3)
BOUND -.1,.1  -.1,2.1  STRESS -2.0E6  0.0  0.0
BOUND 1.9,2.1  -.1,2.1  STRESS -2.0E6  0.0  0.0
CYCLE 1500
PRINT MAX
*
* X-LOAD INCREMENT (LOAD STEP 4)
BOUND -.1,.1  -.1,2.1  STRESS -2.0E6  0.0  0.0
BOUND 1.9,2.1  -.1,2.1  STRESS -2.0E6  0.0  0.0
CYCLE 1500
PRINT MAX
SAVE UDB1
*
* X-LOAD INCREMENT (UNLOAD STEP 5)
BOUND -.1,.1  -.1,2.1  STRESS  2.0E6  0.0  0.0
BOUND 1.9,2.1  -.1,2.1  STRESS  2.0E6  0.0  0.0
CYCLE 1500
PRINT MAX
*
* X-LOAD INCREMENT (UNLOAD STEP 6)
BOUND -.1,.1  -.1,2.1  STRESS  2.0E6  0.0  0.0
BOUND 1.9,2.1  -.1,2.1  STRESS  2.0E6  0.0  0.0
CYCLE 1500
PRINT MAX
*
* X-LOAD INCREMENT (UNLOAD STEP 7)
BOUND -.1,.1  -.1,2.1  STRESS  2.0E6  0.0  0.0
BOUND 1.9,2.1  -.1,2.1  STRESS  2.0E6  0.0  0.0
CYCLE 1500
PRINT MAX
*
```

```
* X-LOAD INCREMENT (UNLOAD STEP 8)
BOUND -.1,.1  -.1,2.1  STRESS  2.0E6  0.0  0.0
BOUND 1.9,2.1  -.1,2.1  STRESS  2.0E6  0.0  0.0
CYCLE 1500
PRINT MAX
PRINT HIST 1 2 3 4
SAVE UDB2
STOP
```

UDEC VERIFICATION TEST C

CRACK SHEAR BY REDUCED FRICTION

The exact solution for total shear displacement across a crack is given by:

$$u_s^{(tot)} = -\frac{2(1-\nu)}{G} \tau \quad (a^2 - x^2)^{1/2}$$

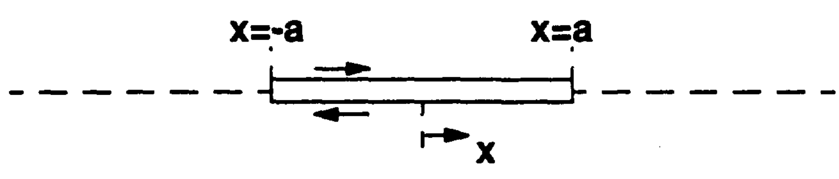
where G = shear modulus,

ν= Poisson's ratio,

τ = shear stress on crack surfaces,

a = half-width of crack, and

x = distance from center of crack.



In UDEC, shear stress, τ, is a stress drop from an initial in-situ stress caused by sliding, or:

$$\tau^{(drop)} = \mu \, \sigma_n - \tau^{(initial)}$$

The conditions used in this problem are:

$$G = 10^9 \text{ Pa},$$

$$\nu = 0.25,$$

$$\mu = 0.167,$$

$$\tau^{(\text{initial})} = 10^7 \text{ Pa},$$

$$\sigma_n = 1.5\text{x}10^7 \text{ Pa, and}$$

$$\tau^{(\text{drop})} = 0.167(1.5\text{x}10^7) - 10^7$$

$$= -\ 7.5\text{x}10^6 \text{ Pa}.$$

An embedded crack was created in UDEC by preventing the ends of a through-going joint from sliding. The crack, shown as line A-B in Fig. C-1, is 10m long and is represented by 11 elements. The UDEC model is composed of two blocks with a total of 768 triangular finite-difference zones. A full boundary element matrix is in effect around the model. The results from the UDEC simulation and the exact solution are plotted in Fig. C-2. The difference in agreement is 6%. Figure C-3 displays contours of the xx-component of the stress tensor. An extensive stress concentration is shown near the ends of the joint.

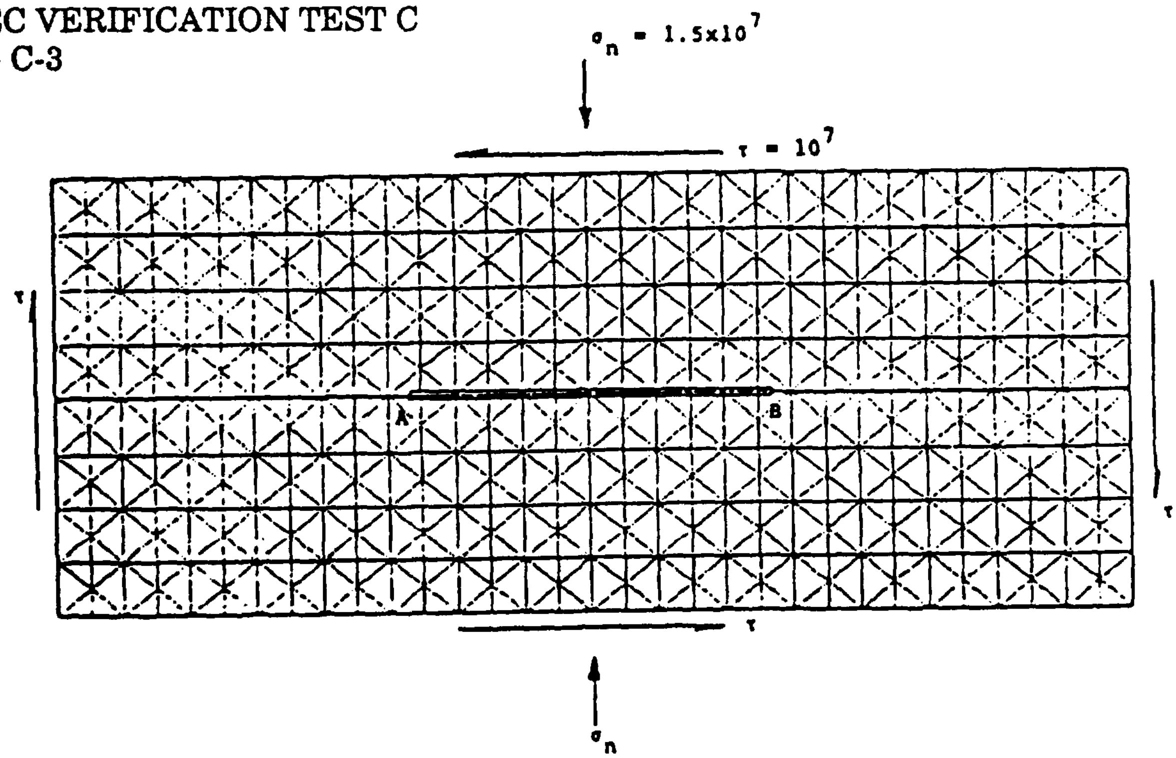


Fig. C-1 UDEC Model — Two Blocks with Triangular Finite-Difference Zones in Each Block

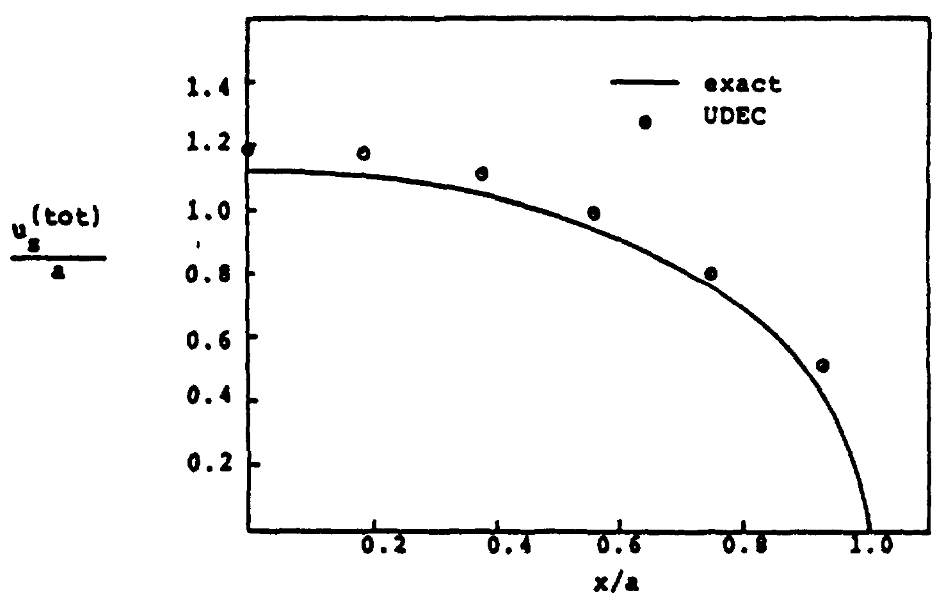


Fig. C-2 Comparison of Results for Crack Shear

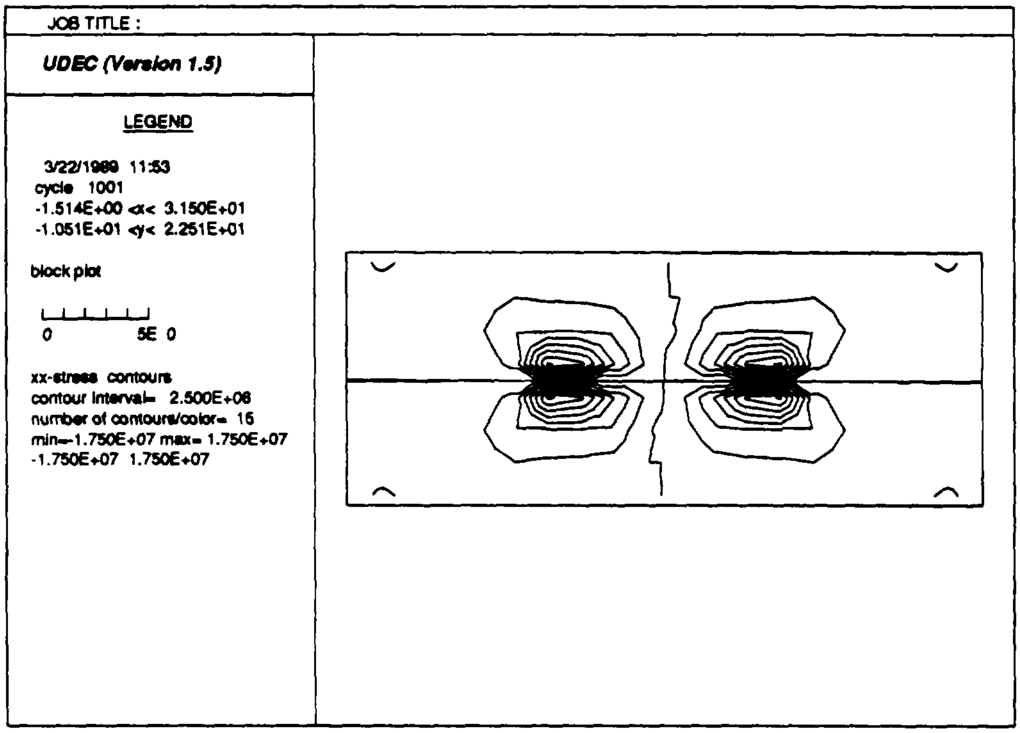


Fig. C-3 xx-Stress Contours

INPUT COMMANDS FOR VERIFICATION TEST C

```
START
*
* VERIFICATION TEST C
* CRACK SHEAR BY REDUCED FRICTION
*
ROUND 0.01
BLOCK 0,0 0,12 30,12 30,0
WIND -1,31  -1,13
SPLIT -1,6   31,6
CHANGE FDEF
GEN 0,30  0,12  AUTO 1.5
PROP M=1 D=3000 K=1.667E9 G=1.0E9 JKN=12.6E9  JKS=5.4E9 JF=2.5
+       (KN=1.0E8 KS=0.5E8 F=2.5)
PROP M=2 D=3000 K=1.667E9 G=1.0E9 JKN=12.6E9  JKS=5.4E9 JF=100
+       (KN=1.0E8 KS=0.5E8 F=100)
CHANGE JMAT=2 JCON=2 MAT=1
CHANGE  10,20  0,12  JMAT=1 JCON=2
DAMP AUTO
HIST N=20 YDISP= 15,6  XDISP= 15,6
HIST TYPE=2
BOUND 0,30  -1,1   STRESS  0.0 -1.0E7 -1.5E7
BOUND 0,30  11,13  STRESS  0.0 -1.0E7 -1.5E7
BOUND -1,1   0,12  STRESS  0.0 -1.0E7  0.0
BOUND 29,31  0,12  STRESS  0.0 -1.0E7  0.0
INSITU  STRESS  0.0 -1.0E7 -1.5E7
CYCLE 1
* LINK BOUNDARY TO B.E. PROGRAM
BE MAT=1
BE GEN -1,31  -1,13
BE FIX  6,-1   35,0
BE STIFF
CHANGE JMAT=2 JCON=2
CHANGE  10,20  0,12  JMAT=1 JCON=2
* REDUCE FRICTION TO INDUCE SLIP
PROP M=1 JF=0.1667
CYCLE 1000
```

```
PRINT MAX
PRINT CONT
SAVE UDC0
STOP
END
```

UDEC VERIFICATION TEST D

ROUGH FOOTING ON COHESIVE MATERIAL

The plasticity model in UDEC is verified by the "Prandtl's Wedge" problem. Prandtl showed that the failure stress for a rough footing on cohesive material is given by the exact solution:

$$q = (2 + \pi) c$$

where q is the stress, and c is the cohesion of the material. The equation is referred to as the Prandtl wedge solution because failure zones or wedges are assumed to develop at collapse in the manner shown in Fig. D-1.

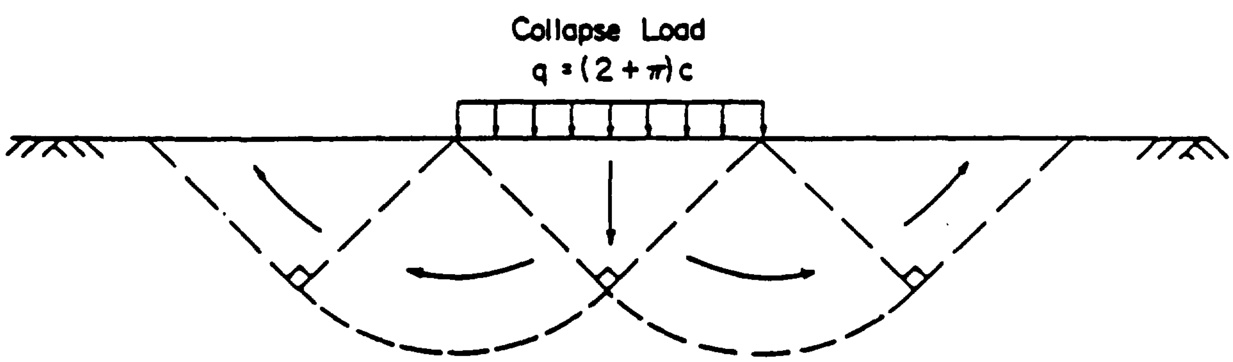


Fig. D-1 Failure Zones at Collapse Loading for Prandtl Wedge Solution

The UDEC model for this problem is shown in Fig. D-2. The model contains two sets of four orthogonal joints which divide the medium into 25 blocks. Lines of symmetry are imposed on the left, right, and bottom sides of the model. A constant velocity of $2.0x10^{-3}$ m/sec is applied at the top-left corner to simulate the footing load. By applying the loading condition in this way, the limiting load at failure can be calculated by monitoring the force build-up resulting from the velocity boundary.

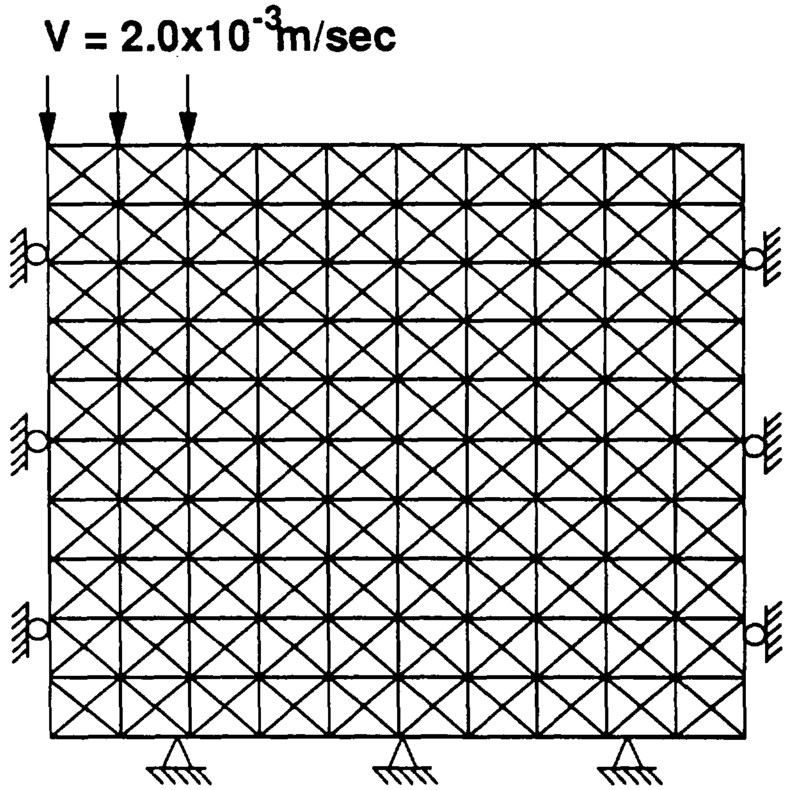


Fig. D-2 UDEC Model

Mohr-Coulomb material behavior was prescribed for the intact blocks with the following properties:

Bulk Modulus = 10^8 Pa

Shear Modulus = 10^8 Pa

Cohesion = 10^4 Pa

Friction Coefficient = 0.0

Only elastic deformation was permitted for the joints. Joint stiffnesses were:

Normal Stiffness = 10^8 Pa/m

Joint Stiffness = 10^8 Pa/m

The failure loading state is shown by the principal stress plot in Fig. D-3. The maximum load at failure is within 3% of the exact solution of $q = 5.14 \times 10^4$ Pa.

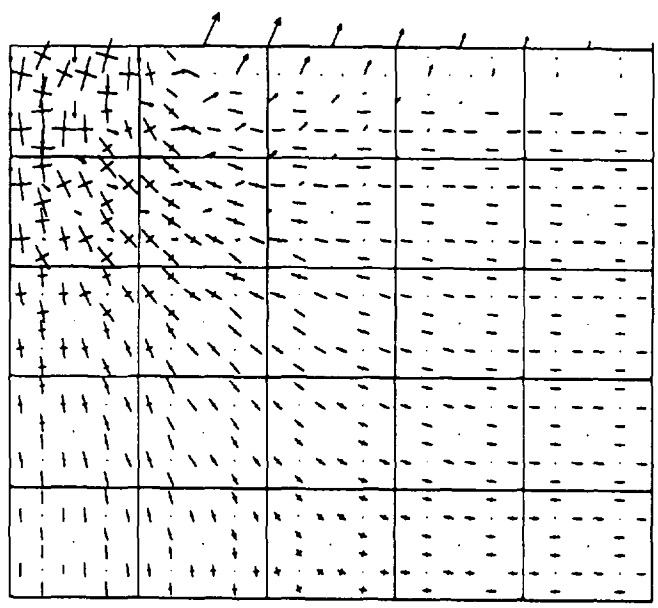


Fig. D-3 Principal Stresses and Displacement Vectors at Collapse

INPUT COMMANDS FOR VERIFICATION TEST D

```
START
*
* VERIFICATION TEST D
* ROUGH FOOTING ON COHESIVE MATERIAL
* WITH GLUED JOINTS
*
ROUND 0.01
BLOCK 0,0 0,10 11.5,10 11.5,0
WIND -1,12 -1,12
SPLIT -1,2 12,2
SPLIT -1,4 12,4
SPLIT -1,6 12,6
SPLIT -1,8 12,8
SPLIT 2.3,11 2.3,-.5
SPLIT 4.6,11 4.6,-.5
SPLIT 6.9,11 6.9,-.5
SPLIT 9.2,11 9.2,-.5
GEN 0,12 0,10 AUTO 1.5
CHANGE JCON=2 JMAT=3 MAT=1 CON=3
PROP MAT=1 D=1000 K=1E8 G=1E8 COH=1E4
PROP MAT=3 JKN=1E8 JKS=1E8 JF=100.0 KN=1E7 KS=1E7 F=100.0
*
BOUND -.1,.1 -3,11 XVEL=0.0
BOUND -.1,2.8 9.9 10.1 XVEL=0.0
BOUND 11.4,11.6 -3,11 XVEL=0.0
BOUND -1,13 -.1,.1 YVEL=0.0
BOUND 0,2.8 9.9,10.1 YVEL=-2E-3
*
DAMP AUTO
HIST N=500 YDISP=1,10 SYY=1,10
CYC 8000
PR MAX
PR BOU
PR HIS 2
SAVE TESTD1
CYC 2000
PR MAX
PR BOU
PR HIS 2
SAVE TESTD2
STOP
END
```

UDEC VERIFICATION TEST E

CIRCULAR TUNNEL EXCAVATION WITH INTERIOR SUPPORT

The structural element logic in UDEC is verified by considering the case of a circular tunnel with interior support excavated in an isotropic elastic medium subject to hydrostatic stresses. Conceptual understanding of the interaction between support loading and rock mass unloading is often explained in terms of a reaction curve for the rock medium and a stress-displacement curve for the support system. These curves are also called characteristic lines, characteristic curves, or rock response curves. In the conceptual model used in this approach, the problem is reduced to consideration of a plane perpendicular to the tunnel axis and all variables (stress, strain and displacement) vary only with radial distance from the tunnel. The ground reaction curve is frequently shown to consist of two parts — a descending portion and an ascending portion as shown in Fig. E-1. The descending portion of the ground reaction curve generally consists of two distinct parts, an elastic part and an inelastic part explained as follows. The model assumes that the excavation of the tunnel is simulated by unloading quasi-statically the boundary of the excavation. Upon unloading the system responds elastically until the elastic limit is reached. Further unloading causes propagation of a plastic or "failed" zone around the excavation. If gravity is neglected and the rock mass is assumed to behave as an isotropic, homogeneous, time-independent continuum, the descending portion of the ground reaction curve can be determined analytically based on material properties.

For the verification problem shown here, it is assumed that strength properties are sufficient such that only elastic behavior need be considered. The expression for radial displacement in an elastic material is given by

$$u_i = \frac{r_i}{2G} (P_o - P_i) \tag{E-1}$$

where r_i, u_i = tunnel radius and radial displacement,

P_o = in-situ stress,

G = shear modulus, and

P_i = internal pressure.

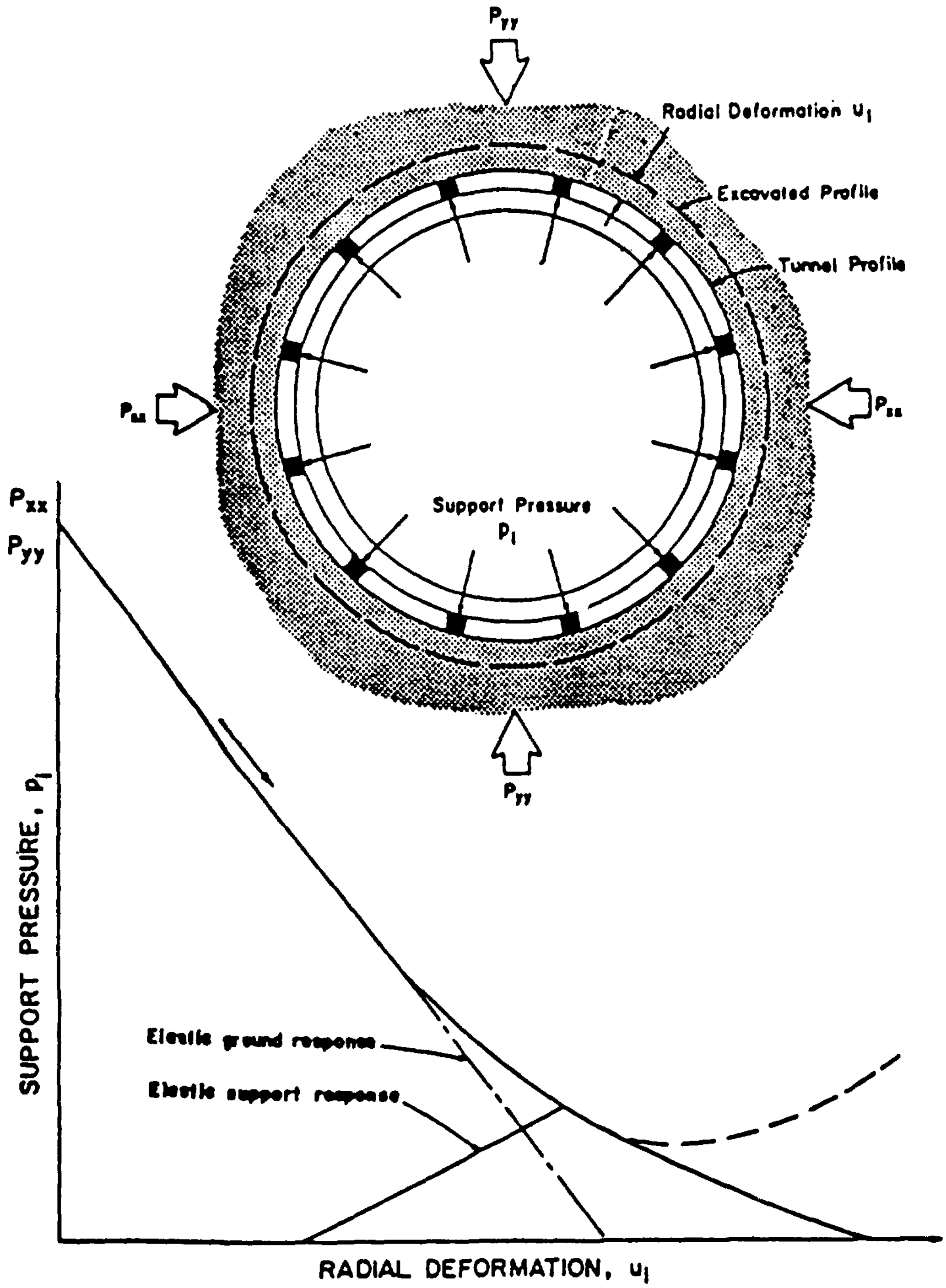


Fig. E-1 Conceptual Representation of Support Reaction and Ground Reaction Curves

Expressions have also been developed to describe the support stiffness of a variety of supports, assuming that the support reaction is radially symmetric. For example, the stiffness of a blocked steel set is given by Daemen (1975) as:

$$\frac{1}{K_{ss}} = \frac{u_i}{P_i} = \frac{Sr_i^2}{EA} + \frac{Sr_i^4}{EI}\left[\frac{\theta(\theta + sc)}{2s^2} - 1\right] + \frac{2Sr_i\theta t_B}{A_B E_B} \qquad \text{(E-2)}$$

where K_{ss} = stiffness of blocked steel set,

A, E, I = steel cross-sectional area, elastic modulus, and moment of inertia, respectively,

S = steel set spacing,

2θ = angle between blocking points,

$n = \pi/\theta$ = number of blocking points,

s = $\sin\theta$,

c = $\cos\theta$,

E_B, t_B = elastic modulus and thickness of blocks, and

A_B = cross-sectional area of blocks.

The verification problem consists of four separate calculations:

(1) tunnel excavation without support (far-field stress constant);

(2) tunnel excavation with support (far-field stress constant);

(3) tunnel excavation without support (far-field boundary fixed); and

(4) tunnel excavation with support (far-field boundary fixed).

The two assumptions concerning far-field boundaries (i.e., constant stress and fixed) are required to "bound" the numerical solution because the analytical solution assumes infinite far-field boundaries. The problem parameters used to describe the ground reaction are:

$$r_i = 1\text{m}$$

$$G = 1 \text{ MPa}$$

$$P_o = 10 \text{ MPa}$$

The problem parameters used to describe the support stiffness are:

$$A = 0.1\text{m}^2$$

$$E = 2.57 \text{ GPa}$$

$$I = 8.33\text{x}10^{-5} \text{ m}^4$$

$$S = 1\text{m}$$

$$n = 24$$

The blocking is described in the numerical model by k_n, the interface normal stiffness (force/disp) between the structure (i.e., steel set) and the rock. In this case, the last term in Eq. (E-2) is replaced by $(2 \; Sr_i)/k_n$, where

$$k_n \text{ is } (A_B E_B)/t_B \;\; (k_n{=}1000 \text{ MN/m}).$$

The usual assumption made in analyzing blocked steel sets is that no shear force is transferred between the rock mass and the steel set. Consequently, friction and cohesion values were not specified (default value is zero).

In setting up the numerical problem, the problem domain was divided into quadrant and concentric rings to facilitate automatic zoning. The discontinuities were assigned high stiffnesses relative to elasticity of the rock mass. These joints were also assigned high strength properties to ensure elastic behavior. The net result is that the joints are essentially "transparent" and do not effect the final result. The first quadrant zone discretization is shown in Fig. E-2.

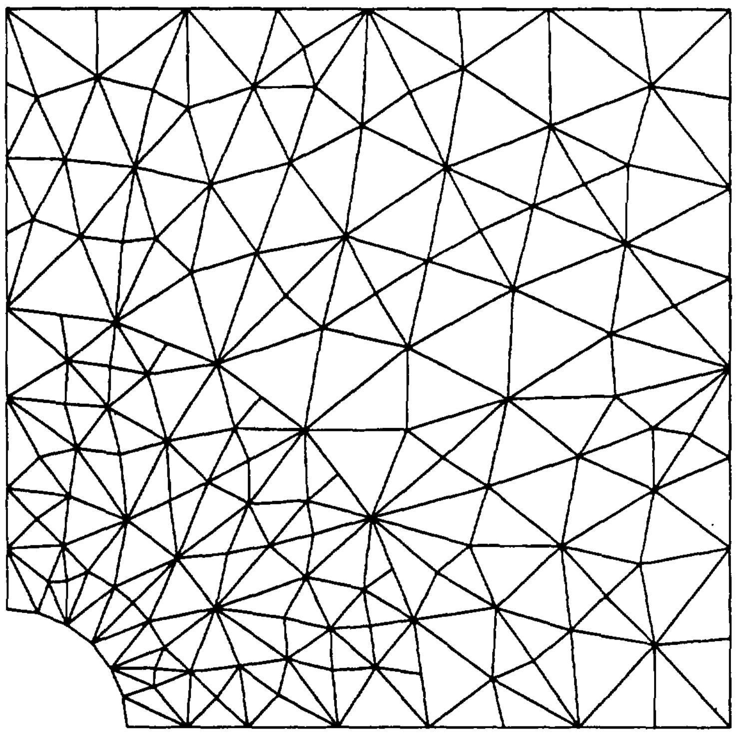


Fig. E-2 First Quadrant Discretization for Verification Problem Involving Circular Excavation (All zones are constant strain finite difference triangles.)

The results of the analyses are compared numerically in Table E-1 and graphically in Fig. E-3. As expected, the analytical solution falls between the numerical solutions obtained assuming different boundary conditions. In analyzing the results, the internal pressure, P_i, supplied by the support can be determined in one of two ways. The internal pressure is given by the thrust or axial force in the structural elements divided by the external radius of the support (i.e., 1m), or by the radial force in the "blocking" divided by its tributary area $(=2\pi r_i S/n)$. Both methods will yield the same value for P_i supplied by the support.

Reference

Daemen, J.J.K. "Tunnel Support Loading Caused by Rock Failure," Ph.D. Thesis, University of Minnesota, 1975; also available as U.S. Army Corps of Engineers Report MRD-3-75.

Table E-1

COMPARISON OF ANALYTICAL AND NUMERICAL SOLUTIONS FOR A TUNNEL IN ISOTROPIC ELASTIC MEDIA SUBJECT TO HYDROSTATIC STRESS

	Analytical Solution	Zero Disp Boundary	Constant Stress Boundary
radial disp (mm) without support	5	4.7	5.2
support stiffness (MPa/m)	241	243	243
radial disp (mm) with support	4.46	4.2	4.6
internal pressure (MPa) from support	1.075	1.021	1.117

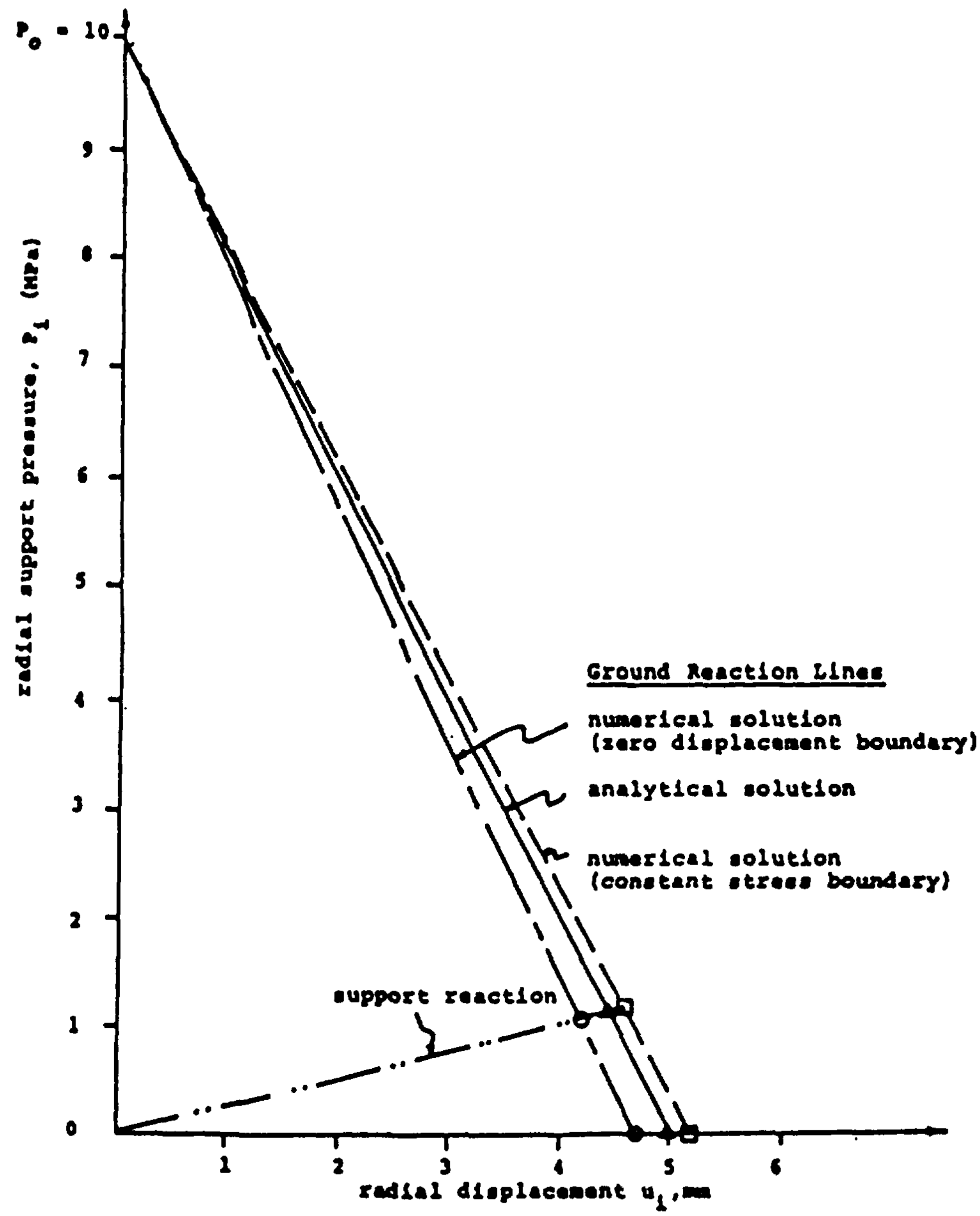


Fig. E-3 Ground Reaction/Support Reaction Lines for Verification Problems

INPUT COMMANDS FOR VERIFICATION TEST E

```
START
HEAD
VERIFICATION PROBLEM - CIRCULAR HOLE IN ISOTROPIC ROCK MASS
ROUND 0.01
BLOCK -6 -6 -6 6 6 6 6 -6
CRACK -10 0 10 0
CRACK 0 -10 0 10
TUNNEL 0 0 1 24
TUNNEL 0 0 2 24
TUNNEL 0 0 3.5 24
GEN 3 5 3 5 AUTO 1.2
GEN 1.5 2.0 1.5 2.0 AUTO .80
GEN .6 1.2 .6 1.2 AUTO 0.6
GEN -5 -3 -5 -3 AUTO 1.2
GEN -2.0 -1.5 -2.0 -1.5 AUTO .80
GEN -1.2 -.6 -1.2 -.6 AUTO 0.6
GEN -5 -3 3 5 AUTO 1.2
GEN -2 -1.5 1.5 2.00 AUTO .80
GEN -1.2 -.6 .6 1.2 AUTO 0.6
GEN 3 5 -5 -3 AUTO 1.2
GEN 1.5 2.0 -2.0 -1.5 AUTO .80
GEN .6 1.2 -1.2 -.6 AUTO 0.6
SAVE VERIFY
DAMP AUTO
BOUND STRESS -10 0 -10
INSITU   STRESS -10 0 -10
PROP  M=1  D=.003  K=2E3  G=1E3  JKN=1E4  JKS=1E4  JFRIC=10.0  JCOH=10.0
JTENS=10.0
CHANGE -6 6 -6 6 JCON=2
DEL -.49 .49 -.49 .49
HIST N=20 XDIS 1 0 XDIS -1 0 YDIS 0 1 YDIS 0 -1 TYPE 1
MSCALE ON  6.756E-5  1
CYC 400
PRI HIS 1 2 3 4
SAVE TUNNELA
STOP
END
```

```
REST VERIFY
* START PROBLEM AGAIN, BUT WITH SUPPORT
DAMP AUTO
BOUND STRESS -10 0 -10
INSITU   STRESS -10 0 -10
PROP  M=1  D=.003  K=2E3  G=1E3  JKN=1E4  JKS=1E4  JFRIC=10.0  JCOH=10.0
JTENS=10.0
CHANGE -6 6 -6 6 JCON=2
DEL -.49 .49 -.49 .49
HIST N=20 XDIS 1 0 XDIS -1 0 YDIS 0 1 YDIS 0 -1 TYPE 1
MSCALE ON  6.756E-5  1
STRU  0 0 7.5 1.1 24 1 .1
PROP M=1  KN=1E3  KS=1E0  CTENS=10
CYC 400
PRI HIS 1 2 3 4
PRI STE STC
SAVE TUNNELB
STOP
END

REST VERIFY
* REPEAT PROBLEM WITH FIXED BOUNDARY
DAMP AUTO
BOUND STRESS -10 0 -10
INSITU   STRESS -10 0 -10
PROP  M=1  D=.003  K=2E3  G=1E3  JKN=1E4  JKS=1E4  JFRIC=10.0  JCOH=10.0
JTENS=10.0
CHANGE -6 6 -6 6 JCON=2
DEL -.49 .49 -.49 .49
HIST N=20 XDIS 1 0 XDIS -1 0 YDIS 0 1 YDIS 0 -1 TYPE 1
MSCALE ON  6.756E-5  1
CYC 1
BOUND XVEL=0 YVEL=0
CYC 399
PRI HIS 1 2 3 4
SAVE TUNNELC
STOP
END
```

```
REST VERIFY
* START PROBLEM AGAIN, BUT WITH SUPPORT
DAMP AUTO
BOUND STRESS -10 0 -10
INSITU   STRESS -10 0 -10
PROP  M=1  D=.003  K=2E3  G=1E3  JKN=1E4  JKS=1E4  JFRIC=10.0  JCOH=10.0
JTENS=10.0
CHANGE -6 6 -6 6 JCON=2
DEL -.49 .49 -.49 .49
HIST N=20 XDIS 1 0 XDIS -1 0 YDIS 0 1 YDIS 0 -1 TYPE 1
MSCALE ON  6.756E-5  1
STRU  0 0 7.5 1.1 24 1 .1
PROP M=1  KN=1E3  KS=1E0  CTENS=10
CY 1
BOUND XVEL=0 YVEL=0
CYC 399
PRI HIS 1 2 3 4
PRI STE STC
SAVE TUNNELD
STOP
END
```

UDEC VERIFICATION TEST F

STEADY-STATE FLUID FLOW WITH FREE SURFACE

The new flow logic in UDEC allows for situations with a free surface, in addition to confined flow problems. The present test compares the UDEC results with a simple analytical solution for 2-D flow in a homogeneous aquifer governed by Darcy's Law:

$$v = k\ i$$

where v = discharge velocity,

k = coefficient of permeability (length/time), and

i = hydraulic gradient.

The problem is shown in Fig. F-1.

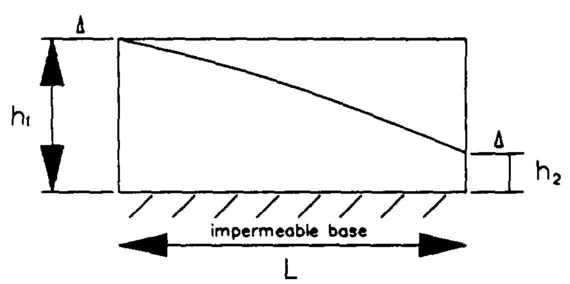


Fig. F-1 Figure Showing Definition of Terms in Dupuit's Formula

Dupuit's formula [see, for example, Harr (1962), p. 42] gives the total discharge (per unit width) as:

$$q = k \frac{h_1^2 - h_2^2}{2L} \tag{F-1}$$

The UDEC model is shown in Fig. 2 (block plot). A system of 2 sets of joints with constant aperture (i.e., stress-independent) was used to simulate the homogeneous isotropic medium.

The dimensions are:

$$L = 8 \text{ m}$$

$$h_1 = 4 \text{ m}$$

$$h_2 = 1 \text{ m}$$

$$\text{joint spacing } (S) = 0.5 \text{ m}$$

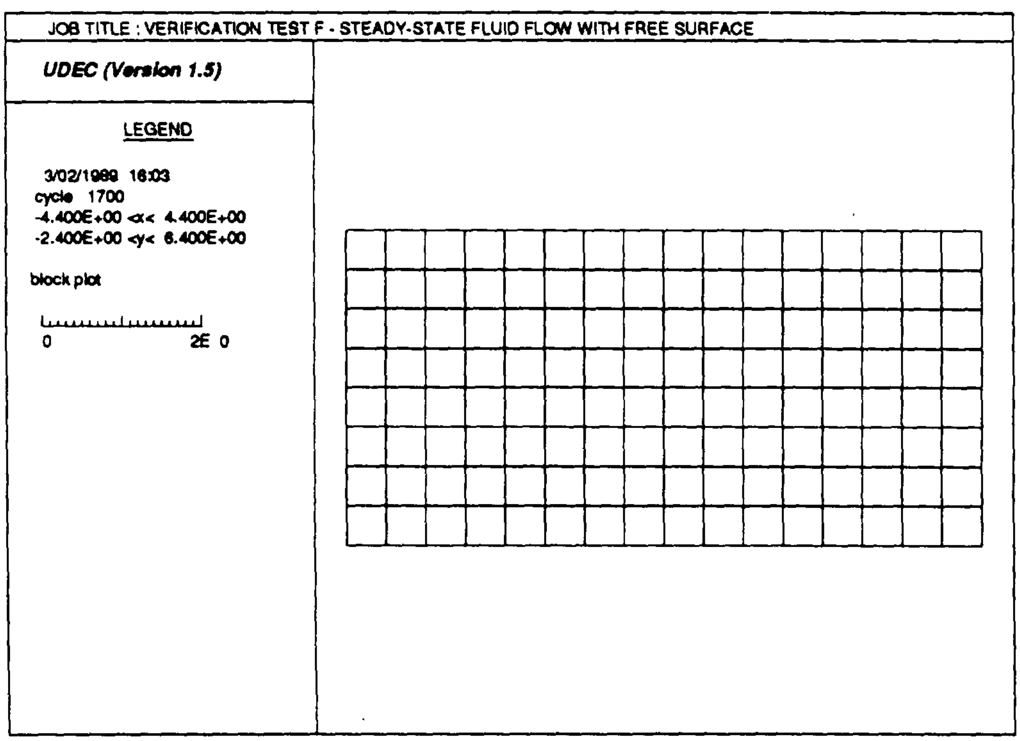


Fig. 2 UDEC Problem Geometry for Verification of Fluid Flow Logic

The flow rate in a single joint of length L, subject to a pressure difference of ΔP, is calculated in UDEC as:

$$q = JPERM\ a^3 \frac{\Delta P}{L} \qquad \text{(F-2)}$$

where JPERM = $(1/12\mu)$, with μ = dynamic viscosity, and a = aperture.

For a system of joints with spacing S, the average velocity for an equivalent continuum would be:

$$v = \frac{q}{S} = \frac{1}{12\mu} a^3 \frac{\Delta P}{L} \frac{1}{S}$$

$$= \frac{1}{12\mu} a^3 \rho_w g \frac{\Delta h}{L} \frac{1}{S}$$

where ρ_w is the fluid mass density (e.g., $\rho_w = 1000\ kg/m^3$ for water), and

g is the gravitational acceleration.

Because $\Delta h/L$ is the hydraulic gradient i in Darcy's Law, the coefficient k corresponds to

$$k = \frac{\rho_w g}{12\mu} \frac{a^3}{S}$$

For the following conditions,

$$a = 0.001 \text{ m}$$

$$\rho_w = 1000 \text{ kg/m}^3$$

$$g = 10 \text{ m/sec}^2$$

$$S = 0.5 \text{ m}$$

$$\text{JPERM} = \frac{1}{12\mu} = 3\text{x}10^8 \text{ MPa}^{-1} \text{ sec}^{-1}$$

we have $k = 6\text{x}10^{-6}$ m/sec.

Then, Dupuit's formula gives

$$Q = 5.625\text{x}10^{-6} \text{ m}^3\text{/sec}$$

The UDEC model gives (by summing discharge flow rates given by the **PRINT FLOW** command)

$$Q = 5.26\text{x}10^{-6} \text{ m}^3\text{/sec.}$$

The error is 6.5%.

Figure 3 shows the flow rates for the steady state. Figure 4 shows the domain pressures.

REFERENCE

Harr, M. E. Groundwater and Seepage. New York: McGraw-Hill Book Company, 1962.

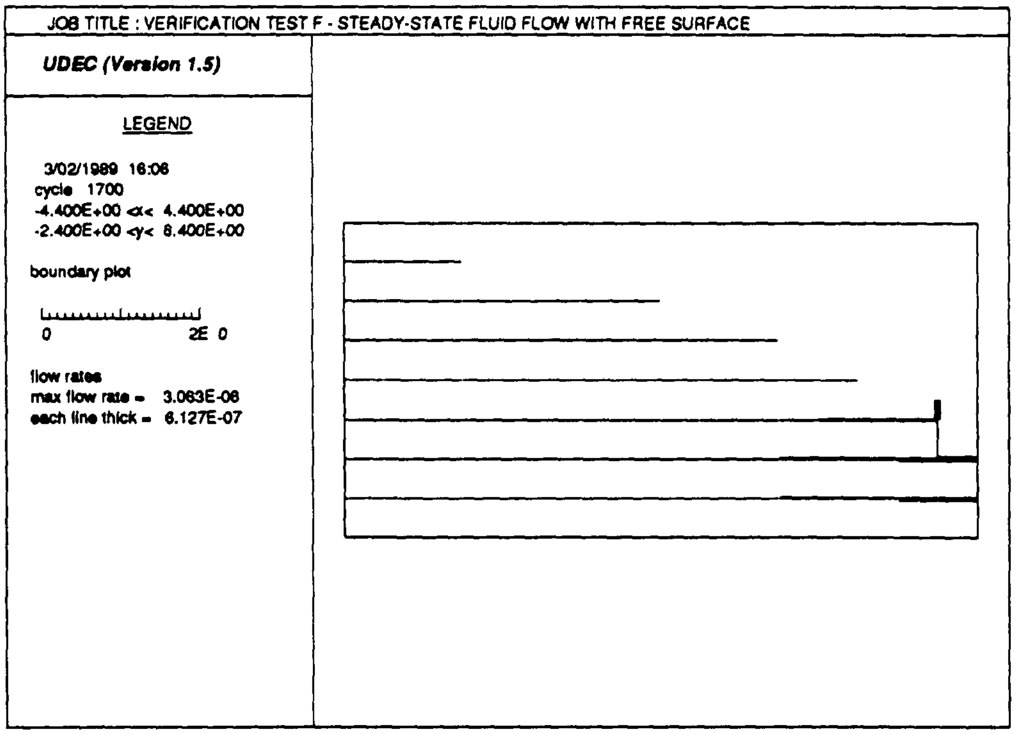


Fig. 3 UDEC Steady-State Flow Rates for Verification Problem

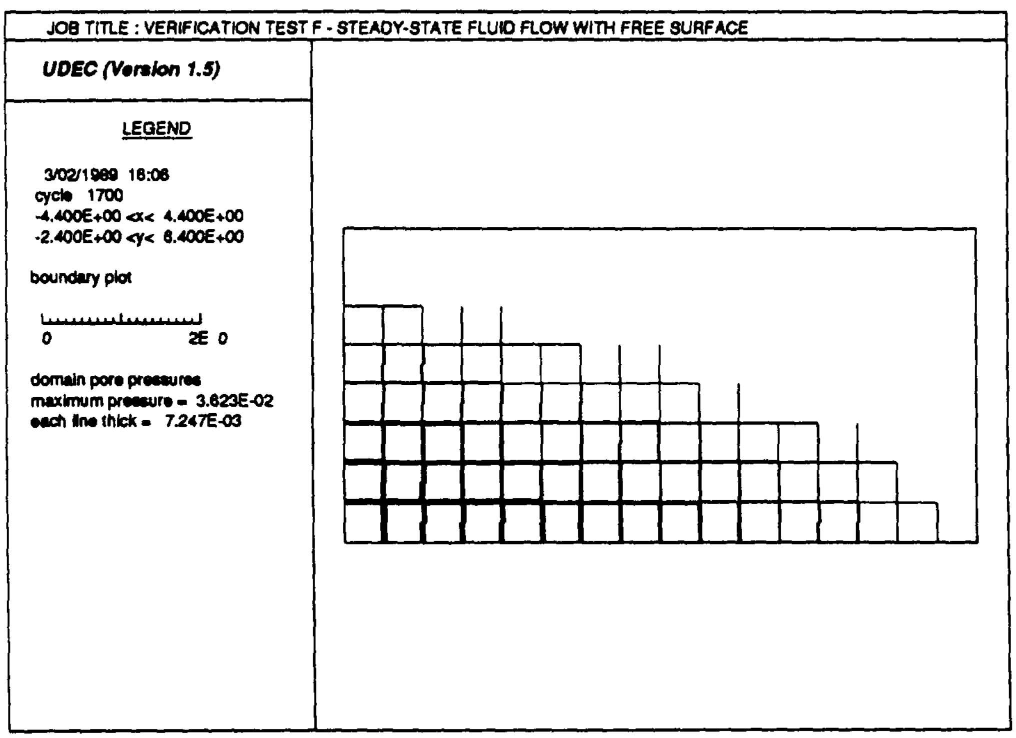


Fig. 4 UDEC Domain Pressures for Verification Problem

INPUT COMMANDS FOR VERIFICATION TEST F

```
*
round 0.01
*
bl -4 0  -4 4  4 4  4 0
*
jset  0,0  100,0  0,0  0.5,0
jset 90,0  100,0  0,0  0.5,0
*
gen  edge 4
*
prop  mat=1  d=0.0025  k=33333  g=20000
prop  mat=1  jkn=10000  jks=10000  jcoh=1.0e6   jten=1.0e6
*
* ste 1 : consolidation under gravity --- no flow
*
gravity  0 -10
*
bound  stress -0.05 0 -0.1  ygrad 0.0125 0 0.025
*
insitu stress -0.05 0 -0.1  ygrad 0.0125 0 0.025
*
bound  -5 5  -0.1 0.1  yvel=0
*
damp auto
*
hist xdis -4 4  ydis -4 4
*
cycle 200

*
* step 2 : flow
*
prop  mat=1  jperm=3.0e8  azero=-1  ares=0.0001
*
```

```
fluid DEN=0.001  BUL=200.0
set -eff
set sflow
*
bound  3.9 4.1  -1 10  xvel=0
*
bound -4.1 -3.9  -1 10  pp 0.04  pgrad -0.01
bound  3.9  4.1  -1 1.1  pp 0.01  pgrad -0.01
*
bound  -5 5  -0.1 0.1  impermeable
*
cycle  1500
*
save wn6f.sav
*
plot  nc  blocks
plot  bou  flow


plot  bou  pp
*
return
```

APPENDIX B

APPLICATION PROBLEMS

The following examples demonstrate how UDEC can be used to model problems from a variety of geomechanics applications. Each example consists of a problem description (including the properties required), an illustration of the problem geometry, and input data plots. The data files and output files are also provided on one of the diskettes enclosed with this manual.

APPLICATION PROBLEM A

SEISMIC-INDUCED GROUNDFALL

A demonstration simulation of a seismic-induced groundfall was performed with UDEC to illustrate the use of the code for analyzing rockburst potential. The model shown in Fig. A-1 was developed based on the configuration and dimensions of the 34-1-554 overcut shown on a section drawing for Fraser Mine, Falconbridge Limited, Sudbury, Ontario. A two-dimensional, plane strain representation was chosen normal to the axis of the overcut. The overcut was modeled as being 5m high and 10m wide.

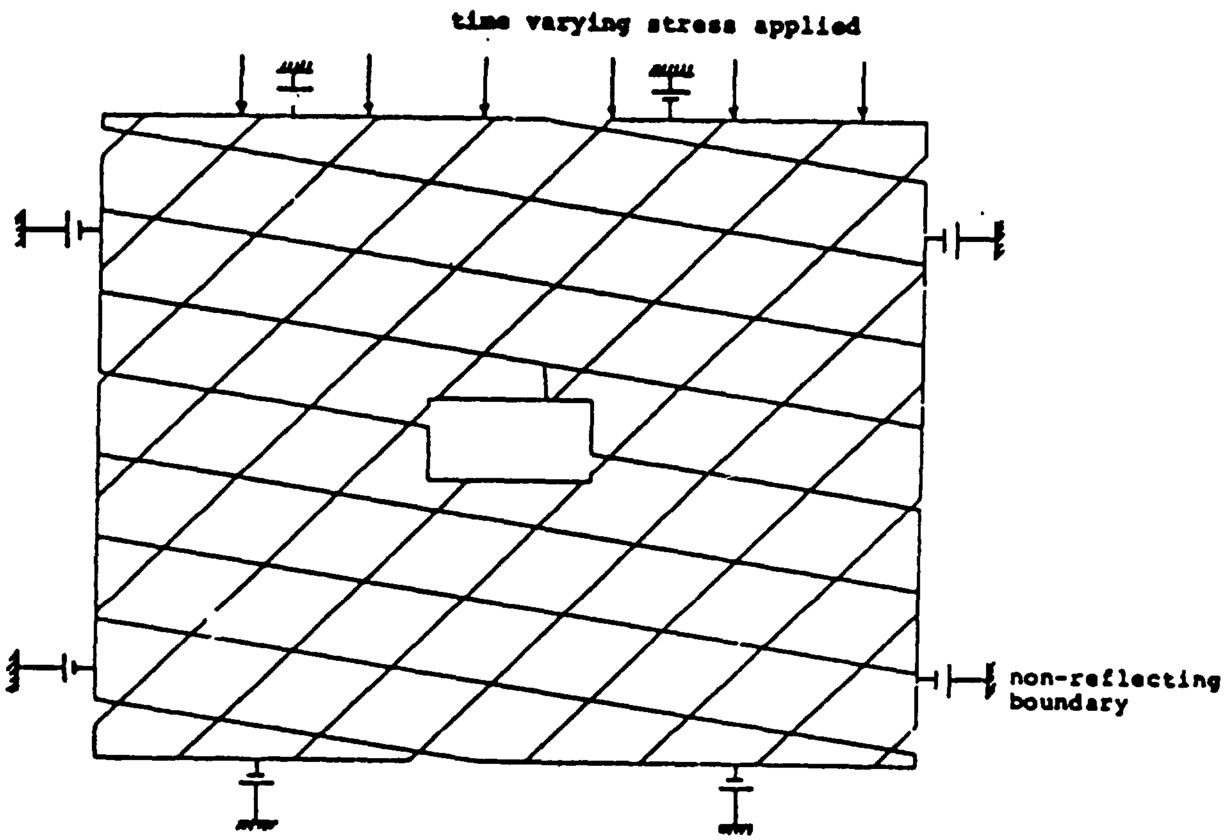


Fig. A-1 UDEC Model

It was assumed that two continuous joint sets intersect the plane of analysis; one with an orientation of 45° and the other with an orientation of 9°. Both sets have a joint spacing of 5m. For demonstration purposes, a near vertical "artificial" joint was also added to the block in the roof of the excavation to enhance the instability.

From the average laboratory test values provided for the intact rock, the following material properties were assumed for the rock blocks:

density	0.003×10^6 kg/m^3
Young's Modulus	75,000 MPa
Poisson's Ratio	0.18

The blocks were assumed to behave elastically only. Mohr-Coulomb behavior was assumed for the joints and typical textbook values were chosen for joint properties:

joint normal stiffness	20,000 MPa/m
joint shear stiffness	20,000 MPa/m
friction angle	30°
cohesion	0

The in-situ stress state was estimated to be isotropic at 24 MPa (assuming vertical loading due to overlying rock at a depth of approximately 800m).

The modeling sequence was performed in three stages. First, the model, without the overcut excavation, was consolidated under the in-situ stresses. Next, the excavation was introduced and the model cycled to an equilibrium state. The stress distribution around the overcut at this stage is illustrated in Fig. A-2. The blocks immediately above and below the overcut have slipped and then stabilized.

In the third stage, two different seismic events with different peak velocities were evaluated. For all seismic simulations, viscous boundaries were intro ıced around the outer perimeter of the problem domain to eliminate wave reflections, thereby simulating an infinite rock mass. Seismic events were represented by a sinusoidal y-directed stress wave applied at the top of the model. The applied stress wave was superimposed on the existing in-situ stresses.

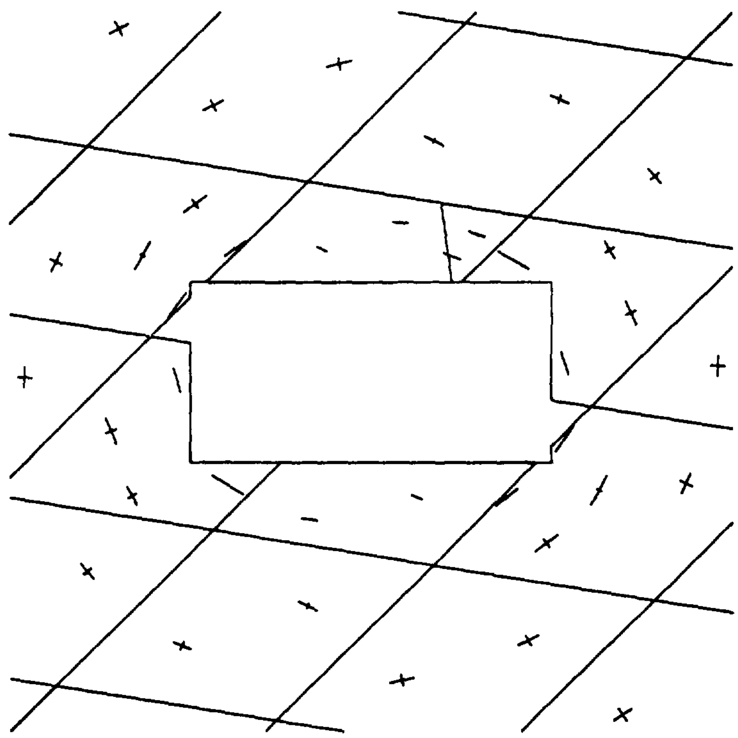


Fig. A-2 Stress Distribution Around Excavation at End of Excavation Stage

In the first simulation, a peak stress of 1.25 MPa was applied. It should be noted that, due to the viscous boundary conditions in effect at the top of the model, the "effective" applied stress is 1.25 MPa/2, or 0.625 MPa. The stress distribution in the roof of the excavation after 0.02 seconds is shown in Fig. A-3. Displacements were monitored at two points (shown in Fig. A-2). Point 1 is located in the left corner of the excavation; Point 2 is located at the right corner of the roof block. Displacement versus time plots (Fig. A-4) for these points essentially show an elastic response.

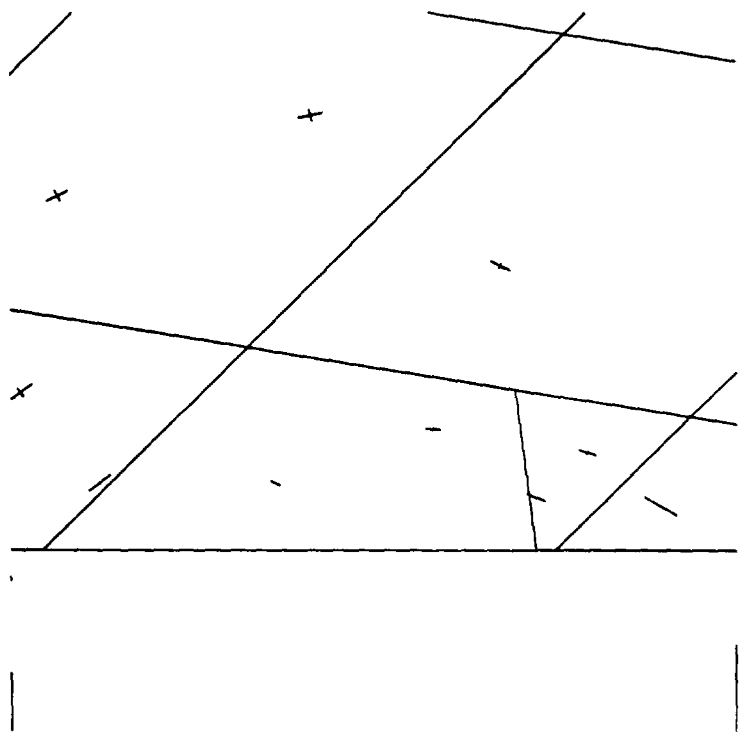


Fig. A-3 Stress Distribution in Roof of Excavation After 0.02 Seconds [applied stress = 1.25xcos(2π100t)]

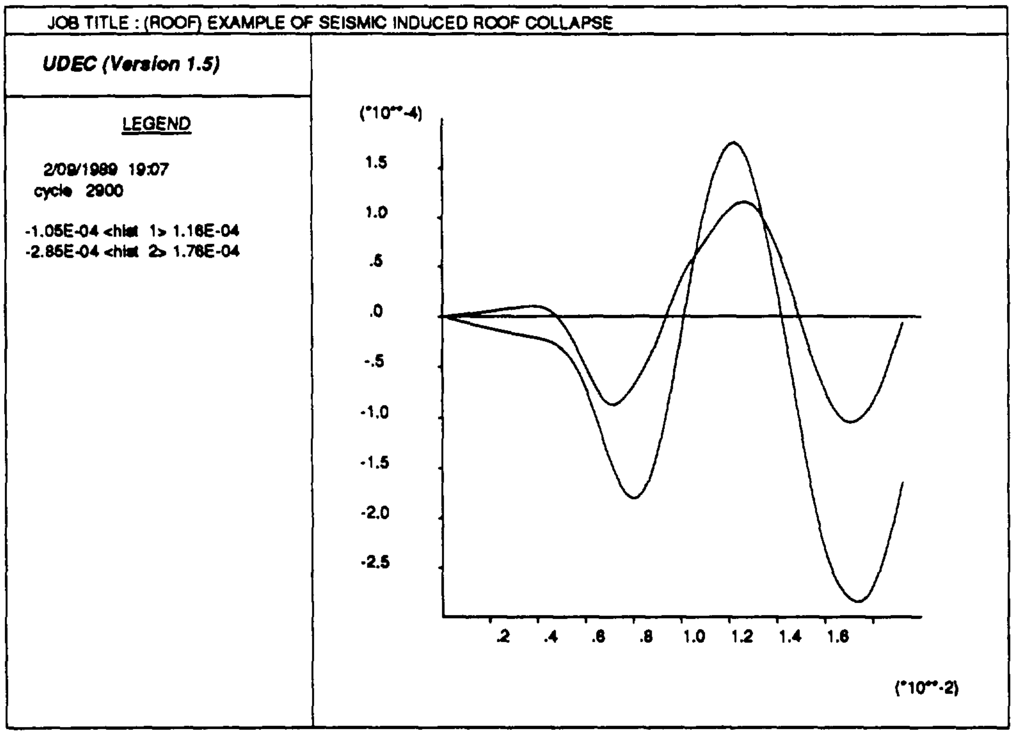


Fig. A-4 Y-Displacement Histories for Two Points on Excavation Boundary [applied stress = 1.25xcos(2π100t)]

It is interesting to compare estimated applied velocities with calculated velocities at the top of the model. The following equation can be used to estimate the applied velocity:

$$V = \frac{\sigma}{2(\rho C_p)}$$

where $C_p = [\ [K + (4/3)G]/\rho\ ]^{1/2}$

Using this equation, the applied maximum velocity would be about 0.04 m/sec. Figure A-5 shows a peak velocity of less than 0.06 m/sec. Differences between estimated velocities and measured velocities result from using the intact rock modulus rather than the equivalent deformation modulus which takes into account the joint deformation.

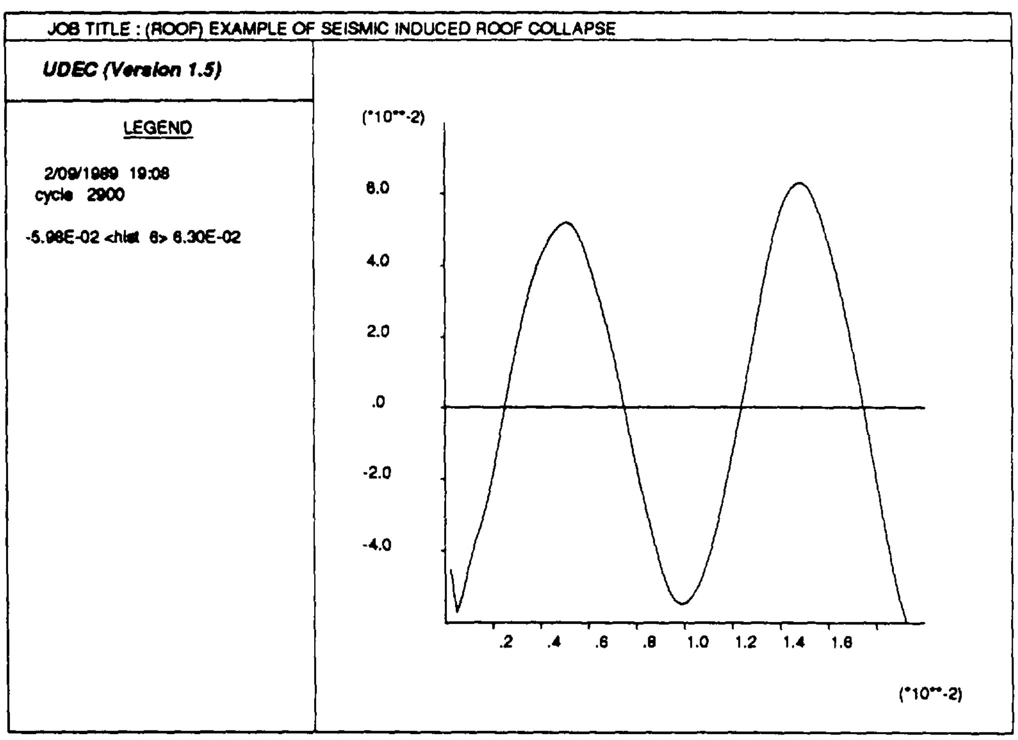


Fig. A-5 Plot of y-Velocity at Top of Model for Applied Stress = 1.25 x cos(2π100t)

In the second example, a stress wave with peak stress of 12.5 MPa ("effective" stress = 6.25 MPa) was applied. The stress distribution in the roof of the excavation after 0.02 seconds is shown in Fig. A-6. This figure shows that the roof block is unstressed, indicating that the block has loosened. Displacement versus time plots (Fig. A-7) also indicate that the block has loosened and is falling. As a matter of interest, the problem geometry and stress distribution at three later times is presented in Figs. A-8 to A-10.

The predicted velocity (from the equation above) at the top of the problem is 0.4 m/sec. The actual recorded velocity is shown in Fig. A-11. Again, differences between predicted and measured velocities result from using intact rock modulus rather than rock mass deformation modulus.

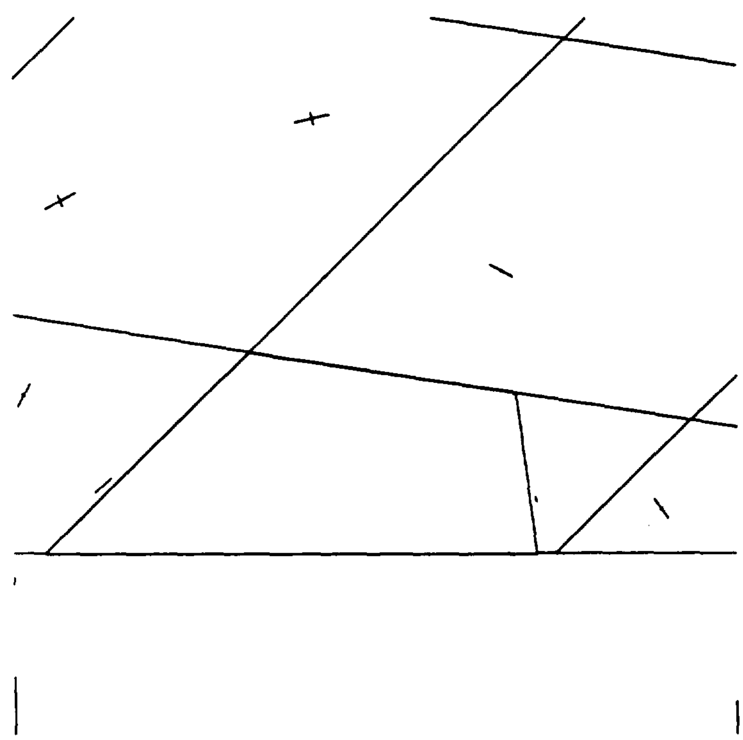


Fig. A-6 Stress Distribution in Roof of Excavation After 0.02 Seconds [applied stress = 12.5xcos(2π100t)]

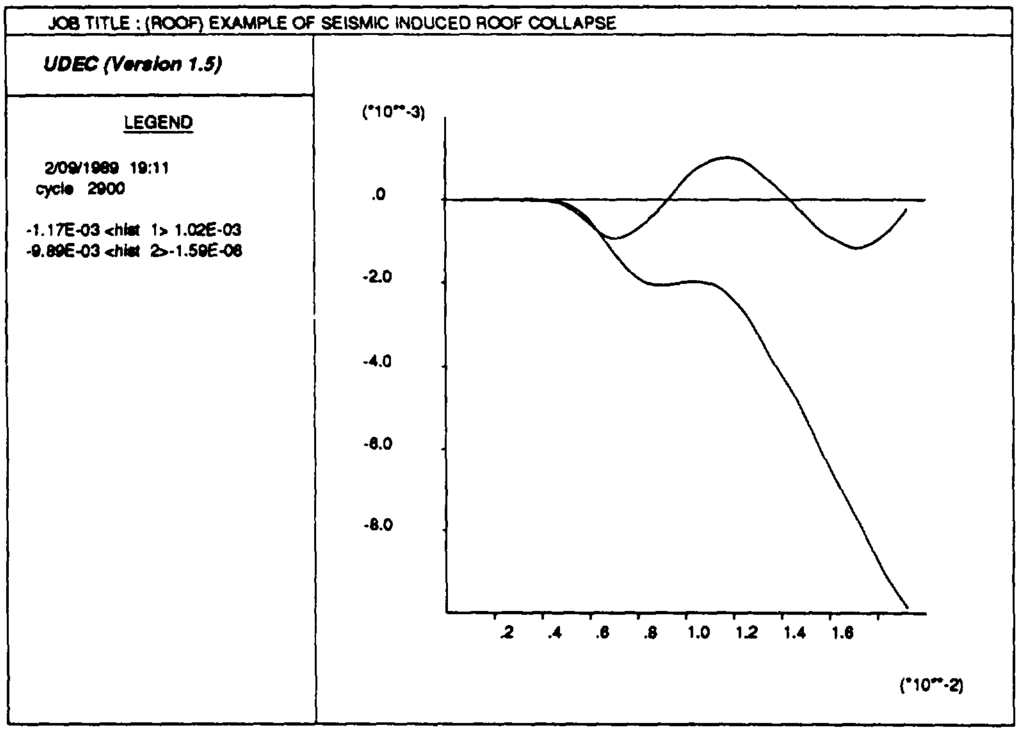


Fig. A-7 Y-Displacement Histories for Two Points on Excavation Boundary [applied stress = 12.5xcos(2π100t)]

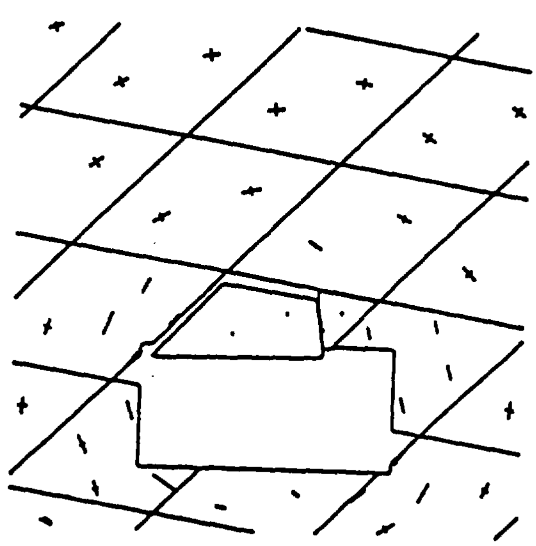


Fig. A-8 Stress Distribution Around Excavation After 0.26 Seconds [applied stress = 12.5xcos(2π100t)]

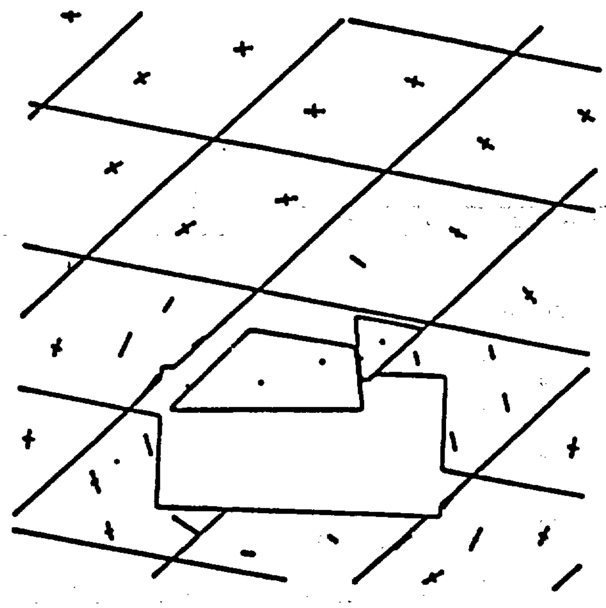


Fig. A-9 Stress Distribution Around Excavation After 0.51 Seconds [applied stress = 12.5xcos(2π100t)]

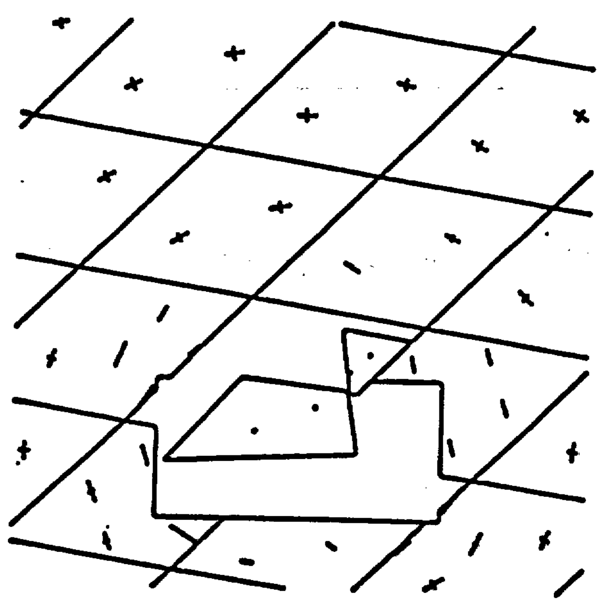


Fig. A-10 Stress Distribution Around Excavation After 0.75 Seconds [applied stress = 12.5xcos(2π100t)]

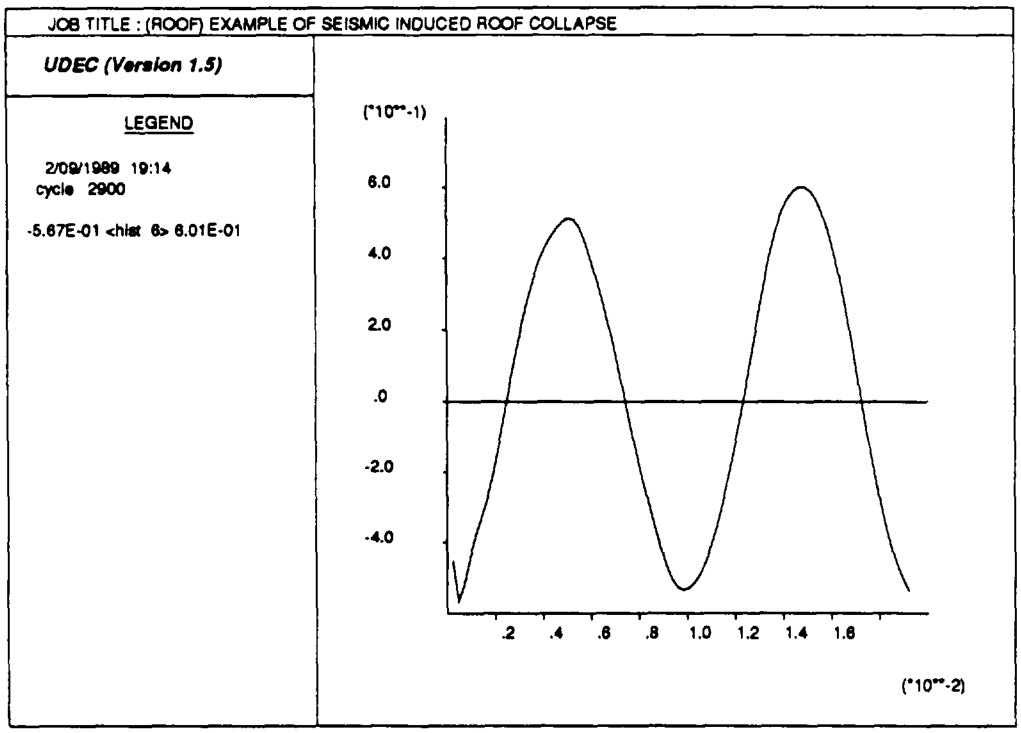


Fig. A-11 Plot of Y Velocity at Top of Model [applied stress = 12.5xcos(2π100t)]

INPUT COMMANDS FOR SEISMIC-INDUCED GROUNDFALL EXAMPLE

```
START
HEADING
(ROOF) EXAMPLE OF SEISMIC INDUCED ROOF COLLAPSE
*
ROUND  0.01
WINDOW -30,30 -30,30
* DEFINE ORIGINAL BOUNDARY OF MODELED REGION
BLOCK  -25,-20 -25,20 25,20  25,-20
* GENERATE JOINT PATTERN OVER ENTIRE ORIGINAL REGION
JREGION  -25,-25  -25,25 25,25 25,-25
JSET 45,0  200,0  0,0  5.0,0   (0,0)
JSET -9,0  200,0  0,0  5.0,0   (0,0)
* PUT IN JOINTS NEEDED FOR THE LATER EXCAVATION
CRACK  -5.01,-2.51  5.01,-2.51
CRACK  -5.01, 2.51  5.01, 2.51
CRACK  -5,-2.5       -5,2.5
CRACK    5,-2.5       5,2.5
CRACK  2.25,2.5 1.93,5.0
* DELETE SMALL BLOCKS TO IMPROVE TIMESTEP
DELETE -30,30 -30,30 2.0
*
*
* GENERATE FDEF ZONES AND ASSIGN JOINT PROPERTIES (MAT=1 &
* JMAT=1;DEFAULT)
GENERATE -30,30  -30,30     AUTO  9.0
CHANGE    -30,30  -30,30    JCONS=2
PROP MAT=1  D=0.00300  K=39060       G=31780
PROP MAT=1  KN=5000   KS=5000  JKN=20000   JKS=20000
PROP MAT=1  F=0.577   JF=0.577
* ROUND  0.1
*
*
* APPLY BOUNDARY CONDITIONS AND INITIAL CONDITIONS TO
* CONSOLIDATE MODEL UNDER FIELD STRESSES
BOUND LSTRESS=-24.0, 0.0, -24.0 0 0 0 .03 0 .03
INSITU STRESS=-24.0, 0.0 , -24.0 YGRAD=.03 0 .03
GRAV  0.0 -10.0
*
*
* SPECIFY CONDITIONS DURING SOLUTION
DAMP  AUTO
*
*
```

```
* TRACK THE DAMPING, X-DISPLACEMENT, AND Y-DISPLACEMENT OVER
TIME
HIST  N=10 DAMP  XDIS=0,7   YDIS=0,7
* DUMP OUT THE XDIS-VALUE TO SCREEN EVERY 10TH TIMESTEP DUR-
ING CYCLE
CYCLE  100
* SAVE CONSOLIDATED STATE
SAVE ROOFG
PRINT  MAX
* MAKE EXCAVATION
DELETE -5,5  -2.5,2.5
* HOLD Y-VELOCITY TO ZERO TO KEEP BLOCK ASSEMBLAGE FROM
RISING DUE TO
* UNBALANCED REACTIVE FORCES NOW THAT WE HAVE REMOVED MASS
BOUND -26,26  -21,-19  YVEL 0.0
BOUND -26,26 19,21 YVEL 0.0
HIST N=50
CYCLE 2000
* SAVE EXCAVATED STATE
SAVE  ROOF1
PRINT  MAX
* ************************************
*  APPLY SEISMIC LOAD FROM TOP (PEAK VELOCITY=0.04 M/SEC)
*
* SET UP NONREFLECTING BOUNDARY
BOUND MAT=1
BOUND -26 -23 -21 21 XVISC
BOUND 23 26 -21 21 XVISC
BOUND -26 26 -21 -19 XVISC YVISC
BOUND -26 26 19 21 XVISC YVISC
* APPLY SINUSOIDAL STRESS WAVE
BOUND -26 26 19 21 STRESS 0 0 -1.25
BOUND YHIST=COS(100.0,0.0175)
PRINT BOUND
DAMP 0.1 1.0 MASS
RESET TIME HIST DISP ROT
HIST N=10 YDIS (-4.48,2.57)
HIST YDIS (0,2.57) YVEL (0,2.57) YVEL (4,2.57) YVEL(-4.48,2.57)
* MORE HISTORY TRACES TO CHECK EXACTLY WHAT IS BEING APPLIED
AT BOUNDARY
* HISTORY NUMBERS 6-16.
HIST YVEL (0,20) YVEL (25,10) YVEL (25,-10) YVEL (0,-20)
HIST YVEL (-25,-10) YVEL (-25,10)
HIST SXX (25,10) SXX (25,-10) SXX (-25,-10) SXX (-25,10)
HIST SYY (0,20)
CYC 800
```

```
PR MAX
SAVE ASEISMIC
STOP
END
RESTART ROOF1
*   APPLY SEISMIC LOAD FROM TOP (PEAK VELOCITY=0.4 M/SEC)
*
* SET UP NONREFLECTING BOUNDARY
BOUND MAT=1
BOUND -26 -23 -21 21 XVISC
BOUND 23 26 -21 21 XVISC
BOUND -26 26 -21 -19 XVISC YVISC
BOUND -26 26 19 21 XVISC YVISC
* APPLY SINUSOIDAL STRESS WAVE
BOUND -26 26 19 21 STRESS 0 0 -12.5
BOUND YHIST=COS(100.0,0.0175)
PRINT BOUND
DAMP 0.1 1.0 MASS
RESET TIME HIST DISP ROT
HIST N=10 YDIS (-4.48,2.57)
HIST YDIS (0,2.57) YVEL (0,2.57) YVEL (4,2.57) YVEL(-4.48,2.57)
* MORE HISTORY TRACES TO CHECK EXACTLY WHAT IS BEING APPLIED
AT BOUNDARY
* HISTORY NUMBERS 6-16.
HIST YVEL (0,20) YVEL (25,10) YVEL (25,-10) YVEL (0,-20)
HIST YVEL (-25,-10) YVEL (-25,10)
HIST SXX (25,10) SXX (25,-10) SXX (-25,-10) SXX (-25,10)
HIST SYY (0,20)
CYC 800
PR MAX
SAVE BSEISMIC
STOP
END
```

APPLICATION PROBLEM B

OPEN STOPING USING VERTICAL RETREAT

A distinct element simulation of a large blasthole open stoping operation is shown to demonstrate the ability to model sequential mining steps. The model is for a quartzite orebody where the potential instability in the stope back is to be evaluated. Of particular concern is the stress concentration in the crown pillar after mining of the stope is completed.

The model for this example is shown in Fig. B-1. A steeply dipping orebody (average dip of 80°) is modeled between the 990m level and the 1190m level of the mine. A low-angle discontinuous joint set is also oriented at 10° dip and average spacing of 30m. The average thickness of the orebody is 14m. The upper stope, above the 1090m level, is mined first; then, mining of the lower level is completed, leaving a 10m crown pillar.

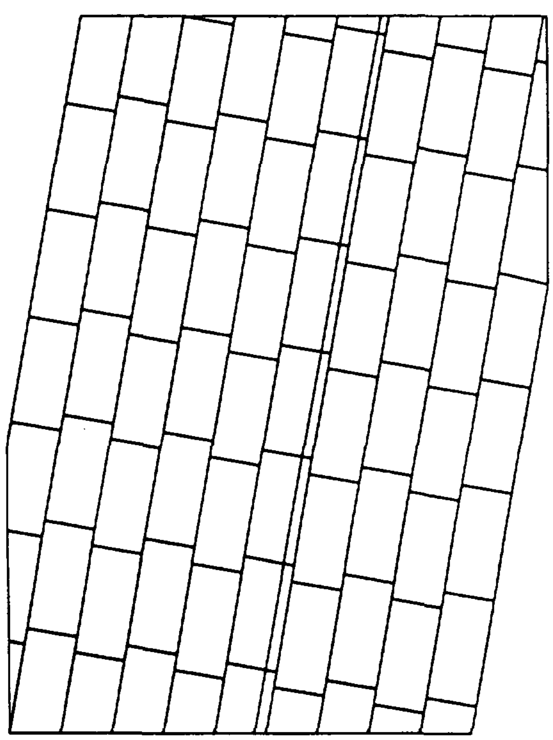


Fig. B-1 Initial Problem Geometry for Blasthole Open Stoping Problem

Four rock types are defined for the model: the hanging wall quartzite, the footwall quartzite, the banded ore, and the weaker schistose ore. Based on the average laboratory test values, the following properties for these rock types were assumed:

	Hanging Wall	Banded Ore	Schistose Ore	Footwall
Young's Modulus (MPa) (MPa)	62	56	40	67
Poisson's Ratio	0.29	0.28	0.33	0.28
Unconfined Compressive Strength (MPa)	186	168	96	198

The following joint properties were estimated:

Joint Normal Stiffness	5000 MPa/m
Joint Shear Stiffness	5000 MPa/m
Joint Friction Angle	27°
Joint Cohesion	0

The pre-mining state of stress was estimated to be hydrostatic with 33 MPa at the 1190m level.

After model consolidation, the mining progressed in five stages. First, the upper level blocks were removed for a stope height of 45m. The lower stope was then mined in four stages of 17m, 15m, 15m and 18m, leaving the 10m crown pillar. The final stress concentration is depicted in Fig. B-2. At this stage, most of the stress is transferred to the abutments. The stress buildup in the crown pillar is shown in Fig. B-3. Note that, at the final stage of the mining sequence, the crown pillar fails and the stresses drop to a residual value. Although backfilling was not simulated in this example, the model can simulate backfill emplacement after excavation and stress histories in the backfill can be monitored similar to that shown in Fig. B-3 for the crown pillar.

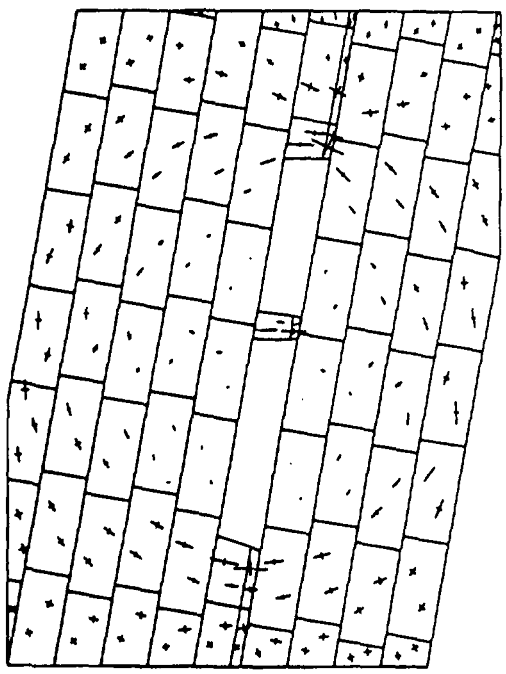


Fig. B-2 Plot of Principal Stresses at End of Mining Sequence

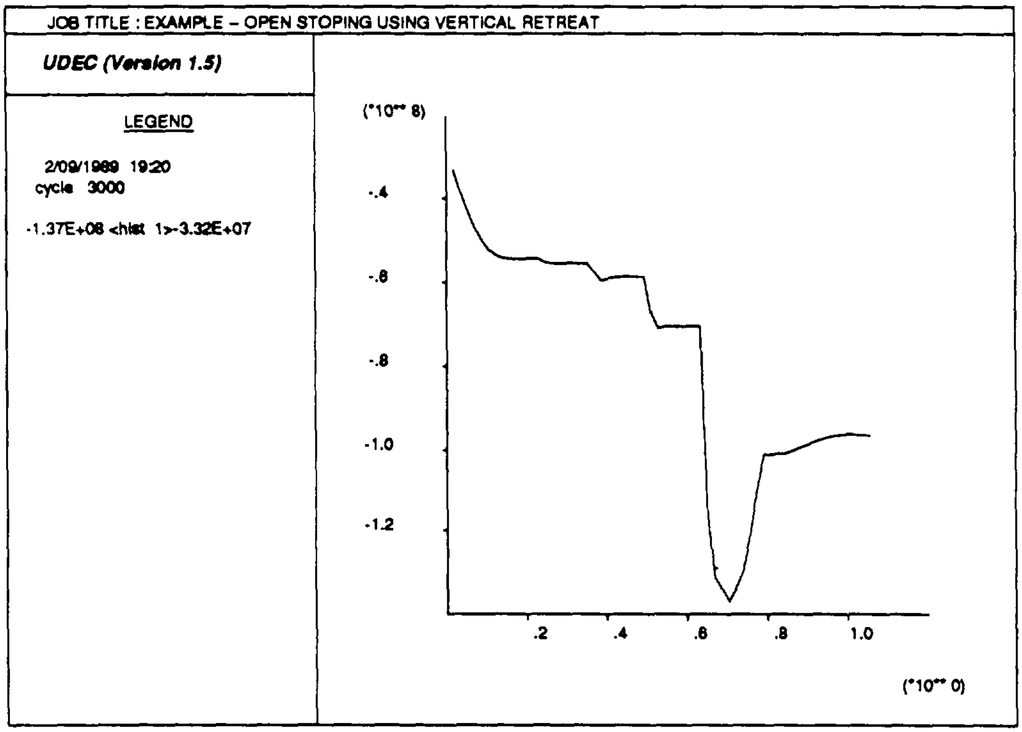


Fig. B-3 Changes in Horizontal Stresses in Crown Pillar with Mining Stages (note: compressive stresses negative)

INPUT COMMANDS FOR OPEN STOPING USING VERTICAL RETREAT EXAMPLE

```
START
HEAD
EXAMPLE -- OPEN STOPING USING VERTICAL RETREAT
WI 0 200 0 200
* ESTABLISH INITIAL GEOMETRY
BLOCK 0 0 0 200 150 200 150 0
JREG 0 0 0 200 150 200 150 0
JSET 80,0 300,0 0,0 14,0 0 0
DEL 0 200 0 200 1210
SPLIT 67.864 0 104.129 200
* ASSIGN MATERIAL PROPETIES HERE
CHANGE MAT=2 CON=3
CHANGE 86 87 0 200 MAT=4 CON=3
CHANGE 75 85.6 0 200 MAT=3 CON=3
CHANGE 87 200 0 200 MAT=5 CON=3
* PUT IN DISCONTINUOUS CROSS JOINTS
JREG 0 0 0 200 150 200 150 0
JSET -10 0 14 0 14 0 30 0 0 0
JSET -10 0 14 0 14 0 30 0 49.48,200
SAVE MINDOL1
* PUT IN JOINTS NEEDED FOR LATER EXCAVATIONS
SPLIT 74 100 89 100
SPLIT 84 155 99 155
SPLIT 63 40 78 35
SPLIT 65 52 81 52
SPLIT 68 67 83 67
SPLIT 71 82 86 82
* CREATE FINITE DIFFERENCE TRIANGLES IN ALL BLOCKS
GEN 0 200 0 200 AUTO 100
* SPECIFY STRESS FIELD
BOUND LSTR -33E6 0 -33E6 0 0 0 2E4 0 2E4
INSITU STR -33E6 0 -33E6 YGRAD 2E4 0 2E4
GRAV 0 -10
```

```
* JOINT PROPERTIES
PROP MAT=1 D=2000 G=26E9 K=44E9
PROP MAT=1 JKN=5E9 JKS=4E9 JFRIC=0.5
PROP MAT=1 KN=5E9 KS=5E9 FRIC=0.5
CHANGE JCON=2 JMAT=1
* HANGING WALL ROCK
PROP MAT=2 D=2000 G=24E9 K=48E9 COH=93E6
* BANDED ORE
PROP MAT=3 D=2000 G=22E9 K=42E9 COH=84E6
* SCHISTOSE ORE
PROP MAT=4 D=2000 G=15E9 K=38E9 COH=48E6
* FOOT WALL
PROP MAT=5 D=2000 G=26E9 K=51E9 COH=99E6
* CONDITIONS DURING SOLUTION
MSCALE ON 2.836E5 5.642E5
DAMP AUTO
* BEGIN EXCAVATION SEQUENCE WITH UPPER STOPE
DEL 78 98 110 155
HIST N=50 SXX 80 105 TYPE 1
HIST SYY 80 105
CYC 600
PR MAX
SAVE MINDOLA3
* EXCAVATE LOWEST PART OF LOWER STOPE
DEL 63 81 35 52
CYC 400
PR MAX
SAVE MINDOLA4
* EXCAVATE NEXT PART OF LOWER STOPE
DEL 65 83 52 67
CYC 400
PR MAX
SAVE MINDOLA5
* EXCAVATE NEXT PART OF LOWER STOPE
DEL 67 86 67 82
CYC 400
PR MAX
SAVE MINDOLA6
```

```
* EXCAVATE LAST PART OF LOWER STOPE
DEL 71 89 82 100
CYC 1200
PR MAX
SAVE MINDOLA9
PR HIS 1 2
STOP
END
```

APPLICATION PROBLEM C

TUNNEL SUPPORT LOADING

This simulation demonstrates the application of UDEC to examine lined tunnels, with specific emphasis on loads developed in the concrete liners.

PROBLEM GEOMETRY

The idealized geometry of the tunnel system is shown in Fig. C-1. The system consists of two tunnels on 12-meter centers at a depth of roughly 70 meters (centerline) beneath the sea bed. The water level is approximately 110 meters above the tunnel centerline. The small (service) tunnel is 5.24 meter diameter with a 37cm-thick concrete liner. The main tunnel is 8.22 meter diameter with a 46cm-thick concrete liner. The service tunnel is driven and lined prior to excavation and lining of the main tunnel.

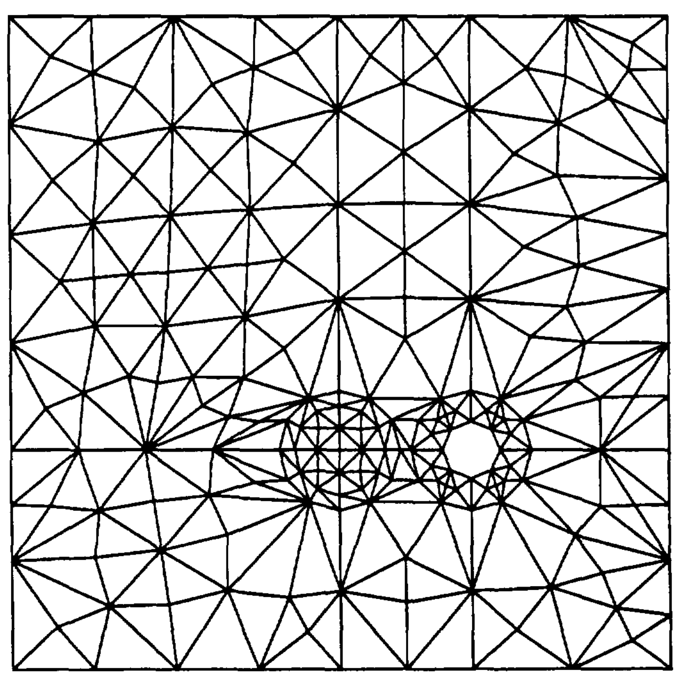


Fig. C-1 Initial Mesh with Service Tunnel Excavated

MATERIAL PROPERTIES

Rock

The tunnels are excavated in rock which has the following material properties:

elastic modulus	133 MPa
Poisson's ratio	0.35
uniaxial compressive strength	8 MPa
cohesion	1 MPa
density	2600 kg/m^3

Concrete Liner

The elastic modulus for the concrete liner is 24 GPa.

CONSTRUCTION STAGES

The sequence of construction activities are:

(1) excavation of service tunnel;

(2) lining of service tunnel;

(3) excavation of main tunnel; and

(4) lining of main tunnel.

The tunnel simulation was run with the steps shown in Fig. C-2.

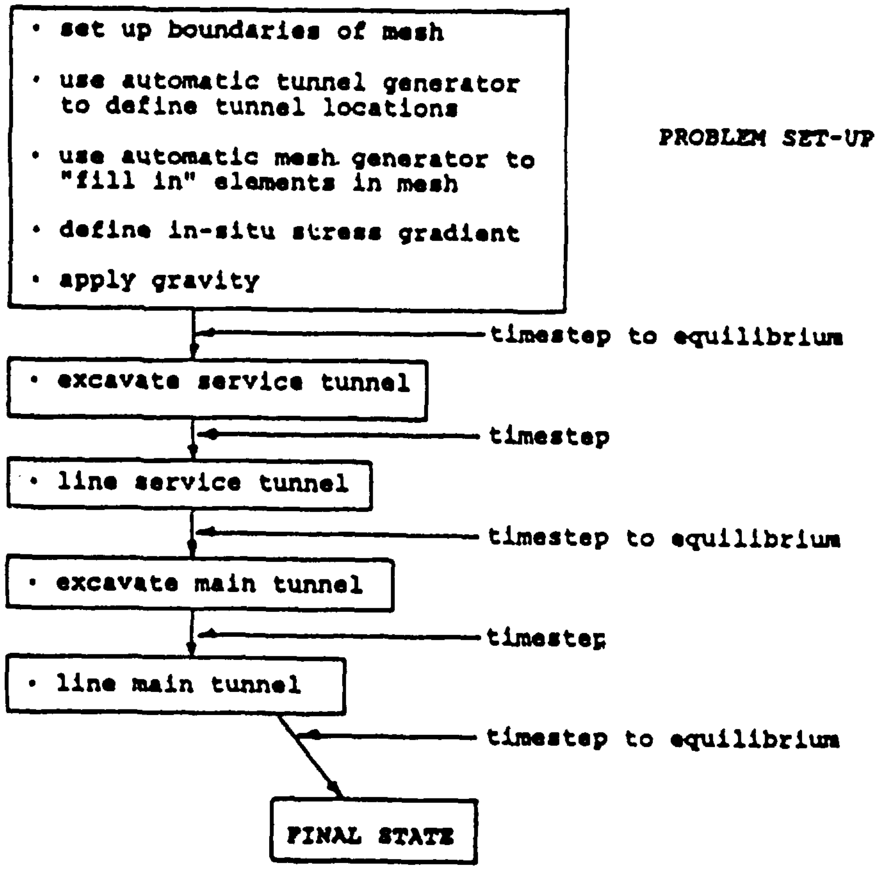


Fig. C-2 Flow Chart for Problem Set-Up and Operation

The mesh used in the calculations is shown in Fig. C-1. The lower and side boundaries are fixed with rollers. The weight of the seawater above the seabed is modeled by applying the equivalent pressure of 30 meters of water head to the top surface of the mesh. Since the tunnels are lined with a waterproof liner, there is no need to perform a transient groundwater flow analysis. The pore water pressure was accounted for by setting the unit weight of the rock to the submerged unit weight. The vertical to horizontal stress ratio was assumed to be hydrostatic.

After gravity stresses have been initialized in the body, the service tunnel is mined and UDEC is cycled until equilibrium is achieved. The resulting elastic displacements are given in Fig. C-3. Next, 16 beam elements are used to model the concrete liner. In the formulation, the lining is discretized into masses which are coupled to the rock via a shear and normal stiffness (Fig. C-4). Each mass has three degrees of freedom — two displacement components and one rotation. As the rock mass deforms, displacements are transferred to the mass in accordance with the interface stiffness. Adjacent beam elements transfer shear and normal forces across their contacts. The axial and shear force and the moment at each node are determined by using the standard stiffness matrix for a beam element:

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

The UDEC program performs all calculations internally; the user need only define the geometry and properties of the support and interface.

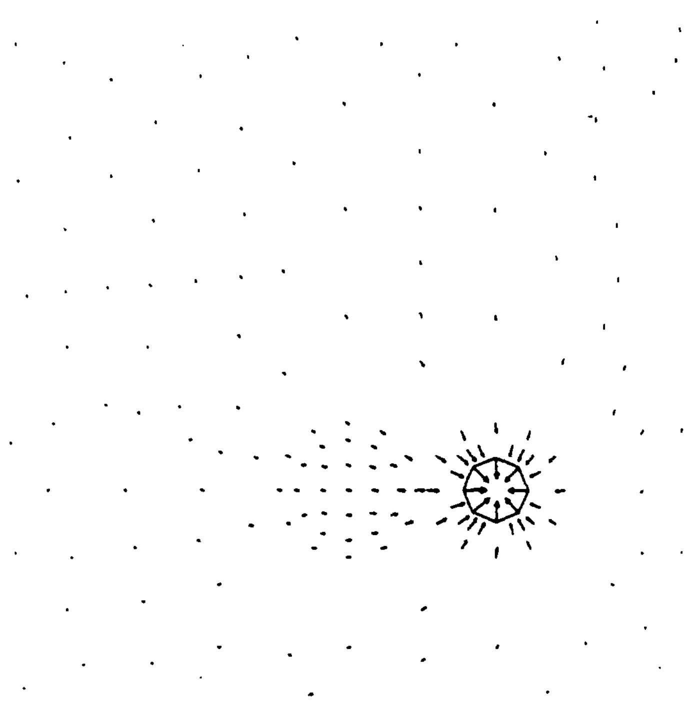


Fig. C-3 Elastic Displacements Due to Excavation of Service Tunnel

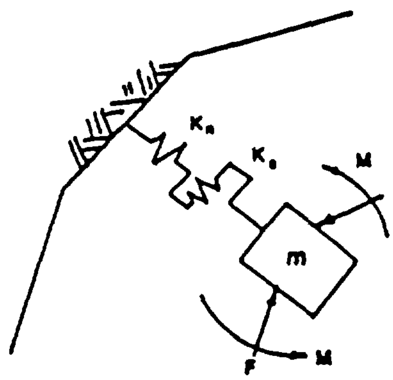


Fig. C-4 Beam Element as Lumped Mass Connected to Rock via Shear and Normal Springs

Figure C-5 is a plot of the finite difference zones and the beam element geometry. The second, main, tunnel is next excavated, which loads the supports in the service tunnel. Timestepping of the solution then continues until equilibrium. Figures C-6 and C-7 illustrate the displacements and principal stress distributions resulting from excavation of the main tunnel. Note that the entire service tunnel translates toward the main tunnel.

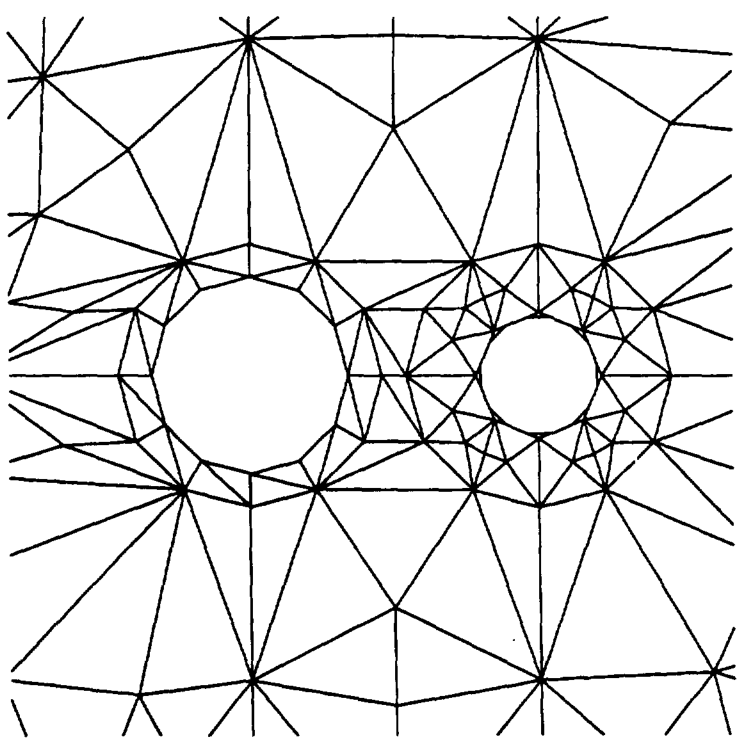


Fig. C-5 Close-Up View of Tunnels and Support in Service Tunnel

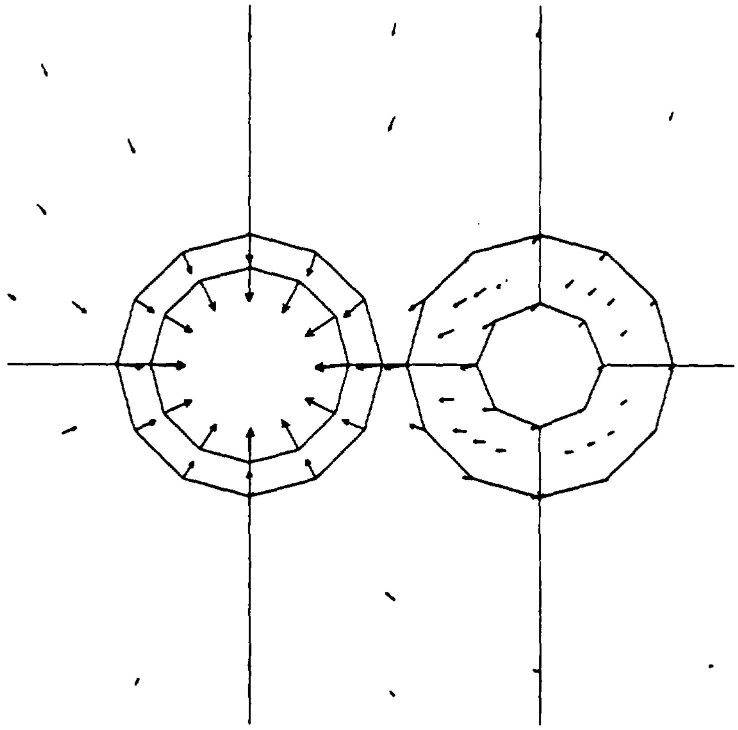


Fig. C-6 Displacements After Mining Main Tunnel

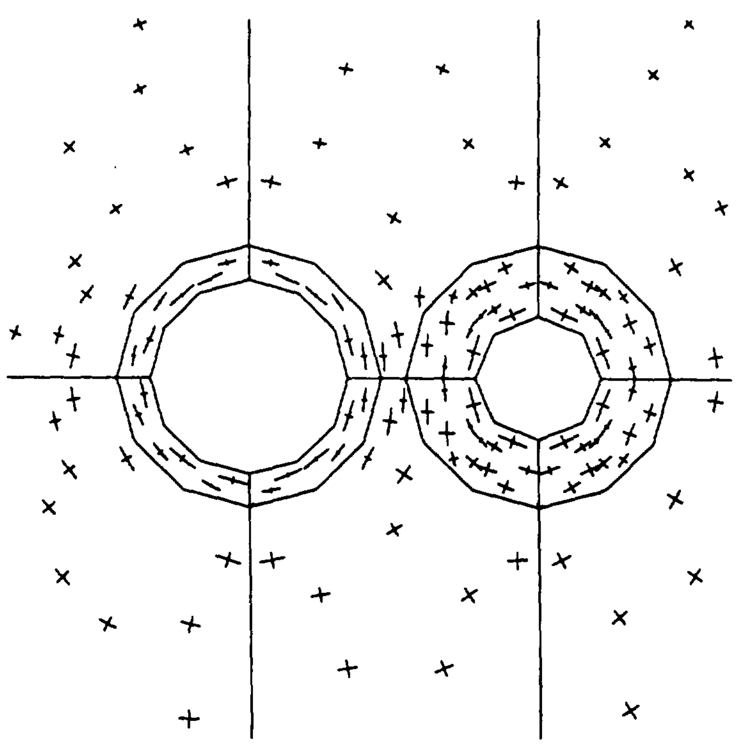


Fig. C-7 Principal Stress Distribution After Mining of Main Tunnel

The greatest axial loads for the service tunnel occurs at the horizontal midplane. At this position, the thrust is $1.7x10^5$N, or an axial stress of 0.46 MPa in the service tunnel. The maximum thrust in the large tunnel is $1.2x10^5$N, or an axial stress of 0.2 MPa. The axial and shear forces and moment distribution for each beam element are given in Table C-1.

Table C-1

FORCES AND MOMENTS IN LINERS

MAIN TUNNEL

ELEMENT	MID-POINT x-coord	MID-POINT y-coord	FORCE axial	FORCE shear	MOM1	MOM2
1	3.368E+01	-7.001E+01	9.901E+04	-4.779E+03	1.437E+04	-2.894E+04
2	3.260E+01	-7.261E+01	1.223E+05	1.755E+04	2.895E+04	2.451E+04
3	3.000E+01	-7.368E+01	1.045E+05	-3.721E+02	-2.444E+04	2.331E+04
4	2.740E+01	-7.260E+01	1.206E+05	-1.606E+04	-2.327E+04	-2.561E+04
5	2.633E+01	-7.000E+01	9.610E+04	3.388E+03	2.564E+04	-1.533E+04
6	2.740E+01	-6.741E+01	4.832E+04	1.026E+04	1.535E+04	1.590E+04
7	3.000E+01	-6.633E+01	2.800E+04	1.575E+03	-1.587E+04	2.066E+04
8	3.260E+01	-6.740E+01	4.810E+04	-1.147E+04	-2.058E+04	-1.437E+04

SERVICE TUNNEL

ELEMENT	MID-POINT x-coord	MID-POINT y-coord	FORCE axial	FORCE shear	MOM1	MOM2
1	4.440E+01	-7.000E+01	1.656E+05	3.893E+03	8.931E+04	-8.558E+04
2	4.422E+01	-7.092E+01	1.587E+05	4.323E+04	8.558E+04	-4.420E+04
3	4.370E+01	-7.170E+01	1.401E+05	9.095E+04	4.420E+04	4.286E+04
4	4.292E+01	-7.222E+01	9.191E+04	5.579E+04	-4.286E+04	9.625E+04
5	4.200E+01	-7.240E+01	1.115E+05	-1.707E+04	-9.625E+04	7.992E+04
6	4.108E+01	-7.222E+01	8.404E+04	-3.491E+04	-7.992E+04	4.654E+04
7	4.030E+01	-7.170E+01	1.044E+05	-9.684E+04	-4.654E+04	-4.607E+04
8	3.977E+01	-7.092E+01	1.257E+05	-4.631E+04	4.607E+04	-9.035E+04
9	3.959E+01	-7.000E+01	1.254E+05	5.047E+03	9.035E+04	-8.552E+04
10	3.977E+01	-6.908E+01	1.060E+05	4.963E+04	8.552E+04	-3.806E+04
11	4.029E+01	-6.830E+01	7.303E+04	8.041E+04	3.806E+04	3.886E+04
12	4.107E+01	-6.778E+01	3.791E+04	4.544E+04	-3.886E+04	8.234E+04
13	4.199E+01	-6.760E+01	3.378E+04	1.661E+04	-8.234E+04	9.823E+04
14	4.292E+01	-6.778E+01	6.005E+04	-6.206E+04	-9.823E+04	3.884E+04
15	4.370E+01	-6.830E+01	1.131E+05	-8.737E+04	-3.884E+04	-4.479E+04
16	4.422E+01	-6.908E+01	1.462E+05	-4.650E+04	4.479E+04	-8.931E+04

TOTAL LINER STRESS

The total stress in the liner is determined by adding the ground-induced load with that induced by the hydrostatic water pressure around the tunnel. The ground loading is calculated by adding the stress induced from axial and bending stresses:

$$\sigma_{ground} = \frac{T}{A} + \frac{Mc}{I} \tag{C-1}$$

where T = thrust (axial force),

A = cross-sectional area,

M = bending moment,

c = distance from outer fiber to neutral axis, and

I = second moment of area = $(1/12)\ bt^3$.

Service Tunnel

$T = 1.7x10^5N$

$A = 0.37m \bullet 1m = 0.37m^2$

$M = 8.9x10^4N\text{-}m$

$c = 0.185m$

$I = (1/12)bt^3 = (1/12) \bullet 1(0.37)^3 = 4.22x10^{-3}m^4$

$$\sigma_{ground} = \frac{1.7x10^5N}{0.37m^2} + \frac{(8.9x10^4N\text{-}m) \bullet (0.185m)}{4.22x10^{-3}m^4}$$

$$\cong 0.46 \text{ MPa} + 3.9 \text{ MPa} = 4.36 \text{ MPa}$$

hydrostatic pressure (assume average depth of 110m)

$\sigma_{water} = P_{hydrostatic} \bullet r/t$

where $P_{hydrostatic}$ = hydrostatic water pressure
r = radius
t = thickness of liner

$\sigma_{water} = 9.8(\text{KPa/m}) \bullet 110m \times (2.62m)/(0.37m)$
$= 1.068 \text{ MPa} \bullet (2.62m/0.37m) = 7.6 \text{ MPa}$

$\sigma_{total} = \sigma_{ground} + \sigma_{water} \cong 4.36 \text{ MPa} + 7.6 \text{ MPa} \cong 12 \text{ MPa}$

Main Tunnel

$$T = 1.2\text{x}10^5\text{N}$$

$$A = 0.46\text{m} \bullet 1\text{ m} = 0.46\text{m}^2$$

$$M = 2.9\text{x}10^4\text{ N-m}$$

$$c = 0.23\text{m}$$

$$I = (1/12)bt^3 = (1/12) \bullet 1(0.46)^3 = 0.008\text{m}^4$$

$$\sigma_{ground} = \frac{1.2\text{x}10^5}{0.46} + \frac{(2.9\text{x}10^4\text{N-m}) \bullet (0.23\text{m})}{8\text{x}10^{-3}\text{ m}^4}$$

$$= 0.26\text{ MPa} + 0.8\text{ MPa} = 1.3\text{ MPa}$$

hydrostatic pressure

$$\sigma_{water} = 1.078\text{ MPa} \bullet (4.11\text{m}/0.46\text{m})$$

$$= 9.6\text{ MPa}$$

$$\sigma_{total} \cong 1.3 + 9.6 \cong 10.9\text{ MPa}$$

DISCUSSION

The thrust distribution varies with angle around the tunnels because the shearing stiffness between the rock and liner was made very large to simulate a "locked" interface. The thrust distribution is consistent with the fact that the entire service tunnel is translating toward the main tunnel. The significant displacement differences around the periphery of the service tunnel lead to larger moments than in the case of the main tunnel, which experiences fairly uniform radial displacements. The compressive stresses resulting from the ground movement are much smaller than those from the hydrostatic water pressure.

INPUT COMMANDS FOR TUNNEL SUPPORT LOADING EXAMPLE

```
START
*
* TUNNEL SUPPORT LOADING
*
BLOCK 0 -30 60 -30 60 -90 0 -90
ROUND 0.1
SPLIT 0 -70 60 -70
SPLIT 30 0 30 -90
SPLIT 42 0 42 -90
TUN 30 -70 4.11 12
TUN 42 -70 2.62  8
TUN 30 -70 5.5 12
TUN 42 -70 5.5 12
GEN 0 90 -50 0 AUTO 10.0
GEN 0 90 -90 -75 AUTO 10.0
GEN 0 90 -75 -50 AUTO 3.0
BOUND STRESS 1.02E5 0  1.02E5 YGRAD 1.34E4 0  1.34E4
INSITU STRESS  1.02E5 0  1.02E5 YGRAD 1.34E4 0 1.34E4
* ROCK PROPERTIES
* PROP MAT=1 D=1340 G=.33E9 K=.99E9 COH=1E6 FRIC=0.577
PROP MAT=1 D=1340 G=.05E9 K=.15E9 COH=1E6 FRIC=0.577
CHANGE MAT=1 CON=3
* ELASTIC JOINT PROPERTIES
PROP  MAT=2  JKN=1E9  JKS=1E9  KN=1E6  KS=1E6  JFRIC=10.0  FRIC=10.0
JTENS=1E6
PROP MAT=2 TENS=1E6
CHANGE JMAT=2 JCON=2
DAMP AUTO
GRAVITY 0 -10
CYC 10
BOUND -1 1 -91 0 XVEL=0.0
BOUND -1 90 -91 -89 YVEL=0.0
BOUND 59 61 -91 0 XVEL=0.0
DEL 40 44 -72 -68
SAVE CHAN1
* HISTORIES AROUND TUNNEL 1
HIST N=20 YDIS 42 -67 SXX 42 -67 TYPE 1
HIST YDIS 42 -73 SXX 42 -73
HIST XDIS 39 -70 SYY 39 -70
HIST XDIS 45 -70 SYY 39 -70
```

```
* HISTORIES AROUND TUNNEL 2
HIST YDIS 30 -65.0 SXX 30 -65.0
HIST YDIS 30 -75.0 SXX 30 -75.0
HIST XDIS 25.0 -70 SYY 25.0 -70
HIST XDIS 35.0 -70 SYY 35.0 -70
RESET DISP JDISP
CYC 600
SAVE CHAN2
PR MAX
STRUCT 42 -70 11.25 3.0 16 5 0.37
PROP MAT=5 D=2400 G=10E9 K=13E9 KN=1E8 KS=1E7 CFRIC=10.0
DEL 28 32 -72 -68
HIST TYPE 15
RESET DISP
CYC 5000
SAVE CHAN3
PR MAX
PR STE STC LM
STRUC 30 -70 22.5 4.3 8 5 0.46
* PROP MAT=1 D=1340 G=.05E9 K=.15E9 COH=1E6 FRIC=.577
HIS TYPE 15
RESET DISP
CYC 5000
PR MAX
PR STE STC LM
SAVE CHAN4
```

APPLICATION PROBLEM D

GRAVITY DAM: FLUID FLOW AND DYNAMIC LOADING

This demonstration problem involves analysis of a 100m-high concrete gravity dam on a jointed rock foundation. Joint spacing was assumed to be 50m, with joints oriented at 20 and -70°. Two loading conditions are studied. First, the effects of filling the reservoir are studied, including an analysis of fluid flow through the rock joints. Second, a dynamic wave is applied to the base of the model to study potential effects of an earthquake-type loading. The numerical analysis was performed in the following sequence.

State 1: Gravity Loads — Empty Reservoir

An in-situ state of stress of $\sigma_H = 0.5\ \sigma_V$ was assumed in the rock mass. Displacements in this stage result from the weight of the dam. The water table was assumed to be at y = 0 (i.e., the base level of the dam). Stresses in the foundation are effective stresses.

State 2: Full Reservoir

During this stage, the water table was assumed to be at the top of the dam, exerting hydrostatic pressure on the upstream side of the dam and rock foundation. The base of the model was fixed with respect to y-displacement, and the sides of the model were fixed with respect to x-displacement. The following conditions were assumed with respect to fluid flow.

1. Joint contacts along the bottom and sides of the model were assumed to have zero permeability.
2. On the rock surface upstream of the dam, the head was fixed at 100m (0.95 MPa) by using the **BOUND PP** command. Downstream, the head was set to zero.
3. The interface between the dam and rock foundation was assumed to have low permeability.
4. The experimental algorithm (Set SFlow) for steady flow was used.

The results are shown in Figs. D-1 to D-5.

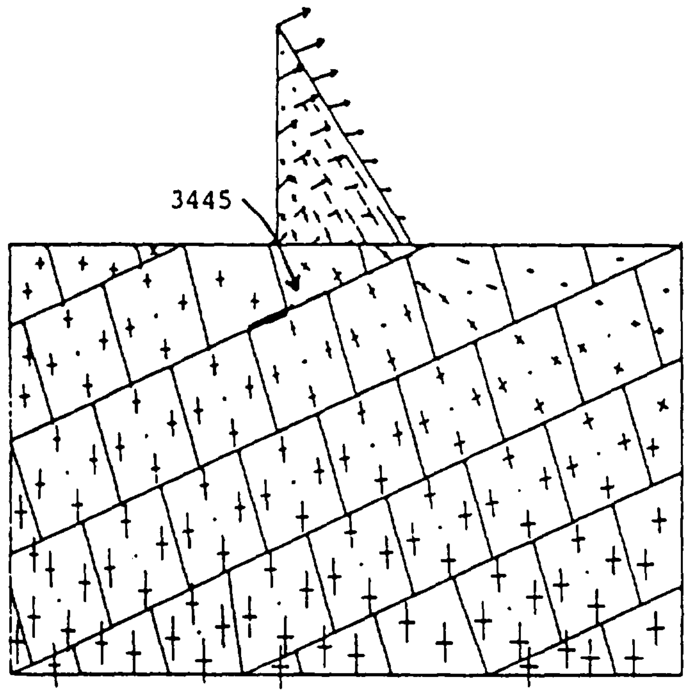


Fig. D-1 Block Plot With Principal Stresses, Displacements, and Relative Joint Shear Displacements. (The displacements result from Stage 2 loading (i.e., reservoir filling) only. Location of Joint Contact 3445 is shown for later study.)

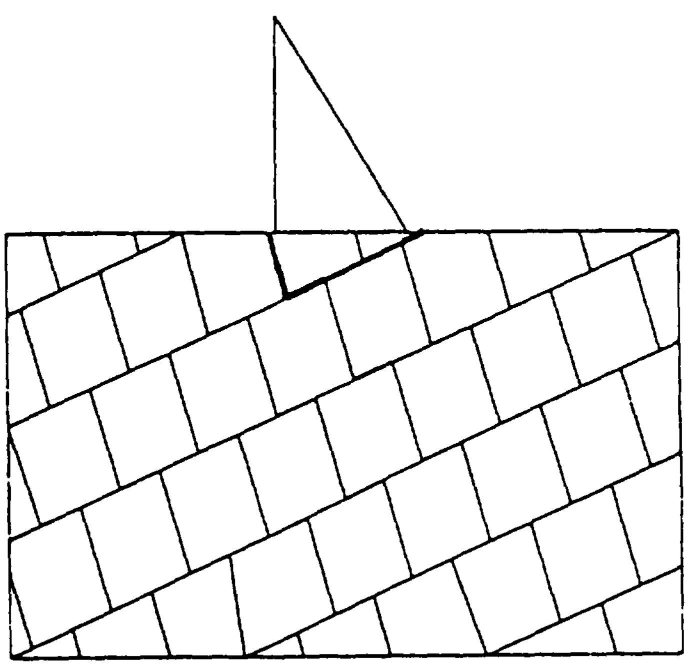


Fig. D-2 Plot of Flow Rates Showing That Most Flow Is Concentrated in One Joint

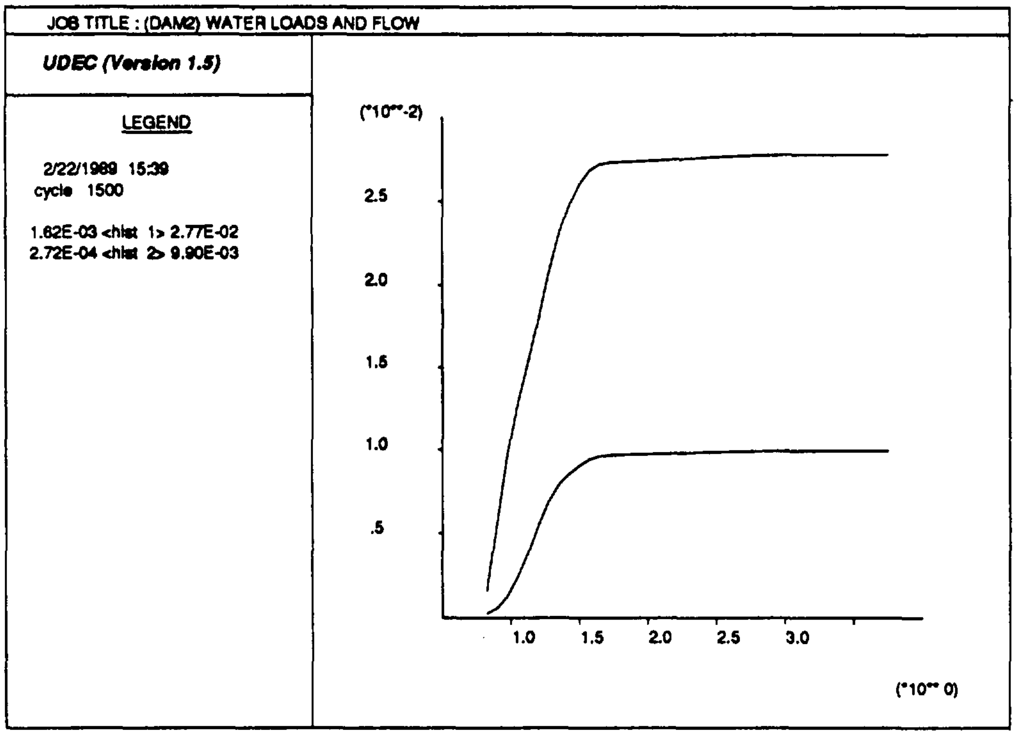


Fig. D-3 x- and y-Components of Displacement at Crest of Dam During Reservoir Filling Showing That Equilibrium Conditions Have Been Achieved

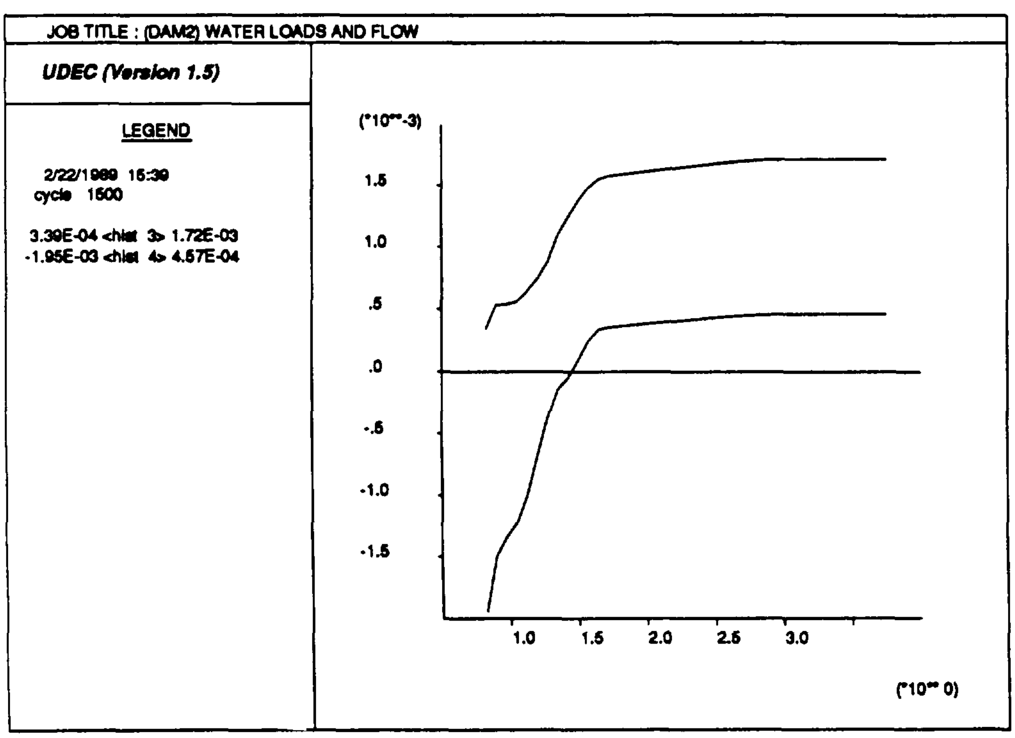


Fig. D-4 Shear and Normal Displacement of Joint 3445 Showing Joint Opening (indicated by positive normal displacement)

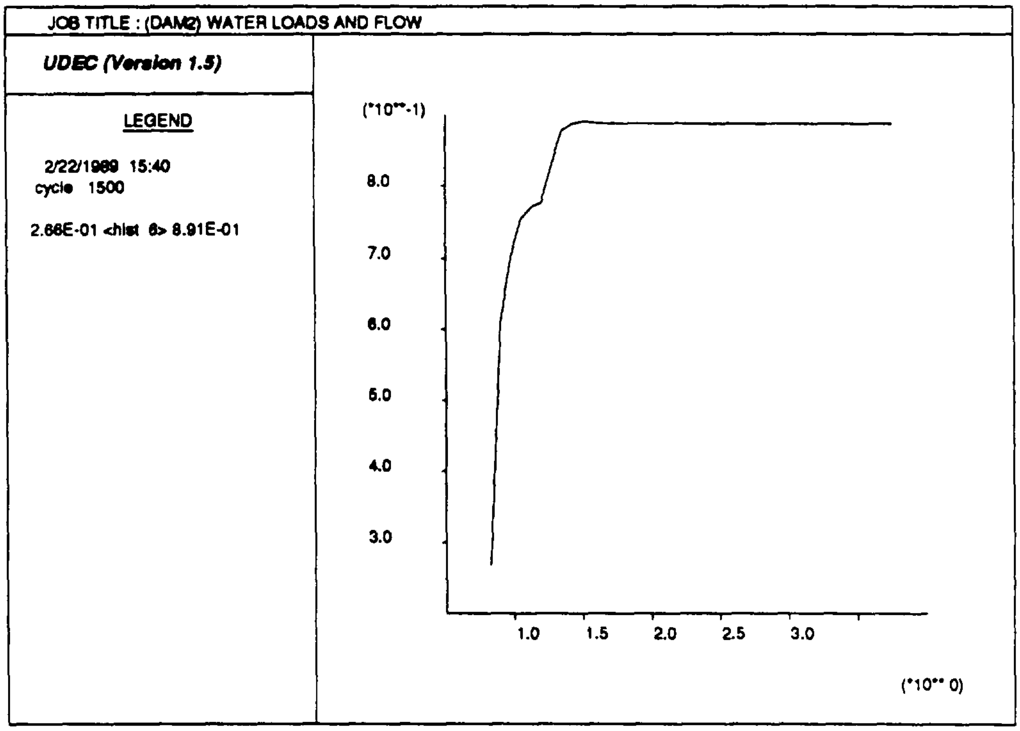


Fig. D-5 Water Pressure in Domain 4358 Next to Contact 3445 Showing That Steady State Conditions Have Been Achieved

Stage 3: Dynamic Loading

In this stage, a vertical propagating sinusoidal shear wave (Freq = 5 Hz) is applied for 1.5 seconds to the base of the model. The following boundary conditions were used:

Base — Stress corresponding to shear was applied along with non-reflecting viscous boundaries to limit reflections.

Sides — Nodes on the sides of the foundation were connected to a 1-D free-field calculation. The same loading used at the base of the model was applied to the bottom of the free field. The free field was discretized into 20 zones with elasto-plastic behavior assumed.

The fluid flow calculation was switched off during this stage. This is analogous to assuming that fluid pressures remained constant during the 1.5 second dynamic loading.

Figures D-6 to D-9 show the results of the analysis.

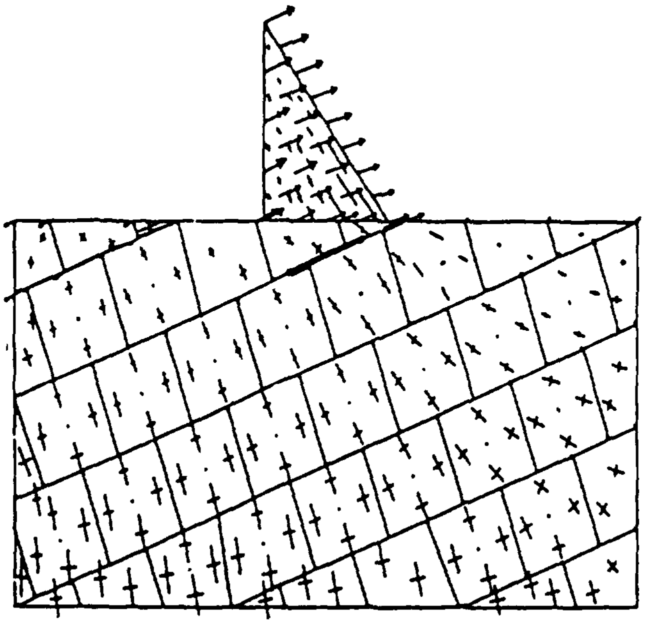


Fig. D-6 Displacements, Principal Stresses, and Joint Shear Displacement in the Problem Resulting from Dynamic Loading. (Note the large amount of slip (0.07m) along the first joint at 20°). The joint upstream at 70° is open (i.e., no effective stress). The large displacement of this wedge indicates the probable failure of the dam.)

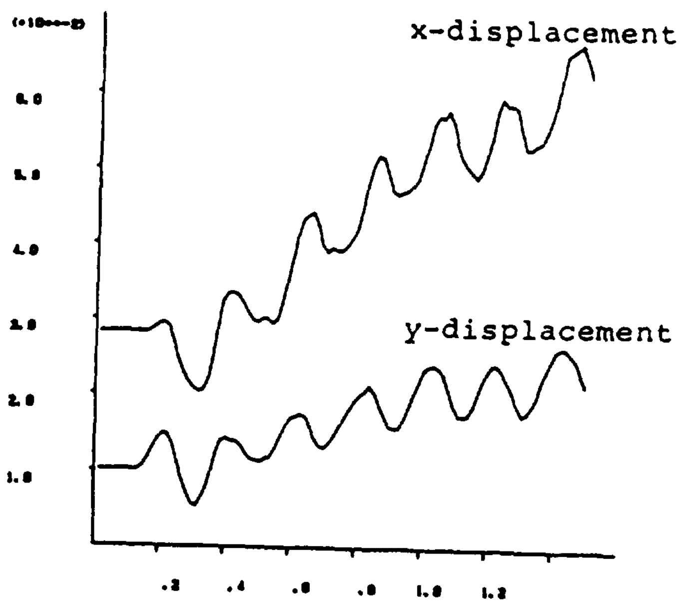


Fig. D-7 Displacements at Dam Crest With Time Showing Accumulation of Horizontal Displacement With Each Load Cycle

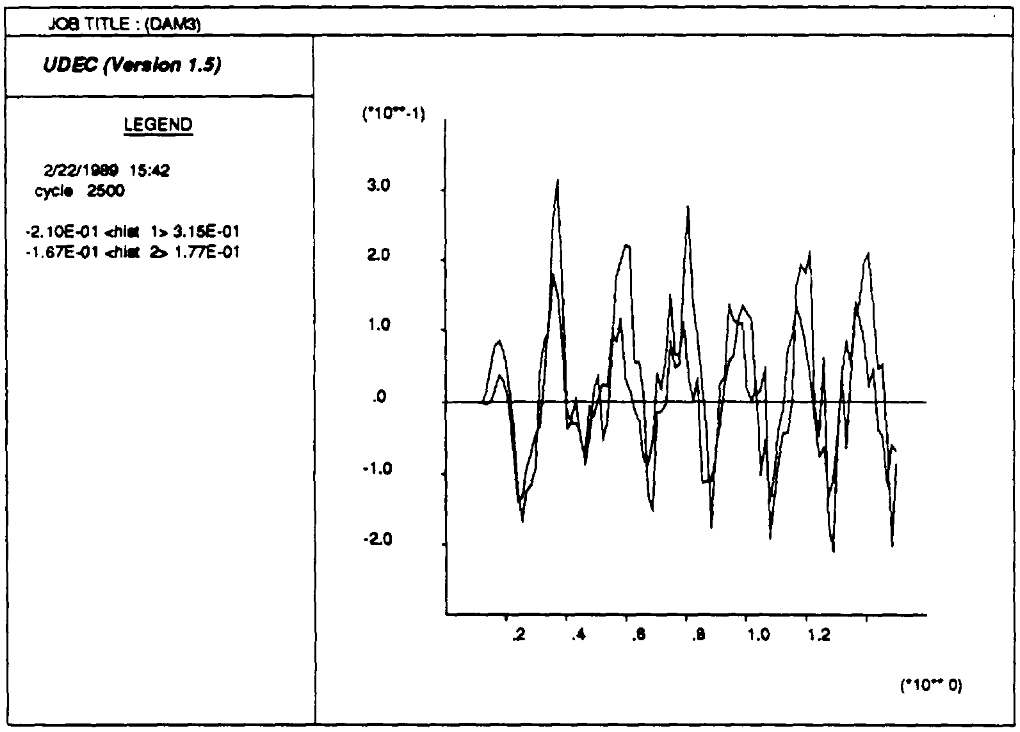


Fig. D-8 Velocity History of Dam Crest With Time

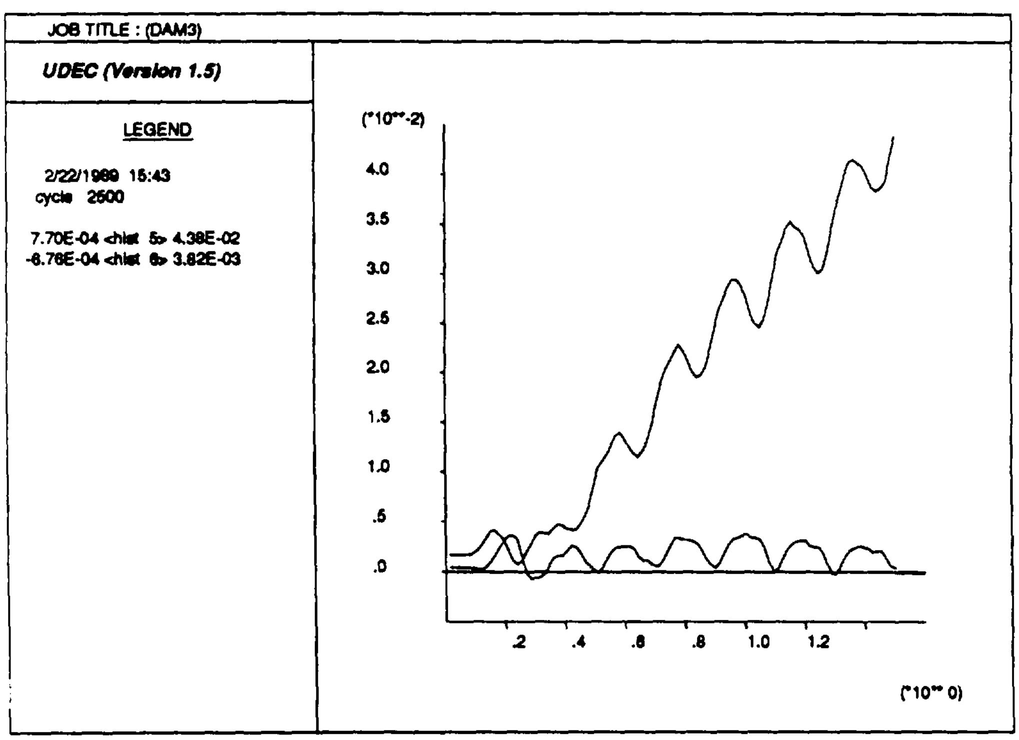


Fig. D-9 Joint Shear and Normal Displacements With Time at Contact 3353, Again Showing Accumulation of Joint Shear Displacement

INPUT COMMANDS FOR GRAVITY DAM: FLUID FLOW AND DYNAMIC LOADING

```
START
*
* DAM (100M) --- DISCONTINUOUS JOINTS : 20 AND -70 DEG.
* GRAVITY LOADING --- INSITU STRESSES (K=0.5)
* EFFECTIVE STRESSES --- FREE-FIELD (20 NODES)
*
HEADING
(DAM1) IN-SITU STRESSES ; GRAVITY
*
* ROCK BLOCKS ; JOINTS (NO COHESION)
PROP MAT=1 D=0.00265 K=33333 G=20000
+       JKN=1000 JKS=1000 JF=0.577
* DAM AND FOUNDATION JOINT
PROP MAT=2 D=0.00240 K=16667 G=12500
+       JKN=1000 JKS=1000 JF=0.577 JCOH=2 JTENS=2
* FREE FIELD (NO COHESION) ; JOINTS (NO COHESION)
PROP MAT=3 D=0.00265 K=33333 G=20000 F=0.577 COH=0.2
+       JKN=1000 JKS=1000 JF=0.577
* FREE FIELD (COHESION) ; JOINTS (COHESION)
PROP MAT=4 D=0.00265 K=33333 G=20000 F=0.577 COH=2
+       JKN=1000 JKS=1000 JF=0.577 JCOH=2
* VISCOUS BOUNDARIES (EQUIVALENT ELASTIC PROPERTIES) ; JOINTS
(COHESION)
PROP MAT=5 D=0.00265 K=11680 G=11111
+       JKN=1000 JKS=1000 JF=0.577 JCOH=2
*
ROUND 0.5
* SET MINIMUM EDGE LENGTH
SET EDGE 8.0
* SET MINIMUM CONTACT LENGTH
SET CLEMIN=5.0
*
WINDOW -210,210 -210,110
BLOCK -200,-200 -200,100 200,100 200,-200
*
```

```
* STRUCTURE : GRAVITY DAM
SPLIT  -201,0  201,0
SPLIT  -40,-1  -40,101
SPLIT   40,0  -40,100
DELETE  -200,-40  0,100
DELETE   40,200   0,100
*
JREGION  -200,-200  -200,0  200,0  200,-200
JSET     20,0  800,0   0,0  50,0  ( 50,0)
JSET    -70,0   50,0  50,0  50,0  (-50.99, 16.45)
JSET    -70,0   50,0  50,0  50,0  (-10.40,-21.98)
*
CHANGE   -200,200  -200,0   FDEF  MAT=1  CONS=1
CHANGE   -40,40    0,100   FDEF  MAT=2  CONS=1
GENERATE -200,200  -200,0   AUTO  60
GENERATE  -40,40   0,100    AUTO  30
*
* ALL JOINTS
CHANGE -210,210 -210,10    JMAT=1  JCONS=2
* COHESION BELOW Y=-150
CHANGE -210,210 -210,-150  JMAT=4  JCONS=2
* FOUNDATION JOINT
CHANGE -41,41 -1,1  ANG=-5,5  JMAT=2  JCONS=2
*
* BOUNDARY CONDITIONS : LATERAL : FREE-FIELD ; BOTTOM : Y-FIXED
BOUND  MAT=5
BOUND  -201,-199  -201,1    XVISC YVISC FF
BOUND   199,201   -201,1    XVISC YVISC FF
BOUND  -201,201   -201,-199  YVEL=0
*
* GENERATE FREE-FIELD (20 NODES) ; COHESION BELOW Y=-150 ; FIXED
BOTTOM
FFIELD  GEN (-200,0) 20
FFIELD  -150,0     MAT=3  CONS=3
FFIELD  -200,-150  MAT=4  CONS=3
FFIELD  XVEL=0 YVEL=0
*
```

```
* SET GRAVITY ; SET IN-SITU STRESSES (EFFECTIVE)
GRAV  0  -9.8
INSITU -210,210 -210,0  YGRAD= 0.008085, 0, 0.016170
DAMP  AUTO
*
* SET FLUID BULK MODULUS TO ZERO (NO FLOW, ONLY BUOYANCY
FORCES)
FLUID  0.001  0.0
SET  YWTAB 0.0
*
* SET FAST CONTACT LOGIC
SET  UPCON  5
SET  DSCAN  6
*
* SET PARTIAL MASS SCALING
FRAC  0.1  1.0
MSCALE  PART  1.5E-3
*

HIST  N=50  TYPE=2  XDIS=-40,100  YDIS=-40,100
*
CYCLE   500
*
PRINT  MAX
PRINT  -50,55  -35,2  CON
*
SAVE  DAM1
*
RESTART DAM1
*
*  WATER LOADS AND FLOW
*
HEAD
(DAM2) WATER LOADS AND FLOW
*
RESET  DIS  ROT  JDIS  HIST
*
```

```
* RESET AUTO DAMPING
DAMP 0.0 1.0 MASS
DAMP AUTO
*
* FLOW PROPERTIES
* ABOVE Y=-150
PROP MAT=1 JPERM=3.0E8 AZERO=0.001 ARES=0.0005
PROP MAT=3 JPERM=0.0   AZERO=0.001 ARES=0.0005
*
* BELOW Y=-150
PROP MAT=4 JPERM=3.0E8 AZERO=0.001 ARES=0.0005
PROP MAT=5 JPERM=0.0   AZERO=0.001 ARES=0.0005
*
* FOUNDATION JOINT
PROP MAT=2 JPERM=3.0E8 AZERO=0.0002 ARES=0.0001
*
* SET MAX. APERTURE
SET CAPRAT=2.0
*
* SET LATERAL AND BOTTOM BOUNDARY CONTACTS TO ZERO
PERMEABILITY
CHANGE -201,-199 -150,1 JMAT=3
CHANGE  199,201  -150,1 JMAT=3
CHANGE -201,-199 -201,-150 JMAT=5
CHANGE  199,201  -201,-150 JMAT=5
CHANGE -201,201  -201,-199 JMAT=5
*
* FIX HEAD UPSTREAM OF DAM (FULL RESERVOIR) ; DOWNSTREAM IS
ZERO
BOUND -201,-39 -1,1 PP=0.980
*
* APPLY VERTICAL WATER LOAD UPSTREAM OF DAM
BOUND -201,-39 -1,1 STRESS= 0, 0, -0.980
*
* APPLY HORIZONTAL LOAD TO DAM
BOUND -40.1,-39.9 -0.1,100.1 LSTRESS=-0.980,0,0 0,0,0 0.0098,0,0
*
```

```
* FIX LATERAL BOUNDARIES (HORIZONTALLY)
BOUND -201,-199 -201,1   XVEL=0
BOUND  199,201  -201,1   XVEL=0
BOUND -201,-199 -199,-1  YFREE
BOUND  199,201  -199,1   YFREE
*
* SWITCH OFF FREE-FIELD
FFIELD OFF
*
* contact 3445 is at x=-33.42 y=-30.47
HIST N=25 XDIS(-40,100) YDIS(-40,100) TYPE=5
HIST      SDIS(3445) NDIS(3445)
* (FLOW RATE THRU CONTACT)
HIST      ADD(3445,21)
* (DOMAIN HEAD) x=-22,-26
HIST      ADD(4358,4)
*
* FLUID PROPERTIES (REAL BULK MODULUS)
FLUID 0.001 2000.0
*
* USE MASS SCALING TO INCREASE TIME STEP
FRAC 0.1 1.0
MSCALE PART 3.0E-3
*
* SWITCH ON FAST FLOW LOGIC
SET SFLOW
SET PTOL=0.02
*
CYCLE 500
*
CYCLE 500
PRINT MAX
PRINT -50,55 -35,2 CON FLOW DOM
*
SAVE DAM2
*
RESTART DAM2
*
```

```
* DYNAMIC LOADING : SHEAR WAVE AT BASE OF MODEL
*
* WITH DOMAIN PRESSURES ; NO FLUID FLOW
*
HEADING
(DAM3)
*
RESET TIME HIST
*
* SWITCH ON FREE-FIELD AGAIN
FFIELD ON INIT
*
* DYNAMIC BOUNDARY CONDITIONS : VISCOUS BOUNDARIES AND FREE-
FIELD
BOUND MAT=5
BOUND -201,-199 -201,1 XVISC YVISC FF
BOUND 199,201 -201,1 XVISC YVISC FF
BOUND -201,201 -201,-199 XVISC YVISC
*
* AMPLITUDE OF SHEAR WAVE : 0.2 MPA.
BOUND -201,201 -201,-199 STRESS 0, 0.4, 0
*
* FREQ. = 5 HZ.
BOUND HIST=SIN(5.0,10.0)
*
* BOUND. COND. AT BASE OF FREE-FIELD
FFIELD SXY=1 XVISC YVISC
*
* FIX Y-VEL AT BOTTOM
BOUND -201,201 -201,-199 YVEL=0
*
HIST N=10 XVEL(-40,100) YVEL(-40,100)
HIST      XDIS(-40,100) YDIS(-40,100)
HIST      SDIS(3445) NDIS(3445) SSTR(3445) SRAT(3445)
*
* SWITCH OFF FLOW
FLUID 0.001 0.0
*
```

```
* DYNAMIC DAMPING
DAMP 0.10 2.0 MASS
*

* PARTIAL MASS SCALING
FRAC 0.15 1.0
MSCALE PART 1.5E-3
*
CYCLE 1000
*
PRINT MAX FF
PRINT -50,55 -35,2 CON
*
SAVE DAM3
STOP
```

APPLICATION PROBLEM E

CONTINUOUSLY-YIELDING JOINT CONSTITUTIVE RELATION (JCONS=3)

The use of the continuously-yielding joint constitutive relation (JCONS=3) is demonstrated in the following examples of direct shear tests. The continuously-yielding model (described in Section 3.3.1) is considered more "realistic" than the standard Mohr-Coulomb joint model (JCONS=2) in that the continuously-yielding model attempts to account for some non-linear behavior observed in physical tests (such as joint shearing damage, normal stiffness dependence on normal stress, and decrease in dilation angle with plastic shear displacement) The essential features of the continuously-yielding model include the following.

1. The curve of shear stress/shear displacement is always tending toward a "target" shear strength for the joint — i.e., the instantaneous gradient of the curve depends directly on the difference between strength and stress.

2. The target shear strength decreases continuously as a function of accumulated plastic displacement (a measure of damage).

3. Dilation angle is taken as the difference between the apparent friction angle (determined by the current shear stress and normal stress) and the residual friction angle.

As a consequence of these assumptions, the model exhibits, automatically, the commonly-observed peak/residual behavior of rock joints. Also, hysteresis is displayed for unloading and reloading cycles of all strain levels, no matter how small.

The model is described as follows:

$$k_n = a_n \sigma_n^{(e_n)} \tag{E-1}$$

$$\sigma_n := \sigma_n + k_n \Delta u_n \tag{E-2}$$

$$k_s^{(o)} = a_s \sigma_n \; s^{(e_s)} \tag{E-3}$$

$$\tau_m = \sigma_n \tan\phi_{eff} \bullet \mathrm{sign}(\Delta u_s) \tag{E-4}$$

Initially, $\phi_{eff} = (\phi_{eff})^{o}$; then

$$\Delta\phi_{eff} = -\frac{1}{R}(\phi_{eff} - \phi)\ \Delta u_s{}^{(p)} \quad \text{(E-5)}$$

If

$$\text{sign}\ (\Delta u_s) = \text{sign}\ [\Delta u_s^{(old)}],$$

then

$$r = \tau/\tau_m \quad \text{(E-6)}$$

$$\Delta u_s^{(old)} = \Delta u_s \quad \text{(E-7)}$$

$$F = (\tau_m - \tau)/[\tau_m(1-r)] \quad \text{(E-8)}$$

$$\tau := \tau + \Delta u_s \bullet F \bullet k_s^{(o)} \quad \text{(E-9)}$$

$$u_s^{(p)} := u_s^{(p)} + |\Delta u_s| \bullet \max(0,\ 1-F) \quad \text{(E-10)}$$

$$i = \max[0,\ \tan^{-1}(|\tau|/\sigma_n) - \phi] \quad \text{(E-11)}$$

$$\sigma_n := \sigma_n + k_n |\Delta u_s|\ \tan(i) \quad \text{(E-12)}$$

where $a_n,\ e_n,\ a_s,\ e_s,\ R$ = the joint model parameters,

ϕ = the intrinsic friction angle,

ϕ_{eff} = the effective friction angle,

k_n = the normal stiffness defined as a function of σ_n,

$k_s^{(o)}$	=	the initial shear stiffness defined as a function of σ_n,
τ	=	the shear stress on the joint,
τ_m	=	the failure or "bounding" shear stress,
Δu_s	=	the current shear displacement increment,
$\Delta u_s^{(old)}$	=	the previous shear displacement increment,
$u_s^{(p)}$	=	the accumulated plastic shear displacement,
r	=	the stress ratio at the last reversal ($r=0$, initially), and
i	=	the effective dilation angle.

The stiffness functions defined by Eqs. (E-1) and (E-3) are the most simple functions consistent with the experimental data. More complex functions (such as hyperbolic laws) may be substituted, if desired.

The effective friction angle (ϕ_{eff}) is to be thought of as the friction angle that would apply if no damage were done (i.e., if no more asperities are sheared off). However, before the corresponding strength can be mobilized, some shear displacement is necessary (which reduces ϕ_{eff}).

The parameter R, which has the dimension of displacement, controls the rate at which ϕ_{eff} decreases with plastic shear displacement. A small value of R causes ϕ_{eff} to decrease rapidly and vice versa. In a more complex model, R should depend on σ_n.

State variable r is a "memory" factor that is responsible for returning the shear stiffness to its initially-high value after reversals in strain rate. When the joint model is installed in UDEC, a limit of 0.75 is imposed on r. This prevents very small fluctuations in strain rate (numerical noise) from forcing the stiffness to very high values. A more satisfactory solution would be to have memory for all reversals, but the model would entail the use of far more computer memory.

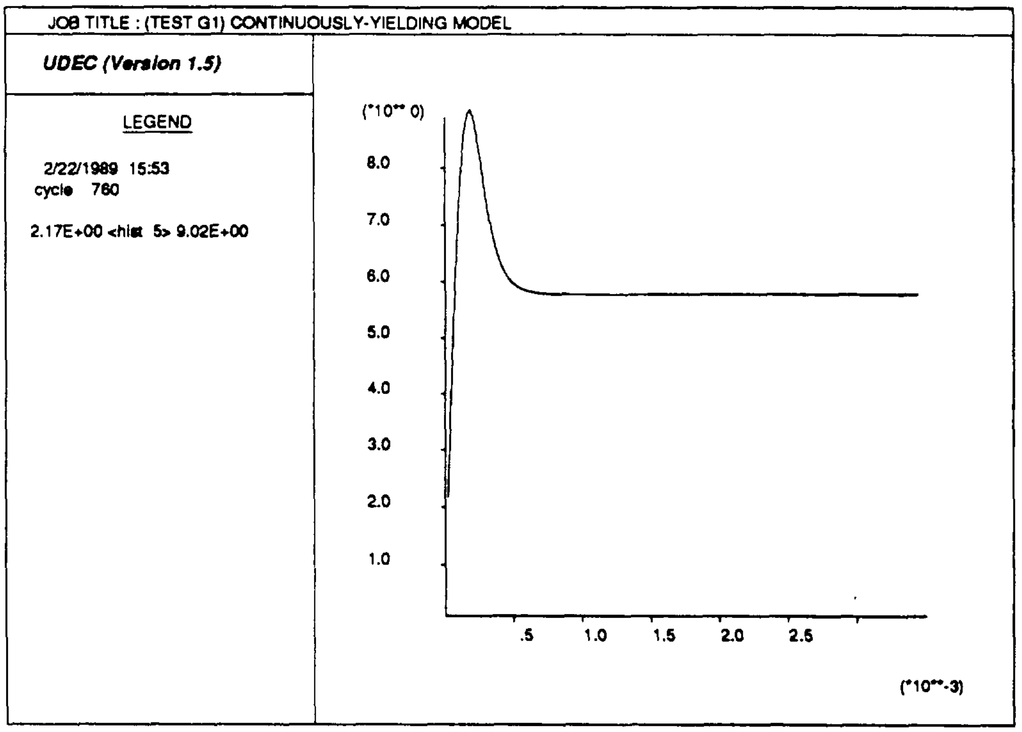
The plastic displacement, $u_s^{(p)}$, always increases. It is evaluated on that part of the applied displacement remaining after the displacement associated with the initial stiffness [given by Eq. (3)] is removed. When using the model to match experimental results, the initial value of ϕ_{eff} can be used as a parameter. This corresponds, physically, to initial pre-shearing, or damage, of the sample.

In the two examples shown here, the effect of two different assumptions concerning $(\phi_{eff})^{\circ}$ [the joint initial friction (JIF) angle] are examined. In the first example, the initial friction angle is assumed to be 59.3° (1.035 radians); in the second, the initial joint friction angle is taken be to 40.1° (0.700 radians). The following problem parameters were the same for both examples.

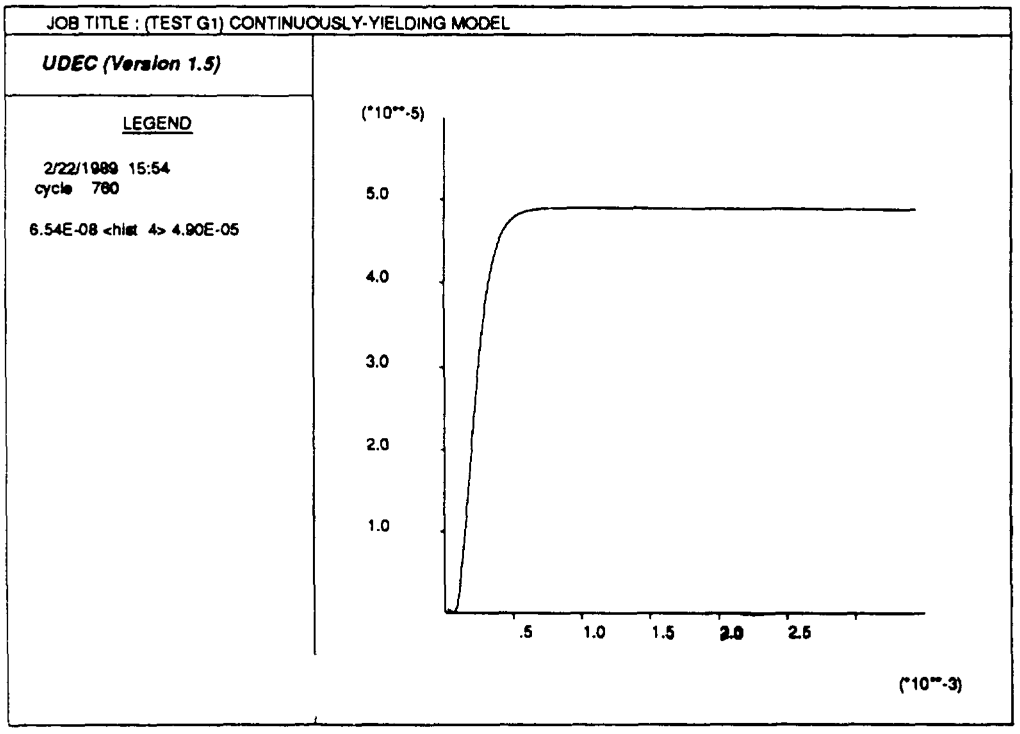
UDEC Parameter	Name	Value
D	density	2600 kg/m^3
K	bulk modulus	4 GPa
G	shear modulus	3 GPa
JKN	a_n (joint normal stiffness)	100 GPa/m
JKS	a_s (joint shear stiffness)	100 GPa/m
JEN	e_n (joint normal stiffness exponent)	0.0
JES	e_s (joint shear stiffness exponent)	0.0
JFRIC	$\tan\phi$ (joint intrinsic friction coefficient)	0.577 (ϕ=30°)
JR	joint roughness parameter	0.1 mm

The direct shear tests were run on a 2m square block which was split horizontally at mid-height. The normal stress on the joint was constant at 10 MPa.

Figure E-1(a) shows a plot of shear stress versus shear displacement for the case where the joint initial friction was 59.3 degrees. The normal displacement versus shear displacement plot for this assumption is shown in Fig. E-1(b). Similar plots of shear stress and normal displacement vs shear displacement for joint initial friction = 40.1° are given in Figs. E-2(a) and (b).

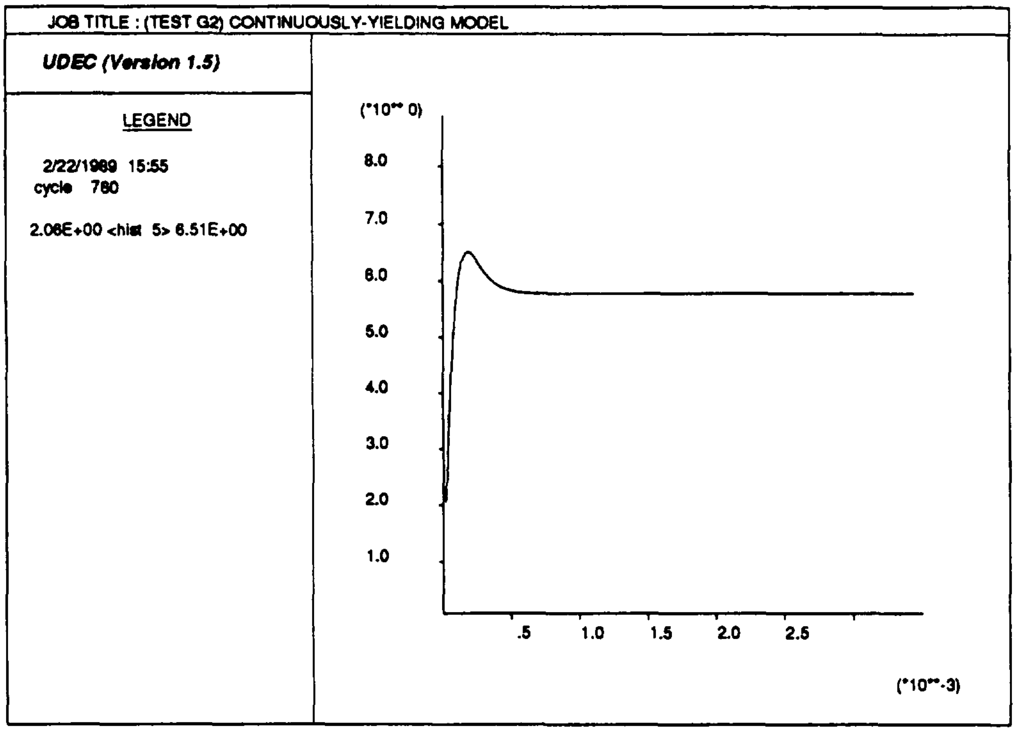


(a) plot of shear stress (MPa) vs shear displacement (m)

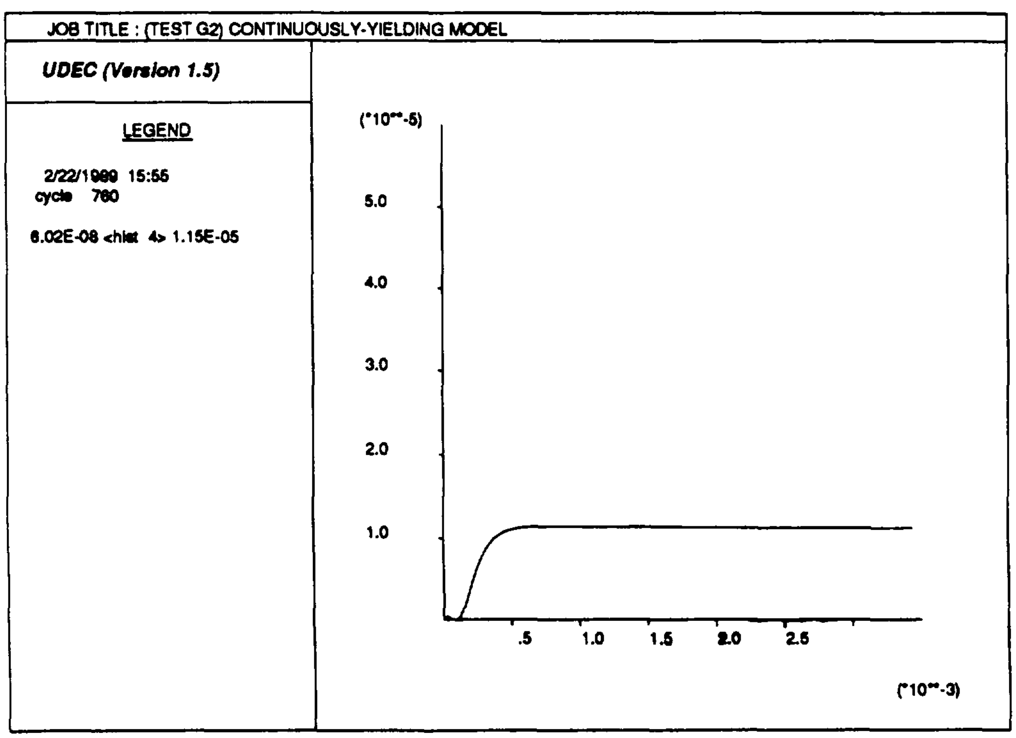


(b) plot of normal displacement (m) vs shear displacement (m)

Fig. E-1 Results of Direct Shear Test Using Continuously-Yielding Joint Constitutive Relation with Initial Joint Friction at 59.3°



(a) plot of shear stress (MPa) vs shear displacement (m)



(b) plot of normal displacement (m) vs shear displacement (m)

Fig. E-2 Results of Direct Shear Test Using Continuously-Yielding Joint Constitutive Relation with Initial Joint Friction at 40.1°

INPUT COMMANDS FOR CONTINUOUSLY-YIELDING JOINT CONSTITUTIVE RELATION (JCONS = 3)

```
START
*
*  CONTINUOUSLY-YIELDING JOINT MODEL
*
*  DIRECT SHEAR TEST
*
*  RUN 1 : HIGH PEAK STRESS
*
HEAD
(TEST G1) CONTINUOUSLY-YIELDING MODEL
*
PROP  MAT=1  D=2.60E-3  K=4000  G=3000
+   KN=10000  KS=10000  JKN=100000  JKS=100000  JEN=0.0  JES=0.0
+   JFRIC=0.577  JIF=1.035  JR=1.0E-4
*
WINDOW  -0.1,2.1  -0.1,2.1
ROUND  0.001
BLOCK  0,0  0,2  2,2  2,0
*
CRACK  -0.1,1  2.1,1
CHANGE  0,100  0,100  FDEF  MAT=1  CONS=1
GEN    0,100  0,100  AUTO  2.1
CHANGE  JMAT=1  JCONS=3
*
INSITU  STRESS    0, 0, -10
BOUND  MAT=1
BOUND  -0.1,2.1  -0.1,0.1  YVEL=0
BOUND  -0.1,2.1  -0.1,2.1  STRESS  0 , 0 , -10
*
DAMP  AUTO
FRAC  0.05   1.0
CYC   10
*
```

```
HIST N=5  XDIS(2,2) YDIS(2,2) TYPE=5
HIST      SDIS(159) NDIS(159) SSTR(159) NSTR(159)
RESET TIME
*
* RESET NORMAL DISPLACEMENTS
RSET 0.0 159 16
RSET 0.0 190 16
*
PR CON BOUND
*
* FIX BOTTOM BLOCK ; IMPOSE VELOCITY ON TOP BLOCK
BOUND COR=133,107    XVEL=0 YVEL=0
BOUND COR=146,120    XVEL= -0.05
*
CYC 750
*
SAVE TESTG1
*
PR CON MAX
*
* PLOT SHEAR STRESS-SHEAR DISPLACEMENT CURVE
* PLOT YWIN=0.1,8.9 XREV ABC=3 HIST 5
*
* PLOT DILATANCY-SHEAR DISPLACEMENT CURVE
* PLOT YWIN=0.01E-5,5.99E-5 XREV ABC=3 HIST 4
*
* PRINT MAX. OF STRESS CURVE
PR HMAX 5
*
START
*
*  CONTINUOUSLY-YIELDING JOINT MODEL
*
*  DIRECT SHEAR TEST
*
*  RUN 2 : LOW PEAK STRESS
*
HEAD
(TEST G2) CONTINUOUSLY-YIELDING MODEL
```

```
*
PROP MAT=1 D=2.60E-3 K=4000  G=3000
+   KN=10000 KS=10000 JKN=100000  JKS=100000 JEN=0.0 JES=0.0
+   JFRIC=0.577 JIF=0.700 JR=1.0E-4
*
WINDOW -0.1,2.1 -0.1,2.1
ROUND 0.001
BLOCK 0,0 0,2 2,2 2,0
*
CRACK -0.1,1 2.1,1
CHANGE 0,100 0,100 FDEF MAT=1 CONS=1
GEN   0,100 0,100 AUTO 2.1
CHANGE JMAT=1 JCONS=3
*
INSITU STRESS   0, 0, -10
BOUND MAT=1
BOUND -0.1,2.1 -0.1,0.1 YVEL=0
BOUND -0.1,2.1 -0.1,2.1 STRESS 0 , 0 , -10
*
DAMP AUTO
FRAC 0.05  1.0
CYC  10
*
HIST N=5  XDIS(2,2) YDIS(2,2) TYPE=5
HIST     SDIS(159) NDIS(159) SSTR(159) NSTR(159)
RESET TIME
*
* RESET NORMAL DISPLACEMENTS
RSET 0.0 159 16
RSET 0.0 190 16
*
PR CON BOUND
*
* FIX BOTTOM BLOCK ; IMPOSE VELOCITY ON TOP BLOCK
BOUND COR=133,107    XVEL=0 YVEL=0
BOUND COR=146,120    XVEL= -0.05
*
CYC 750
*
```

```
SAVE TESTG1
*
PR CON MAX
*
* PLOT SHEAR STRESS-SHEAR DISPLACEMENT CURVE
* PLOT  YWIN=0.1,8.9  XREV  ABC=3  HIST 5
*
* PLOT DILATANCY-SHEAR DISPLACEMENT CURVE
* PLOT  YWIN=0.01E-5,5.99E-5  XREV  ABC=3  HIST 4
*
* PRINT MAX. OF STRESS CURVE
PR HMAX 5
STOP
```

APPLICATION PROBLEM F

LOAD CYCLING A SPECIMEN WITH A SLIPPING CRACK

Several investigators have proposed simple conceptual models of a single, closed crack to explain phenomena associated with the deformational response of jointed rock (e.g., Walsh, 1965, and Jaeger and Cook, 1976). One such model is a single crack embedded in an elastic solid subjected to a cycle of uniaxial compression. Olsson (1982) shows that the stress-strain relation for this model is composed of three distinct components (shown in Fig. F-1):

(1) a loading segment (OA) which features elastic deformation and slip along the crack;

(2) an initial unloading segment (AB), where the crack does not slip; and

(3) a final unloading segment, again with elastic deformation and slip.

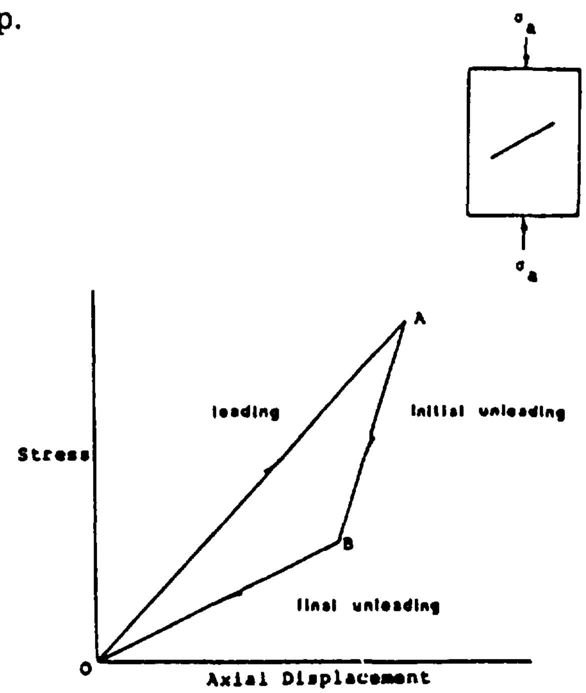
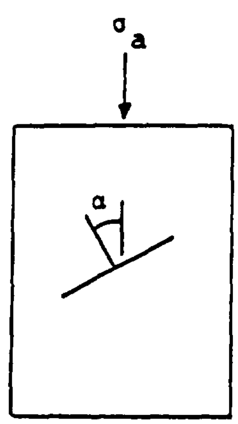


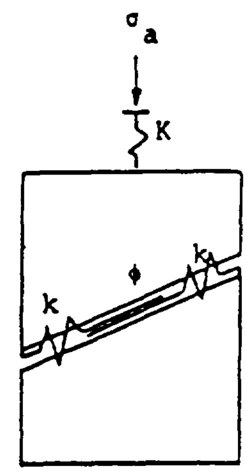
Fig. F-1 Stress-Displacement Relation for Elastic Specimen with Embedded Crack Subjected to Uniaxial Load Cycle (after Olsson, 1982)

This response will occur provided that the angle between the direction of loading and the normal to the crack plane is smaller than the angle of frictional resistance of the crack.

Brady et al. (1985) present relations for these slopes in terms of the elastic stiffness of the solid, the elastic and frictional properties of the crack, and the orientation of the crack with respect to the uniaxial load. The model and equivalent systems are illustrated in Figs. F-2(a) and (b).



(a) specimen with embedded crack



(b) equivalent system

Fig. F-2 Conceptual Model of Elastic Specimen Containing Embedded Crack

This conceptual model was simulated with UDEC for the following problem conditions. A specimen containing a single crack oriented at $\alpha = 45°$ was subject to a cycle of uniaxial compression. The intact portion of the specimen has an elastic modulus, E_i, of 89 GPa and Poisson's ratio, ν_i, of 0.26. The crack has an elastic joint stiffness, k_n, normal to the plane of the crack of 198 GPa/m and a shear stiffness, k_s, parallel to the plane of the crack of 220 GPa/m. These properties are approximately representative of the response of a basalt joint.

For the example shown here, two choices for frictional resistance of the crack ($\phi=50°$ and $\phi=16°$) were studied. Figure F-3 confirms that the initial unload slope for $\phi=16°$ matches the slope for $\phi=50°$.

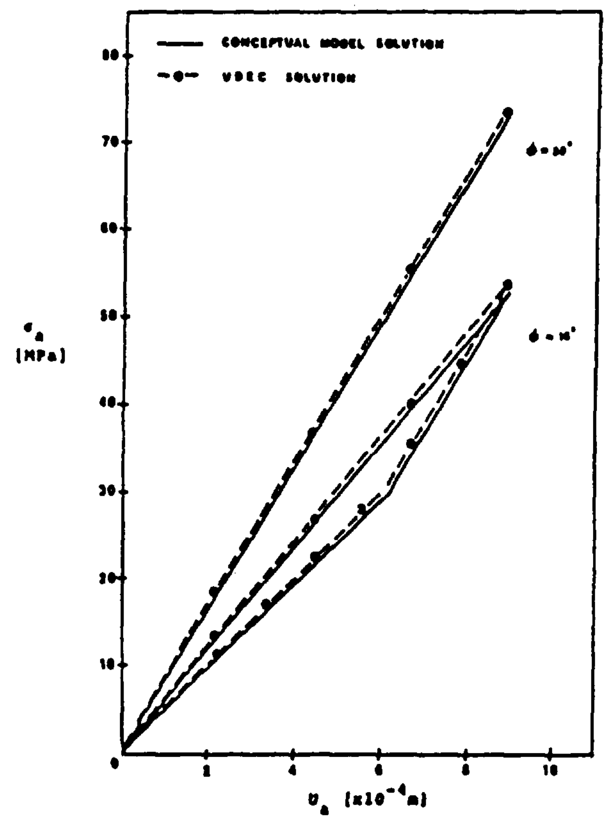


Fig. F-3 Comparison of Results for Slipping Crack Model (For UDEC, U_a is y-displacement at top of the model, and σ_a is the average yy-stress in the model.)

REFERENCES

Brady, B.H.G., M. L. Cramer and R. D. Hart. "Preliminary Analysis of a Loading Test on a Large Basalt Block" (Tech. Note), Int. J. Rock Mech. Min. Sci. & Geomech. Abst., 22, 345-348 (1985).

Jaegar, J. C., and N.G.W. Cook. Fundamentals of Rock Mechanics, 2nd ed., pp. 329-333. London: Chapman and Hall, 1976.

Olsson, W. A. "Experiments on a Slipping Crack," Geophys. Res. Letters, 9(8), pp. 797-800 (1982).

Walsh, J. B. "The Effect of Cracks on the Compressibility of Rock," J. Geophys. Res., 70, 381-389 (1965).

INPUT COMMANDS FOR LOAD CYCLING A SPECIMEN WITH A SLIPPING CRACK

```
START
*
* EXAMPLE APPLICATION PROBLEM
* LOAD CYCLING A SPECIMEN WITH A SLIPPING CRACK
* (FRICTION ANGLE = 16 DEGREES)
*
* (CRACK EXTENSION - NO SLIP)
PROP M=1 D=2850 K=61.8E9 G=35.3E9 JKN=198.0E9 JKS=220.0E9 JF=100.
+    (KN=15E9 KS=15E9 F=100.)
* (CRACK PROPERTIES)
PROP M=2 D=2850 K=61.8E9 G=35.3E9 JKN=198.0E9 JKS=220.0E9 JF=0.287
+    (KN=15E9 KS=15E9 F=0.287)
ROUND 0.01
WINDOW -0.1,1.1 -0.1,2.1
BLOCK 0,0 0,2 1,2 1,0
SPLIT 0,.5 1,1.5
CHANGE FDEF
GEN 0,1 0,2 AUTO 0.5
CHANGE JMAT=1 JCON=2
CHANGE 0.20,0.80 0.74,1.28 JMAT=2
DAMP AUTO
HIST N=50 YDIS 0.5,0.0 YDIS 0.5,2.0
HIST TYPE 1
* Y-DISP. INCREMENT (LOAD STEP 1)
BOUND -.1,1.1 -.1,.1  YVEL= 1.202E-1
BOUND -.1,1.1 1.9,2.1 YVEL=-1.202E-1
CYCLE 200
BOUND -.1,1.1 -.1,.1  YVEL= 0.0
BOUND -.1,1.1 1.9,2.1 YVEL= 0.0
CYCLE 100
PRINT MAX
PRINT CONT
*
```

```
* Y-DISP. INCREMENT (LOAD STEP 2)
BOUND -.1,1.1 -.1,.1  YVEL= 1.202E-1
BOUND -.1,1.1 1.9,2.1 YVEL=-1.202E-1
CYCLE 200
BOUND -.1,1.1 -.1,.1  YVEL= 0.0
BOUND -.1,1.1 1.9,2.1 YVEL= 0.0
CYCLE 100
PRINT MAX
PRINT CONT
*
* Y-DISP. INCREMENT (LOAD STEP 3)
BOUND -.1,1.1 -.1,.1  YVEL= 1.202E-1
BOUND -.1,1.1 1.9,2.1 YVEL=-1.202E-1
CYCLE 200
BOUND -.1,1.1 -.1,.1  YVEL= 0.0
BOUND -.1,1.1 1.9,2.1 YVEL= 0.0
CYCLE 100
PRINT MAX
PRINT CONT
*
* Y-DISP. INCREMENT (LOAD STEP 4)
BOUND -.1,1.1 -.1,.1  YVEL= 1.202E-1
BOUND -.1,1.1 1.9,2.1 YVEL=-1.202E-1
CYCLE 200
BOUND -.1,1.1 -.1,.1  YVEL= 0.0
BOUND -.1,1.1 1.9,2.1 YVEL= 0.0
CYCLE 100
PRINT MAX
PRINT CONT
*
PRINT HIST 1 2
RESET HIST
HIST N=50 YDIS 0.5,0.0 YDIS 0.5,2.0
HIST TYPE 1
*
```

```
* Y-DISP. INCREMENT (UNLOAD STEP 1)
BOUND -.1,1.1 -.1,.1  YVEL=-0.601E-1
BOUND -.1,1.1 1.9,2.1 YVEL= 0.601E-1
CYCLE 200
BOUND -.1,1.1 -.1,.1  YVEL= 0.0
BOUND -.1,1.1 1.9,2.1 YVEL= 0.0
CYCLE 100
PRINT MAX
PRINT CONT
*
* Y-DISP. INCREMENT (UNLOAD STEP 2)
BOUND -.1,1.1 -.1,.1  YVEL=-0.601E-1
BOUND -.1,1.1 1.9,2.1 YVEL= 0.601E-1
CYCLE 200
BOUND -.1,1.1 -.1,.1  YVEL= 0.0
BOUND -.1,1.1 1.9,2.1 YVEL= 0.0
CYCLE 100
PRINT MAX
PRINT CONT
*
* Y-DISP. INCREMENT (UNLOAD STEP 3)
BOUND -.1,1.1 -.1,.1  YVEL=-0.601E-1
BOUND -.1,1.1 1.9,2.1 YVEL= 0.601E-1
CYCLE 200
BOUND -.1,1.1 -.1,.1  YVEL= 0.0
BOUND -.1,1.1 1.9,2.1 YVEL= 0.0
CYCLE 100
PRINT MAX
PRINT CONT
*
* Y-DISP. INCREMENT (UNLOAD STEP 4)
BOUND -.1,1.1 -.1,.1  YVEL=-0.601E-1
BOUND -.1,1.1 1.9,2.1 YVEL= 0.601E-1
CYCLE 200
BOUND -.1,1.1 -.1,.1  YVEL= 0.0
BOUND -.1,1.1 1.9,2.1 YVEL= 0.0
CYCLE 100
PRINT MAX
PRINT CONT
*
```

```
* Y-DISP. INCREMENT (UNLOAD STEP 5)
BOUND -.1,1.1 -.1,.1  YVEL=-1.202E-1
BOUND -.1,1.1 1.9,2.1 YVEL= 1.202E-1
CYCLE 200
BOUND -.1,1.1 -.1,.1  YVEL= 0.0
BOUND -.1,1.1 1.9,2.1 YVEL= 0.0
CYCLE 100
PRINT MAX
PRINT CONT
*
PRINT HIST 1 2
SAVE SLIP
STOP
END
```

APPLICATION PROBLEM G

CEMENT GROUTING SIMULATION

The Bingham fluid model is widely accepted as an appropriate model for cement based grouts [see, for example, Littlejohn (1982) Hassler et al. (1987), and Lombardi (1985)]. This simulation demonstrates use of the Bingham fluid model in UDEC. The problem geometry in Fig. 1 represents a horizontal section in a regularly-jointed rock mass in which a cylindrical hole (1.2m diameter) has been made. The rock mass is assumed to be subject to an initial biaxial in-situ stress (σ_{xx} = 0.2 MPa and σ_{yy} = 0.1 MPa). Grout injection is simulated by maintaining specified pressures within the hole. The pressure is increased in 2000 Pa increments and flow conditions examined at each stage. The hypothetical properties used for the rock, discontinuities and grout are shown in Table 1.

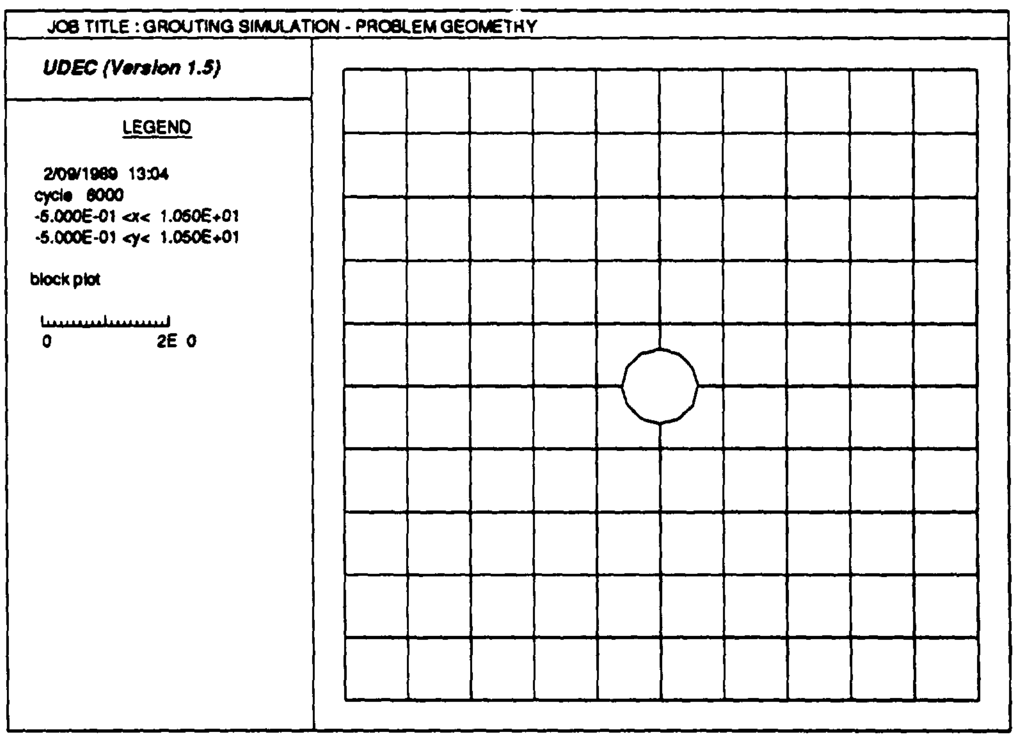


Fig. G-1 Problem Geometry — Grouting Simulation

Table 1

MATERIAL PROPERTIES USED IN CEMENT GROUTING SIMULATION

<u>Intact Rock</u>		
bulk modulus	10	GPa
shear modulus	3	GPa
density	3000	kg/m^3
<u>Joints</u>		
normal stiffness	10	GPa/m
shear stiffness	10	GPa/m
friction angle	45°	
joint permeability constant	$1x10^8$ Pa^{-1} sec^{-1}	
aperture at zero normal stress	0.1	mm
residual aperture at high stress	0.05	mm
<u>Grout</u>		
cohesion	0.1	Pa
density	1000	kg/m^3
bulk modulus	2	GPa

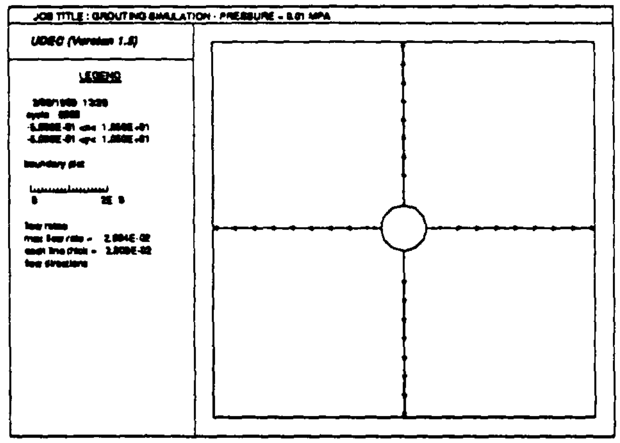
No steady state flow occurs until the pressure in the hole exceeds 8000 Pa. The flow plots for pressures of 10,000 Pa, 12,000 Pa, and 14,000 Pa are shown in Fig. 2. The corresponding changes in grout pressure in joints are shown in Fig. 3.

REFERENCES

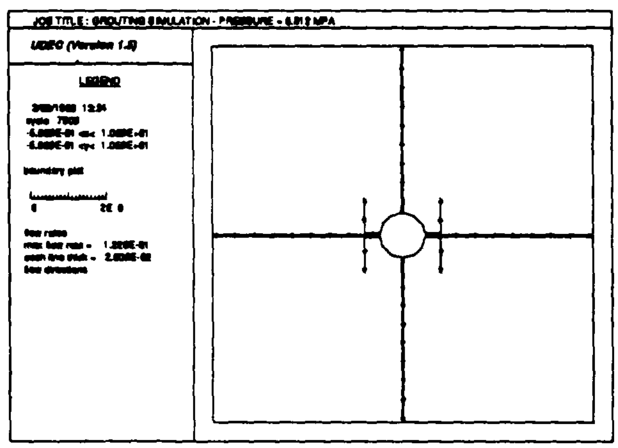
Hassler, L., H. Stille and U. Hakansson. "Simulation of Grouting in Jointed Rock," Proc. 6th Int. Congress on Rock Mechanics (Montreal), pp. 943-946. Rotterdam: A. A. Balkema, 1987.

Littlejohn, G. S. "Design of Cement Based Grouts. Grouting in Geotechnical Engineering," Proceedings, American Society of Civil Engineers, (New Orleans, 1982).

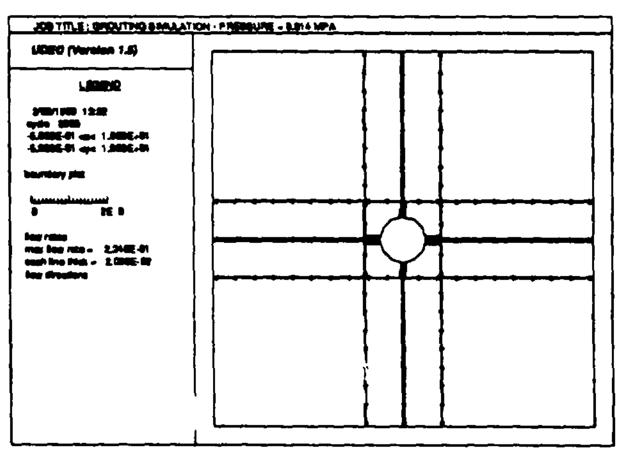
Lombardi, G. "The Role of Cohesion in Cement Grouting of Rock," Quinzième Congres des Grandes Barrage. (Lausanne, 1985).



(a)

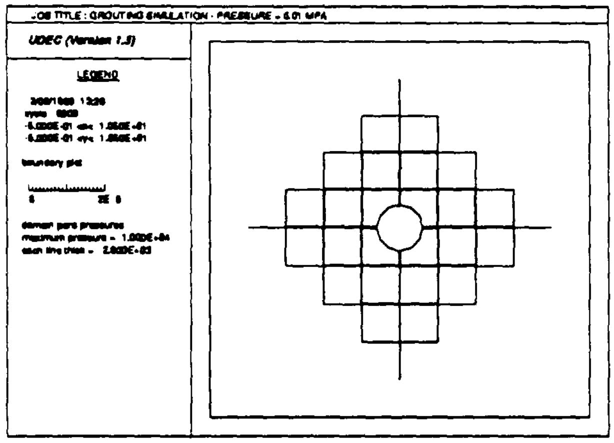


(b)

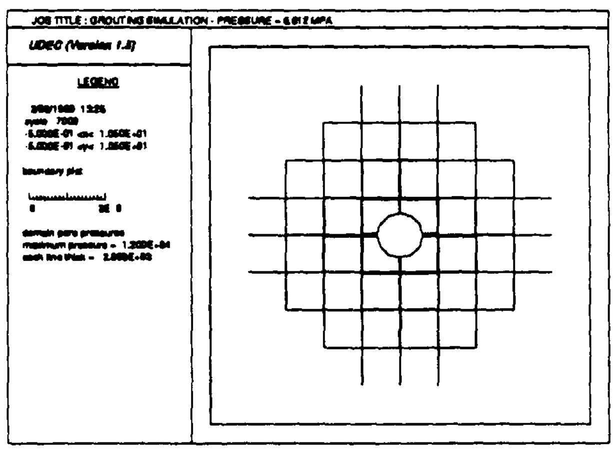


(c)

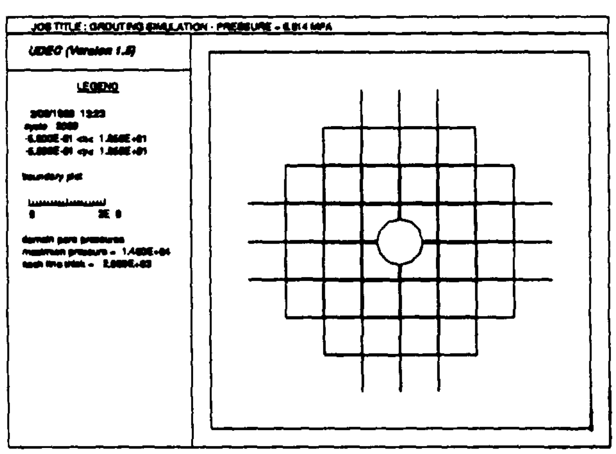
Fig. 2 Flow Rates and Directions for Grouting Simulation: (a) 10,000 Pa, (b) 12,000 Pa, (c) 14,000 Pa



(a)



(b)



(c)

Fig. 3 Grout Pressure in Discontinuities: (a) 10,000 Pa, (b) 12,000 Pa, (c) 14,000 Pa

<u>INPUT COMMANDS FOR VERIFICATION TEST G</u>

```
*
*
* Data file for hypothetical grouting simulation
* Bingham fluid flow logic
*
head
GROUTING SIMULATION
*
* Problem geometry
round 0.01
block 0 0 0 10 10 10 10 0
crack 0 5 10 5
crack 5 0 5 10
tunnel 5 5 0.6 12
jset 0 0 10 0 0 0 2 0
jset 90 0 10 0 0 0 2 0
gen auto 1.25
*
*  Material Properties
*  Intact rock
prop m 1 k 1e10 g 3e9 d 3000
*  Discontinuities
prop m 1 jkn 1e10 jks 1e10 jfric 1 jperm 1e8 azero 0.0001 ares 0.00005
* Insitu stress and boundary condiitons
insitu stres -2e5 0 -1e5 nodis
bound stress -2e5 0 -1e5
* Conditions during Solution
damp auto
mscale on
* create hole
delete 4.5 5.5 4.5 5.5
* cycle to equilibrium
cy 100
save grthola.sav
* use steady state fluid logic
set sflow
* grout properties (density - bulk modulus - cohesion)
fluid den=1000  bul=2e9  coh .1
* pressurize hole (domain 697) in .2e4 increments
pfix dom 697 pres .2e4
cy 1000
save grtholb.sav
pfix dom 697 pres .4e4
cy 1000
```

```
save grtholc.sav
pfix dom 697 pres .6e4
cy 1000
save grthold.sav
pfix dom 697 pres .8e4
cy 1000
save grthole.sav
pfix dom 697 pres 1.0e4
cy 1000
save grtholf.sav
pfix dom 697 pres 1.2e4
cy 1000
save grtholg.sav
pfix dom 697 pres 1.4e4
cy 1000
save grtholh.sav
pfix dom 697 pres 1.6e4
cy 1000
save grtholi.sav
return
stop
```

NRC FORM 335 (2.89) NRCM 1102, 3201, 3202

U.S. NUCLEAR REGULATORY COMMISSION

BIBLIOGRAPHIC DATA SHEET

(See instructions on the reverse)

1. REPORT NUMBER (Assigned by NRC. Add Vol., Supp., Rev., and Addendum Numbers, if any.)

NUREG/CR-5429
Vol. 2

2. TITLE AND SUBTITLE

UDEC (Universal Distinct Element Code)
Version ICG1.5

User's Manual

3. DATE REPORT PUBLISHED

MONTH	YEAR
September	1989

4. FIN OR GRANT NUMBER

FIN D1016

5. AUTHOR(S)

Mark Board

6. TYPE OF REPORT

Formal

7. PERIOD COVERED (Inclusive Dates)

8. PERFORMING ORGANIZATION – NAME AND ADDRESS (If NRC, provide Division, Office or Region, U.S. Nuclear Regulatory Commission, and mailing address; if contractor, provide name and mailing address.)

Itasca Consulting Group, Inc.
1313 5th Street S.E., Suite 210
Minneapolis, MN 55414

9. SPONSORING ORGANIZATION – NAME AND ADDRESS (If NRC, type "Same as above"; if contractor, provide NRC Division, Office or Region, U.S. Nuclear Regulatory Commission, and mailing address.)

Division of High Level Waste Management
Office of Nuclear Material Safety and Safeguards
U.S. Nuclear Regulatory Commission
Washington, DC 20555

10. SUPPLEMENTARY NOTES

11. ABSTRACT (200 words or less)

UDEC is a two-dimensional distinct element code written for analysis of static or dynamic, mechanical, thermomechanical and fracture fluid flow problems in jointed rock or soil. The body to be analyzed is subdivided into a series of blocks which are separated from their neighbors by interface planes which have friction, cohesion and dilation. The blocks themselves may behave as non-linear materials also. The code uses an explicit solution procedure for solving the dynamic equations of motion for the blocks. The large deformation formulation allows interaction between adjacent blocks including slip or separation. General heat transfer logic, fluid flow along the fractures and structural element support are optional features.

12. KEY WORDS/DESCRIPTORS (List words or phrases that will assist researchers in locating the report.)

UDEC, distinct element method, explicit procedure, dynamic, heat transfer, fluid flow, interface planes, large deformation structural elements

13. AVAILABILITY STATEMENT

Unlimited

14. SECURITY CLASSIFICATION

(This Page)

Unclassified

(This Report)

Unclassified

15. NUMBER OF PAGES

16. PRICE

UNITED STATES
NUCLEAR REGULATORY COMMISSION
WASHINGTON, D.C. 20555

SPECIAL FOURTH-CLASS RATE
POSTAGE & FEES PAID
USNRC
PERMIT No. G-67

OFFICIAL BUSINESS
PENALTY FOR PRIVATE USE, $300

D. Chery
4-H-3