

1 COMMAND REFERENCE

This section contains detailed information about all of the commands used by *UDEC*. The commands are described in two major parts.

First, in [Section 1.2](#), there is a summary of the commands, organized into groups of related modeling functions. This summary also provides the recommended command sequence to prepare an input data file.

Second, in [Section 1.3](#), there is an alphabetical listing and a detailed description of all of the commands. Some common input conventions and features are described in [Section 1.1](#).

1.1 Common Conventions and Features

1.1.1 Syntax

UDEC may be operated in “interactive” mode (i.e., commands entered via the keyboard) or “file-driven” mode (i.e., data stored on a data file and read in from the hard disk). In either case, the commands for running a problem are identical, and the particular method of data input depends on user preference.

All input commands are word-oriented and consist of a primary **COMMAND** word followed by one or more **keywords** and numerical input, as required. Some commands (e.g., **PLOT**) accept “switches,” which are keywords that modify the action of the command. Each command has the format

COMMAND **keyword** *value* . . . <**keyword** *value* . . . >

The commands are typed literally on the input line. You will note that only the first few letters are presented in bold type. The program requires only these letters, at a minimum, to be typed for the command to be recognized. Likewise, the keywords, shown in lowercase, are typed literally, and only those letters designated by bold type need to be entered for the keyword to be recognized. However, the entire word for a command or keyword may be entered. By default, the words are not case-sensitive: either uppercase or lowercase letters may be used.

Many of the keywords are followed by a series of numbers (values) that provide the numeric input required by the keyword. Words appearing in ***bold italic*** type stand for numbers. Integers are expected when the word begins with *i*, *j*, *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.

Commands, keywords and numeric values may be separated by any number of spaces, or by any of the following delimiters:

() , =

You will see additional notations with some of the input parameters:

< > denotes optional parameter(s). (The brackets are not to be typed.)

... indicates that an arbitrary number of such parameters may be given.

Anything that follows a semicolon (;) in the input line is taken to be a comment, and is ignored. It is useful to make such comments in the input file when running in batch (i.e., file-driven) mode, since the comments are reproduced in the output. A single input line, including comments, may contain up to 80 characters. An ampersand (&) at the end of a line denotes that the next line is a continuation of keywords or numeric input. The maximum length of a single command, including all continuations, is 5000 characters. A maximum of 1000 input parameters are allowed in one command.

1.1.2 Interactive Input

UDEC supports an 80-character keyboard buffer for the user to type ahead while UDEC is stepping, reading from a data file, or plotting.

There are also several line-editing features that can be used when entering data interactively. These features are summarized in [Table 1.1](#):

Table 1.1 Interactive input editing keys

Key	Effect
any character key	inserts character on input line
<←>	moves cursor left on input line
<→>	moves cursor right on input line
<CTRL ←>	cursor jumps to next input parameter to the left
<CTRL →>	cursor jumps to next input parameter to the right
<BACKSPACE>	deletes character to left of cursor
<DELETE>	deletes character at cursor location
<END>	moves cursor to end of input line
<ESC>	erases line
<F3>	replaces input line with last line in the input buffer
<HOME>	moves cursor to beginning of input line

1.1.3 Commands Accepting the range Phrase

Most *UDEC* commands accept an optional **range** keyword phrase, which *must* be given at the end of the command line if the action of the command is to be limited to a specified range within a model.

The range is a volume of the model (either connected or disconnected) that is defined by the intersection one or more range elements. The range elements are defined by a keyword phrase that follows the **range** keyword – e.g.,

```
print contact disp range 0,1 0,1 angle 0, 30
```

The following keyword phrases are available.

(1) General range

<i>x1 xu y1 yu</i>	lower and upper limits for <i>x</i> and <i>y</i> (if included, must be first after range keyword)
above	<i>x1 y1 x2 y2</i> table <i>n</i> item which lies vertically above the specified line segment or segments in table <i>n</i>
annulus	<i>xc yc r1 r2</i> annular range with center (<i>xc</i> , <i>yc</i>) and radii <i>r1</i> and <i>r2</i>
below	<i>x1 y1 x2 y2</i> table <i>n</i> item which lies vertically below the specified line segment or segments in table <i>n</i>
inside	table <i>n</i> item whose coordinates lie inside of the closed boundary specified by table <i>n</i>
left	<i>x1 y1 x2 y2</i> table <i>n</i> item which lies horizontally to the left of the specified line segment or segments in table <i>n</i>
outside	table <i>n</i> item whose coordinates lie outside of the closed boundary specified by table <i>n</i>

region *x1, y1 x2, y2 x3, y3 x4, y4*

arbitrary four-cornered area

right *x1 y1 x2 y2*
table *n*

item which lies horizontally to the right of the specified line segment or segments in table *n*

window plotting window

xrange *x1 x2*

lower and upper limits for *x*

yrange *y1 y2*

lower and upper limits for *y*

(2) Domain range

atdomain *x, y*

only for domain with center nearest *x, y*

domain *n1 <n2 ... >*

only domains with specific numbers (max = 10)

(3) Joint set range

jregion *n*

joint set defined with the **JREGION id = *n*** command

(4) Block range (also applies to corners, gridpoints and zones)

area *a*

only blocks with area < *a*

atblock *x1 y1*

block which contains coordinates *x1 y1*

block *n1 <n2 ... >*

only blocks specified in list (max = 10)

group name
 only blocks in group

material *n1 <n2 ... >*
 only blocks with specified material numbers (max = 10)

(5) Contact range

angle *a1 a2*
 only contacts on joints with angle between *a1* and *a2*

block *n1 <n2 ... >*
 only contacts between blocks on the list (max = 10)

contact *n1 <n2 ... >*
 only contacts specified (max = 10)

ginterface name 1, name 2
 only contacts between blocks that are members of named groups

group name
 only contacts (joints) in group name

hinterface only contacts between hidden and visible blocks

id *n1 <n2 ... >*
 only contacts with ID on the list (max =10)

interface *n1 n2*
 only contacts between blocks *n1* and *n2*

jmaterial *n1 <n2 ... >*
 only contacts with specific material numbers (max = 10)

minterface *n1 n2*
 only contacts between blocks of material *n1* and *n2*

(6) Zone range

atzone	<i>x, y</i>
	only the zone containing the point <i>x, y</i>
group	name
	only zones in group

(7) Gridpoint range

atgridpoint	<i>x, y</i>
	only the gridpoint nearest the point <i>x, y</i>

1.1.4 Online Help

The **help** keyword is available to list all of the keywords that apply for each *UDEC* command.* For example, to find all of the keywords that apply to the **PRINT** command, type

```
print help
```

The **?** character may be used in place of the **help** keyword.

* Note that some keywords listed by **help** are not described in the manual. These refer to features in *UDEC* that are not available for general use.

1.2 Commands by Function

The following is a recommended sequence for command input, according to function. In general, commands may be given in any logical order; however, certain commands must precede others. These are identified in this section. Only the primary command words and most frequently used keywords are presented; a detailed description of all keywords is given in [Section 1.3](#).

1.2.1 Specify Program Control

Certain commands allow the user to start new analyses without leaving *UDEC*, or restart previous model simulations and continue from the last analysis stage. The following commands provide program control.

CALL	reads into <i>UDEC</i> a user-prepared remote input command file.
CONTINUE	continues reading a paused data file.
NEW	starts a new problem without exiting <i>UDEC</i> .
PAUSE	pauses reading a data file.
QUIT	stops execution of <i>UDEC</i> (synonym: STOP).
RESTORE	restores an existing saved state from a previously executed problem.
RETURN	returns program control from the remote input mode to the local, interactive mode.
SAVE	saves the current state of the analysis in a file.
STOP	stops execution of <i>UDEC</i> (synonym: QUIT).

It is best to give **SAVEd** files a different extension (e.g., “.SAV”) than input files (e.g., “.DAT”), to avoid confusion when a saved state is **RESTORED** or an input file is **CALLED**.

1.2.2 Specify Special Calculation Modes

UDEC performs mechanical (plane-strain) calculations as the standard mode. Optional calculation modes are also available and are specified with the **CONFIG** command. **CONFIG** *must* be given before the **BLOCK** command if any of the following options are desired. The keywords are:

axi	axisymmetric mode
cell	alternate contact detection logic
cppudm	user-defined and extended constitutive models
creep	enable creep models to be used

fluid	allocates additional memory structures used for fluid-flow logic in <i>UDEC</i> .
p_stress	plane stress (only valid for models: elastic, anisotropic, strain softening (ss) and all DLL models)
thermal	thermal analysis

1.2.3 Input Problem Geometry

There is a general procedure to establish the problem geometry:

- (1) specify a constant distance for corner rounding using the **ROUND** command (default = 0.5);
- (2) specify a minimum value of block edge length using the **SET edge** command (default is twice the corner rounding length);
- (3) create a single rigid block defining the original boundary of the modeled region using the **BLOCK** command; and
- (4) divide this original block into smaller blocks to create the desired assemblage of blocks.

All possible future blocks must be created during this step because blocks can no longer be created once execution begins. This includes, for example, the creation of shapes for regions that will be excavated at a later stage in the analysis. 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** or **STEP** command, or upon internally discretizing any blocks to form deformable blocks, *UDEC* will delete any joints that do not completely intersect a block. Be careful when entering cracks because there is no facility for deleting a crack once it has been formed. In addition, once rigid blocks are changed to deformable blocks, they can no longer be cracked or split.

These commands are used to establish geometries of block assemblage:

ARC	creates a series of cracks in an arc pattern.
CRACK	creates a single crack in a block.
DELETE	deletes block(s).
EDGE	sets minimum block edge length.
GROUP	Blocks, zones or contacts may be assigned named groups.
HIDE	hides block from view, and from the action of model-building commands.

JDELETE	deletes joints that do not completely intersect a block.
JOIN_BLOCK	joins contacts based on block contact ranges.
JOIN_CONTACT	joins contacts based on contact location ranges.
JREGION	defines a region for automatic joint-set generation.
JSET	generates a joint set from statistically based parameters.
LINK	connects a loose block to main block system.
SHOW	makes hidden blocks visible.
SPLIT	splits block along line.
TUNNEL	generates a tunnel geometry by introducing a circular pattern of cracks.
VORONOI	generates Voronoi tessellation.

1.2.4 Specify Block Deformability

All blocks are rigid, by default. Blocks can be made deformable by using the **GENERATE** command. The **GENERATE** command discretizes specified blocks into constant-strain finite-difference triangular zones.

Constitutive models for deformable blocks are assigned in one of two ways:

1. The null, elastic, Mohr-Coulomb and Drucker-Prager models are assigned to a single block or regions of blocks with the **CHANGE cons** command. Models assigned using the **CHANGE** command are referred to as *block models*. Block model properties are assigned to property numbers in the **PROPERTY mat** command. The property numbers are assigned to blocks with the **CHANGE mat** command.
2. The null, elastic, Mohr-Coulomb, strain-softening, ubiquitous-joint and double-yield models are specified for a range of *zones* within the blocks. The models are accessed via the **ZONE** command. The implementation of this command requires additional memory to store the extra model variables. Models assigned using the **ZONE** command are referred to as *zone models*. Zone model properties are assigned with the **ZONE** command.

For thermal analysis, thermal properties are assigned to blocks with the **PROPERTY** command.

1.2.5 Assign Joint Constitutive Models and Properties

Constitutive models and properties are assigned to contacts in one of two ways:

1. Joint constitutive models are assigned to contacts with the **CHANGE jcons** command. Joint model properties are assigned to property numbers with the **PROPERTY jmat** command. The property numbers are assigned to contacts with the **CHANGE jmat** command.
2. Joint constitutive models are specified for a range of contacts via the **JOINT** command. Joint properties are assigned directly to contacts using the **JOINT** command. Use the **JMODEL** command for user-defined joint models.

For fluid flow analysis, properties for fluid in the joints are assigned with the **FLUID** command.

1.2.6 Assign Initial Conditions

Initial problem and model-solution conditions are assigned with the following commands.

GRAVITY	specifies gravity.						
INITEM	sets initial temperatures at block corners and gridpoints.						
INITIAL	sets initial loads, velocities and displacements.						
INSITU	sets initial stresses in all deformable blocks and joints and pressures in domains.						
RESET	resets selected variables to zero.						
SET	allows the user to initialize both problem and model conditions by selecting one or more keywords, such as						
	<table> <tr> <td>flow</td><td>turns on/off fluid flow modes (compressible or incompressible transient flow, or steady-state flow).</td></tr> <tr> <td>gravity</td><td>specifies gravity.</td></tr> <tr> <td>mech</td><td>turns on/off the mechanical solution scheme.</td></tr> </table>	flow	turns on/off fluid flow modes (compressible or incompressible transient flow, or steady-state flow).	gravity	specifies gravity.	mech	turns on/off the mechanical solution scheme.
flow	turns on/off fluid flow modes (compressible or incompressible transient flow, or steady-state flow).						
gravity	specifies gravity.						
mech	turns on/off the mechanical solution scheme.						
TADD	assigns initial temperatures in an angular region. Temperature values are input via the TABLE command.						
WELL	prescribes a fluid source in a domain.						

1.2.7 *Apply Boundary Conditions*

Model boundary conditions are prescribed with the following commands.

BE	defines conditions for the boundary-element model.
BOUNDARY	applies mechanical and hydraulic boundary conditions.
FBOUNDARY	applies porous medium grid to the outer boundary.
FFIELD	prescribes dynamic free field.
FIX/FREE	allows velocity at block centroids to be fixed (i.e., prevented from changing) or freed (i.e., allowed to change) for selected rigid blocks.
LOAD	prescribes loads at block centroids of rigid blocks.
PFIX/PFREE	allows pressure to be fixed or freed in selected domains.
TFIX/TFREE	allows temperature to be fixed or freed at selected locations.
THAPP	applies thermal boundary conditions.

1.2.8 *Specify Structural Support*

Four types of artificial passive support may be specified:

CABLE	generates global reinforcing elements across joints and within fully deformable blocks for modeling fully bonded passive reinforcement that explicitly considers the shear behavior of the grout annulus.
REINFORCE	generates local reinforcement elements across joints for modeling fully bonded passive reinforcement in hard rocks.
STRUCT	generates structural (i.e., beam) elements (e.g., surface support such as for modeling tunnel lining).
SUPPORT	specifies a one-dimensional nonlinear spring support element.

1.2.9 *Specify User-Defined Variables or Functions*

The embedded programming language in *UDEC*, *FISH*, may be invoked to define special variables or functions that a user would like for a specific problem. *FISH* statements (described in [Section 1](#) in the *FISH* volume) are any statements given between the *UDEC* commands **DEFINE** and **END**.

Variations in model conditions can also be made using the **TABLE** command.

1.2.10 Specify Solution Conditions

The following commands are used to specify numerical conditions to achieve a solution.

DAMPING	material damping (used to absorb kinetic energy)
FRACTION	fraction of critical timestep used in solution
MSCALE	sets zone and block masses for mass-scaling.
SET	sets certain internal variables, such as time interval, to update contacts.

1.2.11 Monitor Model Conditions during the Solution Process

Commands are available to monitor the change in model variables as the solution process progresses. This is helpful both to ascertain when an equilibrium or failure state has been reached, and to monitor the change in variables during transient calculations, such as dynamic wave propagation or groundwater flow.

HISTORY	causes a record to be made of the changes in a variable as timestepping proceeds. The resulting plots help the user identify when a steady-state condition is reached.
----------------	--

1.2.12 Solve the Problem

Once the appropriate problem conditions are defined in the *UDEC* model, the problem is solved by taking a series of calculation steps. The following commands permit either automatic solution of the *UDEC* model or user control of the solution process.

CYCLE <i>n</i>	executes <i>n</i> timesteps.
RUN	executes thermal timesteps for a specified thermal time. Thermal problems are solved with this command.
SOLVE	enables the automatic detection of a mechanical equilibrium condition. The calculation is performed until a preset limiting condition is reached. Limiting conditions can be modified by keywords through the SOLVE command.
STEP <i>n</i>	executes <i>n</i> timesteps.

For both **SOLVE** and **STEP** (or **CYCLE**), the maximum out-of-balance force for the model is continually printed to the screen. The user may interrupt the calculational stepping at any time by pressing the <Esc> key. *UDEC* will return full control to the user after the current step is complete; the user may then check the solution and save the state, or continue with the analysis.

1.2.13 *Generate Model Output*

Several commands are available to allow the user to examine the current problem state:

COPY	copies a plot to a hardcopy device.
DUMP	prints out variables in the main array of <i>UDEC</i> .
HEADING	assigns a plot title. (synonym: TITLE)
LABEL	provides alternate labels for histories and tables. The LABEL command also allows the user to add labels to plots.
MOVIE	controls the capture of screen plots and permits rapid replay as a “movie.”
PLOT	requests a plot, on the screen, of various problem variables, including the HISTORY of a variable.
PRINT	prints output for problem conditions and main variables of the model.
SET	log allows the user to save a record of interactive sessions to a file. The SET command also provides several controls over different plotting conditions.
STRESS	prints principal stresses on a grid over the model.
TITLE	assigns a plot title. (synonym: HEADING)
WINDOW	changes the viewing window for a plot.

1.2.14 *Access the DOS System*

The *UDEC* command **SYSTEM** is available to allow the user to access DOS commands (e.g., COPY, DEL, DIR, REN and TYPE) without leaving *UDEC*. The user may also start a DOS session by entering the **SYSTEM** command; return to *UDEC* by typing the DOS command EXIT.

1.3 *UDEC Commands – Detailed Listing*

ARC

ARC *xc yc xb yb theta ncracks*

This command creates a pattern of *ncracks* 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**BE**

keyword <keyword>

Boundary-element (limit 300) conditions are applied to the boundary domain generated by the keyword **gen**. The keywords and associated parameters are:

ff_bulkmod bulk modulus for outside rock mass*

ff_density density for outside rock mass*

ff_nu Poisson's ratio for outside rock mass*

ff_shearmod shear modulus for outside rock mass*

ff_ymod Young's modulus for outside rock mass*

fix *xh yh xv yv*

fixes a point (*xh, yh*) in the *x*-direction and a point (*xv, yv*) in the *y*-direction outside the model region (must precede the **stiff** keyword).

gen <*xl xu yl yu*>
<**corner** *ic1 ic2*>

generates a boundary-element domain along the outer boundary of a region described by *xl* to *xu* and *yl* to *yu*, or block corners *ic1* and *ic2*. (The affected boundary runs clockwise from *ic1* to *ic2*.)

half half-plane solution for boundary-element domain, assuming surface at *y* = 0 (must precede the **stiff** keyword)

mat *n**

material number *n* assigned to far-field properties

off turns off boundary elements.

stiff automatically generates stiffness matrix.

It is recommended that the user apply the **BE** commands after the model comes to initial (i.e., consolidation) equilibrium under in-situ loading conditions.** (Use the **BOUNDARY stress** command, for example, to apply far-field stresses initially.) The recommended order of the **BE** commands is:

* Properties for boundary element domain (outside rock mass) may be specified by material number, bulk and shear modulus, or Young's modulus and Poisson's ratio.

** Warning: All model stresses *must* be specified in units of MPa for the boundary-element boundary. Boundary-elements can only be applied for the plane-strain condition (see [Section 3.4.4.3](#) in the **User's Guide**).

BE

BE gen**BE half** (if half-plane solution is used)**BE mat****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

BLOCK <keyword *v*> <*x1 y1 x2 y2* ... >

creates a rigid block with corner coordinates (*x1,y1*), (*x2, y2*), etc., given in a clockwise direction.

The following optional keywords may also be given.

angle *ang*

Corners are added to the block on straight edges so that any angle with its sides on two consecutive corners and its apex at the block's center never exceeds *ang*.

NOTE: This keyword should be used to control the boundary mesh refinement whenever an **FBOUNDARY** command is to be invoked.

circular *xc yc r n*

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.

constitutive *n*

Constitutive number *n* is assigned to the block. The default is *n* = 1. (See the **CHANGE** command.)

fill *x y <mat n> <cons n>*

may be used as an alternative to the null models. The problem with null models is that it is difficult to maintain the shape of the zones in the excavated area if large displacements occur. The fill step may result in a bad zone geometry, and cycling will not be possible. The alternative is to simply delete the excavated blocks and then use the **BLOCK fill** command to refill the void. The new block may be cut with the **CRACK**, **SPLIT** or **JSET** command (use **SET cutting on** first). The new block may also be zoned with the **GENERATE** command. This method of excavating and filling will also allow trimming of the top of the fill block so that the fill does not fit tight up against the top of the excavation. This command only works for fully enclosed excavations, and will not work to fill excavations that intersect the outer boundary of the model.

cons n is the constitutive model to be used for the fill material.

mat n is the material number to be assigned to the fill material.

x y refers to a point inside the void to be filled.

BLOCK **material****material** *n*

Material number *n* is assigned to the block. The default is *n* = 1. (See the **PROPERTY** command.)

test <*v*>

shortcut command to create a square block of width *v*. If *v* is omitted, the block will have a width or height of 10. The lower-left corner of the block is always (0, 0). The upper-right corner of the block is (*v*, *v*). The default is (10, 10).

Normally, one **BLOCK** command may be used per run. Additional blocks may be created with a **CRACK**, **TUNNEL**, **ARC**, **JSET** or **VORONOI** command, and unwanted blocks deleted with the **DELETE** command. Any block may be changed to deformable using the **GENERATE** command.

When **CONFIG cell** has been specified, multiple **BLOCK** commands may be used. Also, the **BLOCK fill** command may be used at any time.

BOUNDARY

BOUNDARY keyword <keyword> ... <**range**...>
 <**corner** *ic1 ic2*>

Boundary conditions are specified for an external boundary over a specified range (see [Section 1.1.3](#)) 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 forces and stresses can be applied to both rigid and deformable blocks. Boundary velocities can only be applied to deformable blocks. Boundary conditions are defined by a given boundary condition keyword. The keywords are divided into ten boundary types. The keywords and associated parameters are as follows.

1. Load Boundary

xload *fx*
 x-direction load to apply to each gridpoint (see note)

yload *fy*
 y-direction load to apply to each gridpoint (see note)

2. Free Boundary

xfree removes boundary condition and all loads in the *x*-direction.

yfree removes boundary condition and all loads in the *y*-direction.

3. Stress Boundary

stress *sxxo sxyo syyo*
 boundary stress parameters: *xx*-stress, *xy*-stress and *yy*-stress (see note)

xgrad *sxxx sxyx syyx*

BOUNDARY ygrad**ygrad****sxx** **sxy** **syy**

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

$$sxx = sxxo + (sxxx \cdot x) + (sxxxy \cdot y)$$

$$sxy = sxyo + (sxyx \cdot x) + (sxyy \cdot y)$$

$$syy = syyo + (syyx \cdot x) + (syyy \cdot y)$$

xgrad and **ygrad** must be given on the same input line as the **stress** keyword (use & for continuation if input is too long).

NOTE: All loads and stresses are assumed to be constant and permanent by default,* and are *added* to the existing permanent loads.

CAUTION: The stress boundary affects all degrees of freedom. For example, if **BOUNDARY stress** follows a **BOUNDARY xvel** or **yvel** command affecting the same gridpoint(s), then the effect of the previously prescribed **xvel** or **yvel** will be lost.

4. Velocity (Displacement) Boundary**gvel****vxo**, **vyo** **vxx**, **vxy** **vyx**, **vyy**

apply velocity gradient to boundary

$$vx = vxo + (vxx \cdot x) + (vxy \cdot y)$$

$$vy = vyo + (vyx \cdot x) + (vyy \cdot y)$$

nvel**vn**

normal direction velocity for deformable blocks

svel**vs**

shear direction velocity for deformable blocks

xvel**vx**

x-direction velocity for deformable blocks

yvel**vy**

y-direction velocity for deformable blocks

* Transient (time-varying) loading is applied if a history keyword phrase (**history**) is given on the *same* input line as the load or stress (see load and velocity histories, below).

5. Hydraulic Boundary

impermeable	impermeable boundary for wetting fluid
nwimpermeable	impermeable boundary for non-wetting fluid
nwpermeable	defines a boundary as permeable to non-wetting fluid for joints intersecting the boundary.
nwppressure	<i>fpnw</i> non-wetting fluid pressure at boundary (the default is <i>fp</i> = 0.0)
nwpxgrad	<i>fpnwx</i> non-wetting fluid pressure gradient along <i>x</i> -axis
nwpygrad	<i>fpnwy</i> non-wetting fluid pressure gradient along <i>y</i> -axis $pnw = fpnw + (fpnwx \cdot x) + (fpnwy \cdot y)$
permeable	defines a boundary as permeable for joints intersecting the boundary.
pp	<i>fpw</i> wetting fluid pressure at boundary; must still use the xload , yload or stress keyword to set mechanical boundary conditions. (The default is <i>fp</i> = 0.0.)
pxgrad	<i>fpwx</i> wetting fluid pressure gradient along <i>x</i> -axis; must still use the xload , yload or stress keyword to set mechanical boundary conditions
pygrad	<i>fpwy</i> wetting fluid pressure gradient along <i>y</i> -axis; must still use the xload , yload or stress keyword to set mechanical boundary conditions $pw = fpw + (fpwx \cdot x) + (fpwy \cdot y)$
saturation	<i>fs</i> wetting fluid saturation on the boundary
satxgrad	<i>fsx</i> saturation gradient along <i>x</i> -axis

BOUNDARY satygrad

satygrad	<i>fsy</i> saturation gradient along y-axis $s = fs + (fsx \cdot x) + (fsy \cdot y)$
seepage	seepage boundary condition. If a seepage boundary condition is assigned to a boundary, then (1) wetting and non-wetting fluid pressures are fixed (wetting fluid pressure is equal to non-wetting fluid pressure); and (2) the unsaturated boundary is impermeable for wetting fluid.

6. Nonreflecting (Viscous) Boundary

Nonreflecting boundaries are available for dynamic analyses:

ff_bulkmod	far-field bulk modulus*
ff_density	far-field density*
ff_nu	far-field Poisson's ratio*
ff_shearmod	far-field shear modulus*
ff_ymod	far-field Young's modulus*
mat	<i>n</i> * material number <i>n</i> assigned to far-field properties (required for non-reflecting boundaries)
xvisc	nonreflecting boundary in <i>x</i> -direction
yvisc	nonreflecting boundary in <i>y</i> -direction

* Far-field elastic properties can be specified in three ways: by material number (in which case the properties are specified using the **PROPERTY** command); by density, bulk and shear modulus; and by density, Young's modulus and Poisson's ratio.

7. Free-Field Boundary

ffield A free-field boundary condition is automatically applied to the lateral outer boundaries of the model. The boundaries must be vertical and straight. The **BOUNDARY ffield** command must be given before changing other boundary conditions for the dynamic stage of an analysis. The properties and constitutive models of the zones adjacent to the free-field boundary are copied and used by the dynamic free-field. The model should be in static equilibrium prior to using the command. The free-field itself is automatically cycled to equilibrium when the command is given. The viscous boundary conditions along the sides of the model are applied automatically. The free-field performs a small-strain calculation even though the adjacent zones may be in large-strain. The **FFIELD** command may be used instead of, or in addition to, the **BOUNDARY ffield** command to manually control the free-field, but it is usually not needed. See [Section 4.3.1.4](#) in **Special Features** for more details. **BOUNDARY ffield** cannot be used with **CONFIG cell**.

8. Boundary Multipliers

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

nmul	<i>nm</i>
	<i>n</i> -direction load multiplier (default = 1.0)
smul	<i>sm</i>
	<i>s</i> -direction load multiplier (default = 1.0)
xmul	<i>xm</i>
	<i>x</i> -direction load multiplier (default = 1.0)
ymul	<i>ym</i>
	<i>y</i> -direction load multiplier (default = 1.0)

9. Histories for Loads, Velocities and Hydraulic Pressures

The keyword **history** defines a time history that is a *multiplier* for all the load, stress, *x*- and *y*-velocities, and hydraulic pressures (e.g., **pp**, **pygrad** and **pxgrad**) given on the same input line.

NOTE: Loads assigned a history are considered *transient* loads, and are stored separately from any permanent loads assigned to the same corners. If a corner already has a transient load, any new load with the *same* history type (see below) is *added* to it; however, a new transient load with a *different* history type *replaces* the old transient load.

Boundary loads, stresses or velocities can be applied as a constant, sinusoidal or user-defined *FISH* function, or they may consist of a series of points given by a **TABLE** command or read from a file. The keywords *must* be given on the same line specifying the load, stress or velocity.

history	keyword	
	cosine	<i>freq time</i> cosine wave with frequency <i>freq</i> (cycles/sec) applied for a problem time period of <i>time</i>
	n	History time record <i>n</i> (previously read into <i>UDEC</i> with the hread keyword) is applied.
	name	The history multiplier is a user-defined <i>FISH</i> function, where name is the function name.
	sine	<i>freq time</i> sine wave with frequency <i>freq</i> (cycles/sec) applied for a problem time period of <i>time</i>
	table	<i>n</i> history is input using the TABLE command. TABLE n consists of a list of pairs: <i>time, f(time)</i> . Linear interpolation is performed between the given discrete points.

Histories can be assigned in specific directions by replacing the **history** keyword with **nhist**, **shist**, **xhist** and **yhist** for histories in the normal, shear, *x*- or *y*-directions, respectively.

10. Input History Storage

The keyword and associated parameters for storing the input history are

hread *n* filename

reads time record *n* from file filename.

The file should be formatted and be organized in the following form.

Line 1 heading of up to 80 characters

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

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

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.

BOUNDARY**BOUNDARY** **interior** keyword. . . <**range**. . . >

Loads and velocities are applied to block edges of an interior boundary (i.e., not lying on the outer boundary) within a specified range (see [Section 1.1.3](#)). For example, the range keyword **domain *n*** restricts the range to block edges on the boundary of interior domain ***n***.

1. Load Boundary

xload *fx*
 x-direction load

yload *fy*
 y-direction load

2. Free Boundary

xfree removes boundary condition and all loads in *x*-direction.

yfree removes boundary condition and all loads in *y*-direction.

3. Stress Boundary

stress *sxxo sxyo syyo*
 boundary stress parameters: *xx*-stress, *xy*-stress and *yy*-stress

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

$$sxx = sxxo + (sxxx \cdot x) + (sxxy \cdot y)$$

$$sxy = sxyo + (sxyx \cdot x) + (sxyy \cdot y)$$

$$syy = syyo + (syyx \cdot x) + (syyy \cdot y)$$

The same limitations for applying loads and stresses for the exterior boundary also apply for interior boundaries. The optional **history** keyword (followed by the history type parameters listed for the **BOUNDARY** command) defines the loads as transient. If no history is given, loads are assumed to be constant and permanent, and are *added* to existing permanent loads. New transient loads are only added to existing transient loads if the history type (see **BOUNDARY** histories) is the same. If the new load has a different history type than the existing load, the old load is removed and *replaced*

by the new load (i.e., for each corner, only a single transient load may exist in each of the coordinate directions).

4. Velocity (Displacement) Boundary

nvel	vn	normal direction velocity for deformable blocks
svel	vs	shear direction velocity for deformable blocks
xvel	vx	x-direction velocity for deformable blocks
yvel	vy	y-direction velocity for deformable blocks

5. Histories for Loads and Velocities

The keyword **history** defines a time history that is a *multiplier* for all the load, stress, and *x*- and *y*-velocities given on the same input line.

NOTE: Loads assigned a history are considered *transient* loads, and are stored separately from any permanent loads assigned to the same corners. If a corner already has a transient load, any new load with the *same* history type (see below) is *added* to it; however, a new transient load with a *different* history type *replaces* the old transient load.

Interior boundary loads, stresses or velocities can be applied as a constant, sinusoidal or user-defined *FISH* function, or they may consist of a series of points given by a **TABLE** command or read from a file. The keywords *must* be given on the same line specifying the load, stress or velocity.

history	keyword	
	cosine	<i>freq time</i> cosine wave with frequency <i>freq</i> (cycles/sec) applied for a problem time period of <i>time</i>
	n	History time record <i>n</i> (previously read into <i>UDEC</i> with the hread keyword) is applied.
	name	The history multiplier is a user-defined <i>FISH</i> function, where <i>name</i> is the function name.

sine	<i>freq time</i>
	sine wave with frequency <i>freq</i> (cycles/sec) applied for a problem time period of <i>time</i>
table	<i>n</i>
	history is input using the TABLE command. TABLE <i>n</i> consists of a list of pairs: <i>time, f(time)</i> . Linear interpolation is performed between the given discrete points.

Histories can be assigned in specific directions by replacing the **history** keyword with **nhist**, **shist**, **xhist** and **yhist** for histories in the normal, shear, *x*- or *y*-directions, respectively.

6. Input History Storage

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

hread *n* filename
reads time record *n* from file filename.

The file should be formatted and be organized in the form

Line 1 heading of up to 80 characters

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

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

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.

7. Delete

Interior boundaries should be deleted prior to deleting blocks that are part of the boundary.

CABLE

CABLE *x1 y1 x2 y2 npoint mats matg* <preten> <keyword>
delete <range>

Execution of this command creates reinforcing elements that explicitly model the shear behavior of a 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 deformable blocks, and the **CABLE** command must be invoked *after* the **GENERATE** command. Reinforcement properties and grout properties are specified and stored using the **PROPERTY** command. The following parameters are also required.

matg material property number for grout. (The **PROPERTY** command should be used to specify **cb_kbond** and **cb_sbond** for grout.)

NOTE: The units for **cb_kbond** and **cb_sbond** are [force / cable length / displacement] and [force / cable length], respectively.

mats material property number for reinforcing (i.e., steel). (The **PROPERTY** command should be used to specify **cb_ymod**, **cb_yield**, **cb_ycomp**, **cb_fstrain** and **cb_density** for the material.)

npoint number of lumped mass nodal points (*npoint* ≥ 2)

Pretensioning of the reinforcing can be specified by providing a value (in units of force) for the optional parameter *preten*. Always use the **PRINT property cable** command to check property assignment.

The following keywords may be used.

connect The optional **connect** keyword will shift the cable node end closest to a structural (beam) element node to coincide with the beam node. The structural element nodes must be created before the cable nodes. The connection is not allowed to fail. The common, or shared, node will not appear in the list of cable nodes, but it does appear in the list of structural element nodes. The cable element list will show the common node ID number at one end of the cable element, and the normal cable node at the other. Beam and cable nodes are entirely separate entities. This is because the beam nodes are in contact with the surface of a block, whereas the cable nodes are connected to the model in the interior of a zone. All common nodes are therefore structural element nodes, rather than cable nodes. As a consequence, it is possible for the cable element information display to indicate identical node IDs at either end when the **connect** keyword is used.

CABLE**delete**

delete The optional **delete** keyword will remove all cable elements from the specified range. If no range is given, all cable elements will be deleted.

extend The **extend** keyword causes the end of the current cable to connect to the end node of an existing cable element.

first_node *n*
 sets material number for first node of cable element.*

last_node *n*
 sets material number for last node of cable element.*

The cable reinforcing model is described in [Section 1](#) in **Special Features**.

The material properties for segments of an existing cable element can be changed with the **CHANGE cable** command.

* Neither **first_node** nor **last_node** can be used to modify the properties of a connection between the cable element and a structural element created using the **connect** keyword.

CALL**CALL** filename

A remote input file, filename, can be invoked with the **CALL** command. Any series of input instructions can be placed in this file so that *UDEC* will run unattended. Additional files may, in turn, be **CALL**ed from a command file; there is no limit to the level of call nesting. However, make sure to prevent files from calling each other recursively (e.g., file “ABC” calls file “DEF” which then calls “ABC”), because the results will not be correct and the computer disk may become full. If no file name is given, a file open dialog will open.

The **RETURN** command causes control to be returned to the user (if a single level of calling was used) or to the calling file at the next line after the **CALL** command. The <Esc> key terminates input from command files and returns control to the user, no matter to what level calls are nested.

If plots are made to the screen while a **CALL** file is in control, *UDEC* will not pause for the user to hit <ENTER>, as in interactive mode. However, the user can pause the plot by using the keyword **hold** in the **PLOT** command. Screen plots will be saved to the movie file if **MOVIE** mode is set **on**.

The **PAUSE** command causes the processing of the data file to stop. The user may enter any *UDEC* command, other than **NEW** or **CALL**, and then type **CONTINUE** to continue reading the data file.

A **CYCLE** command that was initiated in a data file may be interrupted by pressing the <Esc> key. The command **CYCLE continue** will cause *UDEC* to complete the interrupted **CYCLE** command and continue processing the data file. The **CONTINUE** command will cause *UDEC* to continue processing the data file, but *UDEC* will not complete the interrupted **CYCLE** command.

CAVE

CAVE <keyword>

This command allows the user to simulate some of the effects of rock fracturing that occur in a block-cave mining operation. The area of cave is defined as a Mohr-Coulomb material. The stresses in a block are reset to zero whenever the user-specified percentage of zones has yielded. New material property numbers and constitutive laws may also be assigned. The following keywords may be used.

newconst *n*

new constitutive model number *n* to be assigned. If this value is not set, the constitutive model will remain unchanged.

newmaterial *n*

new material number *n* to be assigned. If this value is not set, the material number will remain unchanged.

percent *value*

percentage of zones in a block that must fail in order to trigger the changes.

zero <off>

zero the stresses when **percent** is exceeded. If this switch is not used, stresses will not be changed.

CHANGE

CHANGE keyword <keyword> <**range**. . . >

Block, joint and cable element material characteristics are prescribed and changed with the **CHANGE** command. All blocks with centroids lying within the optional range (see [Section 1.1.3](#)) have block material characteristics changed. Likewise, all joints with contact coordinates lying within the optional range, or cables with nodes lying within the optional range, have material characteristics changed. If no range is specified, all blocks and joints will have characteristics changed according to the keywords given below. Extended zone models and user-defined models are assigned using the extended **ZONE** command. User-defined joint models are defined using the **JMODEL** command. Local storage joint models are assigned using the **JOINT** command.

The following keywords are used to change characteristics.

1. Block Characteristics

cons *n*

Constitutive number *n* is assigned to designated deformable blocks (see [Table 1.2](#)).

mat *n*

Material property number *n* is assigned to designated rigid or deformable blocks. (All blocks initially default to **mat** = 1. The maximum value for *n* is 50.)

Table 1.2 *Constitutive models for deformable blocks*

cons	Model Description
0	null material (The null model is used to model excavated material. The stresses within the null block are automatically set to zero.)
1	linearly elastic, isotropic (default)
3	elastic/plastic, Mohr-Coulomb failure (This model should be used with caution since accurate solution to plasticity problems requires that the triangular zoning have a gridpoint at the centroid of each block. GENERATE quad zoning should be used whenever possible to improve plasticity analyses. However, no significant errors have been noted in problems for which the above criterion has not been met.)
6	elastic/plastic, Drucker-Prager failure (The same caution discussed above for the Mohr-Coulomb model (cons = 3) also applies for the Drucker-Prager model.)

NOTES:

1. Constitutive models may also be assigned to regions of zones within blocks with the **ZONE** command.
2. All block constitutive models are described in [Section 1](#) in **Constitutive Models**.

2. Joint Characteristics

jcons	<i>n</i>	Constitutive number <i>n</i> is assigned to designated contacts (see Table 1.3).
jmat	<i>n</i>	Material property number <i>n</i> is assigned to designated contacts. (All contacts initially default to jmat = 1. The maximum value for <i>n</i> is 50.)

Table 1.3 *Joint constitutive models*

jcons	Model Description
1	point contact elastic/plastic with Coulomb slip failure (units are [force/displacement] for contact stiffnesses, and [force] for cohesion and tension)
2	joint area* contact elastic/plastic with Coulomb slip failure (units are [stress] for cohesion and tension, and [stress/displacement] for joint stiffnesses) (default)
3	continuously yielding joint model (see Section 2 in Constitutive Models for a detailed explanation)
5	same as jcons = 2, except that the internal fracture flag is set for each joint segment when joint shear or tensile strength is exceeded. If the fracture flag is set, residual values for friction, cohesion and tension are used in all subsequent calculations.
7	optional Barton-Bandis (BB) joint model. See Section 3 in Constitutive Models for details.
10	joined contacts. Joined contacts cannot slip or open.

* NOTE: The minimum joint area is limited to twice the rounding length, so that it is not necessary to specify point contact properties if both point contacts and area contacts occur between blocks.

3. Cable Characteristics

cable *matg* <*mats*>

Material property number *matg* is assigned to designated cable nodes. Material property number *mats* is assigned to designated cable elements. Midpoint of element must lie within the range to be changed. (All cable elements and nodes default to **mat=1**. The maximum value for *matg* or *mats* is 50.)

4. Domain Characteristics

dmat *n*

Material property number *n* is assigned to designated domains. (All domains initially default to **dmat=1**. The maximum value for *n* is 50.)

CONFIG

CONFIG keyword <keyword ... >

This command allows the user to specify, in advance, optional calculation modes that need extra memory to be assigned to each zone or gridpoint. The options are cell space detection logic, plane stress, fluid flow, creep, user-defined models and heat transfer. If any of these options are desired, **CONFIG** *must* be given before the **BLOCK** command. The following keywords apply.

axi axisymmetry. This mode works for mechanical calculation for deformable blocks only. It cannot be used with fluid flow or thermal calculation. Neither is it valid for any structural elements.

cell *n, n*
alternate contact detection logic. *n, n* refers to the number of cells in the *x*- and *y*-directions. This refers to the method used in *UDEC* to determine which blocks to consider for contact. This method is slower than the “domain” logic, but allows models with blocks that may bounce from one block to another. Fluid flow is not available with the cell logic.

cppudm extended and user-defined constitutive models

creep creep analysis

energy turns on energy summation.

fluid allocates additional memory structures used for fluid-flow logic in *UDEC*.

p_stress plane stress (only for elastic analysis or zone model **ss**)

thermal thermal analysis (see [Section 3](#) in **Special Features**)

CONTINUE

CONTINUE This command allows the user to resume reading a data file. Reading of the data file will pause if a **PAUSE** command is encountered, or if there is an error in a command line. If an error has occurred, the user may edit the input line using the interactive input editing keys (see [Table 1.1](#)) to correct the error. **CONTINUE** will then resume reading the data file on the next line.

NOTE: Changes to the input line are local, and are *not* written to the original data file.

COPY

COPY <filename>

Execution of this command causes a hardcopy plot of the previous screen plot to be made. If the optional filename is specified, output for this plot will be directed to the specified file. If no file name is specified, the plot data will be sent to the currently defined output device (see the **SET output** and **SET plot** commands). The default device for interactive operation is **UDEC.PNG**.

CRACK

CRACK table *n* <id = *n*> <join>
 x1 y1 x2 y2 <range. . . > <id = *n*> <join>

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. Two cracks will be connected if their endpoint locations are within a distance of twice the rounding length from each other. All discontinuous cracks that do not link to form blocks are deleted when the **GENERATE** or **CYCLE** command is executed.

The **CRACK table** form of the command will generate a continuous crack from the coordinates in table *n*. The data in the table must be previously defined.

The **CHANGE** command should be used to assign material property and constitutive numbers to the contacts along the crack. (NOTE: **CRACK** commands must be issued before blocks are made deformable.)

The optional ID number is used to specify the joint ID number for any contacts created by use of this command. If no ID is specified, a sequential number is used.

If the optional **join** keyword is used, any new contacts formed because of the crack will be joined contacts. Contacts that have been joined will be listed as having a constitutive model of 10.

CYCLE

CYCLE***n***

<keyword>

Executes ***n*** timesteps. (**CYCLE** 0 is permitted as a check on data.) If the <Esc> key is pressed during execution, *UDEC* will return control to the user after the current cycle is completed. (Also see **STEP** and **SOLVE**.)

Alternatively, cycling can be controlled by specifying one of the following keywords.

continue

This keyword causes the calculation to continue execution of the **CYCLE** command after interruption, by use of the <Esc> key. The remaining cycles will be performed. If the **CYCLE** command was issued from a remote data file, the file will continue to be read after cycling is complete.

ftime***f***

If the fluid flow calculator is on, fluid steps are executed. ***f*** is the flow duration time. **CYCLE *n*** can also be used for a flow calculation, in which ***n*** is the number of flow steps.

NOTES:

- (1) The fluid timestep must be set by the user (see the **SET dtflow** command).
- (2) During each flow step, a number of mechanical steps will be executed (see **SET maxmech**).
- (3) If the <Esc> key is touched, the run is interrupted after the next fluid step is completed.

time***t***

t is mechanical time duration, in seconds, for this increment of cycling.

DAMPING

DAMPING keyword

selects damping type for static and dynamic analysis. The following keywords apply.

auto <*fac mult1 mult2*>

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 values of 0.99 for *mult1* and 1.05 for *mult2* are optimum in most cases.

combined <*value*>

combined local damping (default for creep modeling). By default, the damping value is 0.8. This type of damping is useful for models where there is a constant motion in one direction. The other forms of damping rely on the velocity changing sign to be effective.

contact *mat 1, mat 2, Beta-j-n, Beta-j-s, Beta-p-n, Beta-p-s*

allows setting of different stiffness damping factors, depending on the material numbers and contact types. *mat 1* and *mat 2* refer to the material numbers of the blocks in contact. *Beta-j-n* and *Beta-j-s* are the normal and shear stiffness damping factors to be used for an edge-to-edge contact. *Beta-p-n* and *Beta-p-s* are the normal and shear stiffness damping factors to be used for corner-to-edge contacts. This method of setting the stiffness damping factors gives greater control for block bounce simulations.

The general equation for the stiffness damping force is

$$f = -\beta \times K \times l \Delta\gamma$$

where f = damping force;

β = damping factor;

K = stiffness;

l = length of contact; and

$\Delta\gamma$ = change in velocity.

DAMPING *fcrit freq*

fcrit freq <stiffness> <mass>

This 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: By specifying stiffness damping, the critical timestep for numerical stability will automatically be reduced. It is still possible for instability to result if large deformation occurs. In such a case, lower the timestep with the **FRACTION** command.) Damping considerations for dynamic analysis are discussed in more detail in [Section 4](#) in **Special Features**.

initial The keyword **initial** sets the **auto** damping constant to its initial (high) value (same as the **RESET damp** command).

local <value>

local damping. The damping *value* is 0.8 by default. Local damping is the default for static analysis.

NOTE: Mass scaling (see the command **MSCALE**) is performed automatically when the command **DAMPING auto** or **DAMPING local** is issued. Mass scaling is turned off when the command **DAMPING 0,0** is invoked.

DEFINE

DEFINE function-name

END **DEFINE** and **END** are commands used to define a function written in *FISH*, the embedded language built into *UDEC*. Statements (described in [Section 1](#) in the ***FISH** volume*) between the **DEFINE** and **END** commands are compiled and stored in compact form for later execution. Compilation errors are reported as the statements are processed. These “source” statements are not retained by *UDEC*. Hence, *FISH* functions normally should be prepared as data files that can be corrected and modified if errors are found.

FISH is a useful means by which to create new variables, to print or plot, to control conditions during *UDEC* execution, to create special distributions of properties, or to analyze *UDEC* output in some special way. [Section 1](#) in the ***FISH** volume* describes the operation and use of *FISH* in detail.

DELETE

DELETE <range ... >

All blocks with centroids in the optional **range** (see [Section 1.1.3](#)) are deleted.

If no range is specified, the range includes the entire model.

For example, small blocks within a range can be selectively deleted by using the **range** keyword phrase **area *a***. If the block area is smaller than *a*, the block will be deleted.

DUMP

DUMP *n m* <**offset**>

dumps memory to the screen from the main array beginning at address *n*, and continuing for *m* address locations. A number of internal indices are also listed. The item “mfree” gives the lowest memory location that is currently free.

Using the keyword **offset** causes the addresses to be displayed as offsets from *n*.

This is a diagnostic tool and does not affect the running of any models.

EDGE

EDGE *emin*

The minimum block edge is set to *emin*. (The default is twice the rounding length.) *emin* must be greater than or equal to twice the rounding length (same as the **SET edge** command). If used, **EDGE** *must* be specified prior to the **BLOCK** command.

FBOUNDARY keyword *v* ...

A porous-medium grid is created around the outer boundary of the model with the **FBOUNDARY** command. The following keywords and associated parameters are available.

k11 *k11*

k12 *k12*

k22 *k22*

three components of the permeability tensor (in which *k21* = *k12*)

n *n*

number of radial elements around the outer boundary. (The default is *n* = 2.)

pp *p_o*

initial fluid pressure at boundary and throughout porous-medium grid. (If a gradient is specified, *p_o* is the pore pressure at the origin of the grid.)

pxgrad *pxg*

initial pressure gradient along the *x*-axis

pygrad *pyg*

initial pressure gradient along the *y*-axis

NOTE: The pressure at any internal node is computed as

$$p = p_o + pxg \cdot x + pyg \cdot y$$

where *x* and *y* are the coordinates of the node.

radius *rad*

total radius of mesh, with origin at the centroid of the block

rat *rat*

ratio between radial size of an element on adjacent element. (The default *rat* = 1.)

NOTES:

1. Nodes at the outer boundary of the grid retain a fixed fluid pressure.
2. The fluid boundary *must* be created for a single-block system (i.e., before any joints are created). The number of fluid-boundary zones in a circumferential direction is equal to the number of sides between corners on the single block that is created. Use the **BLOCK angle** command to control the number of corners.
3. A **BOUNDARY** command must be given before cycling, because this boundary serves to transfer information between the flow calculations in the rock joints and the flow calculation in the porous-medium region. For example, the command **BOUNDARY pp = 0** will establish the boundary linked list.
4. At present, the porous-medium logic is only intended for steady-state flow analysis; a relaxation factor is used for convergence.
5. The formulation is small-strain only; it does not allow for changes in boundary linkages.
6. The formulation is for fully saturated flow. No free-flow surface may be modeled.
7. See [Section 2.2.7](#) in **Special Features** for an explanation of the formulation, and example applications.

FFIELD

FFIELD keyword <keyword... > <**range**... >

This command is used to manually manipulate the free-field boundaries. In most cases, it is easier to use the automatic free field (see **BOUNDARY ffield**). A dynamic free field is generated. This dynamic free field consists of a one-dimensional 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 ffield** command. The free-field boundary may be applied to one or both lateral boundaries of the problem. Both lateral boundaries should be vertical, but they may have different heights. See [Section 4 in Special Features](#) and [Section 4 in the Example Applications](#) for further explanation and application of this command.

delete Delete the free-field boundary.

ff_bulkmod ν
far-field bulk modulus*

ff_density ν
far-field density*

ff_nu ν
far-field Poisson's ratio*

ff_shearmod ν
far-field shear modulus*

ff_ymod ν
far-field Young's modulus*

mat i
far-field property number

* Far-field elastic properties can be specified in three ways: by material number (in which case the properties are specified using the **PROPERTY** command); by bulk and shear modulus; and by Young's modulus and Poisson's ratio.

For the commands above, an optional far field range may be specified by using the following **range** phrases.

range	keyword	
	left	applies only to left-hand side free-field mesh
	right	applies only to right-hand side free-field mesh
	yrange	<i>yl yu</i> restricts the range to $yl < y < yu$

The free-field mesh is generated with the following keywords.

gen	keyword	
	left	only left-hand side free-field is generated (optional)
	npoints	<i>n</i> number of free-field gridpoints (required)
	right	only right-hand side free-field is generated (optional)
	yrange	<i>yl yu</i> lower and upper y-coordinates (required)

The free-field calculation can be switched on and off with the following keywords.

off	switches off the free-field calculation.
on	switches on the free-field calculation.

Free-field zones can be assigned material models with the following keywords.

change	keyword	
	cons	<i>n</i> assigns constitutive models to free-field zones.
	mat	<i>n</i> assigns material numbers to free-field zones.

FFIELD

Free-field stresses, pore pressures, displacements and velocities may be initialized with the following keywords.

initial	keyword
	pp <i>value</i> < <i>grad</i> >*
	zone pore-pressure
	sxx <i>value</i> < <i>grad</i> >*
	zone <i>xx</i> -stress
	sxy <i>value</i> < <i>grad</i> >*
	zone <i>xy</i> -stress
	syy <i>value</i> < <i>grad</i> >*
	zone <i>yy</i> -stress
	szz <i>value</i> < <i>grad</i> >*
	zone <i>zz</i> -stress
	xdis <i>value</i>
	gridpoint <i>x</i> -displacement
	xvel <i>value</i>
	gridpoint <i>x</i> -velocity
	ydis <i>value</i>
	gridpoint <i>y</i> -displacement
	yvel <i>value</i>
	gridpoint <i>y</i> -velocity

* An optional gradient is applied by assigning a value to ***grad*** where $\text{actual_value} = \text{value} + \text{grad} \cdot y$.

Boundary conditions are applied with the following keywords.

base	keyword	
	assigns conditions to the free-field base	
	sxy	<i>value</i> <i>xy-stress</i>
	syy	<i>value</i> <i>yy-stress</i>
	xvel	<i>value</i> <i>x-velocity</i>
	xvisc	viscous boundary in the <i>x</i> -direction
	yvel	<i>value</i> <i>y-velocity</i>
	yvisc	viscous boundary in the <i>y</i> -direction
	top	keyword
	assigns conditions to the free-field top	
	sxy	<i>value</i> <i>xy-stress</i>
	syy	<i>value</i> <i>yy-stress</i>
	xvel	<i>value</i> <i>x-velocity</i>
	xvisc	viscous boundary in <i>x</i> -direction
	yvel	<i>value</i> <i>y-velocity</i>
	yvisc	viscous boundary in <i>y</i> -direction

FFIELD

Time-varying stresses or velocities can be defined by using the **history** keyword phrase (see the **BOUNDARY** command) on the same input line, following the boundary condition keyword.

Other options may be specified with the following keywords.

delete deletes free-field mesh.

set_ref current free-field stresses are taken as static reference stresses. This command is automatically executed when the free-field is applied to the grid (i.e., **BOUNDARY ffield**).

FIX

FIX <keyword> <**range**...>

All blocks with centroids in the specified range (see [Section 1.1.3](#)) have current velocities fixed. Deformable blocks cannot be fixed. Velocity histories may be applied using *FISH* functions. The following keywords limit fixing to a specified degree of freedom.

rotation	fix only rotation
x	fix only in <i>x</i> -direction
y	fix only in <i>y</i> -direction

FLUID keyword *value* <keyword *value* ... >

Fluid flow properties are set to *value* by using the following keywords.

bulkw *value*

wetting bulk modulus [stress]

bulkw is used for compressible (both single and two-phase) fluid flow (**SET flow compressible**) and fluid pressure calculations. If the command **SET flow off** is given and **bulkw** is nonzero, wetting fluid pressures are generated due to joint deformation. **bulkw** is not required for **SET flow steady** or **SET flow incompressible**. The default is **bulkw** = 0.

cohw *value*

yield strength [stress]

cohw is the yield strength for non-Newtonian (wetting) fluids (e.g., Bingham flow; see [Section 2](#) in **Special Features**). The default is **cohw** = 0.

dens *value*

wetting fluid density [mass/volume]

dens is used to calculate the hydrostatic pressure in wetting fluid gradient, if gravity is present.

dtable *n*

Table number *n* is used to look up variations of fluid density as a function of temperature.

gas_alpha *v*

sets gas constant alpha*, used for calculating gas fluid density and bulk modulus.

gas_bulkmin *v*

sets minimum bulk modulus for gas. Used for zero pressure.

gas_constant *v*

sets gas constant B*, used for calculating gas fluid density.

* gas density = B*(pressure)^α; gas bulk modulus = $\frac{\text{pressure}}{\alpha}$

FLUID

gas_density

gas_densitymin *v*

sets minimum density for gas. Used for zero pressure.

ktable *n*

Table number *n* is used to look up variations of joint permeability as a function of temperature.

nwbulk *value*

non-wetting fluid bulk modulus if **gas** is **off**, or ratio between non-wetting fluid bulk modulus and non-wetting fluid pressure if **gas** is **on**.

nwbulk is used for two-phase (**SET twophase on**) fluid flow and fluid pressure calculations. If **twophase** is **on**, non-wetting fluid bulk modulus must be larger than zero, and pressures are generated in non-wetting fluid due to joint deformation.

nwdens *value*

non-wetting fluid density [mass/volume]

nwdens is used to calculate the hydrostatic pressure gradient in non-wetting fluid, if gravity is present.

FRACTION

FRACTION *fb* <*fz*> <*fw*>

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

FRACTION

FREE

FREE <range... >

All rigid blocks with centroids in the selected range (see [Section 1.1.3](#)) are set free.
By default, all blocks are free initially.

GENERATE

GENERATE keyword *value* <...> <**range**...>

All blocks with centroids lying within the optional **range** (see [Section 1.1.3](#)) are discretized into deformable triangular finite-difference zones. If no range is given, then all blocks will be discretized.

Zone generation can be performed automatically or manually. For automatic generation, one of the following keywords *must* be specified.

edge *edmax*

automatic generation of zones for an arbitrarily shaped block. The parameter *edmax* defines the maximum edge length of the triangular zones.

mixed subdivides triangular zones into 3 subzones with an overlay of 4 zones (12 total). The strain of these zones is averaged to avoid hourglass effects. This command only subdivides existing zones; **GENERATE edge** or **GENERATE quad** must be used first.

quad *xw* <*yw*>

automatic generation of diagonally opposed triangular zones to improve plastic flow calculation. Parameters *xw* and *yw* are zone widths in the *x*- and *y*-directions. If *yw* is not given, the parameter defaults to *xw*. This form of zoning will not work for all block shapes; blocks must contain 4 or 5 corners, and at least 3 of the corners must not be colinear. For blocks with 5 corners, the block is split into quadrilateral regions before zoning. For best results, the block sides should not be too different in size. If **GENERATE quad** does not work, or produces irregular zoning, the **GENERATE edge** command should be used for the remaining blocks.

single mixed discretization with single layer of zones (no overlay), when used instead of the **mixed** keyword

An optional keyword is provided to assist with automatic generation:

match forces corners to be added to blocks to ensure that all gridpoints will match those on adjoining blocks.

GENERATE

Manual generation must be performed for *each* individual block. Only one block should be within the specified range. For manual generation of zones, the following keyword is used.

manual **gridpoint** *glist* ... **zone** *zlist* ...

For manual generation, a list of **gridpoints** *glist* and **zones** *zlist* must be given. The format for *glist* is *x1 y1 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 *ll 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., the last gridpoint is number 1). The triple must be given in *clockwise* order. (Perform **CYCLE 0**, then **PRINT zone state** to ensure that no negative zone volumes are created.) Both *glist* and *zlist* may extend over an arbitrary number of continuation lines, but doubles and triples should not be split over two lines. When using manual zoning, 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.

The following optional keyword can be used with all other **GENERATE** keywords.

rezone This allows a block to be rezoned. This command should be used to correct problems with zone degeneration that can result from deformation of null blocks. It may be necessary to rezone a block containing null material if the block is then assigned non-null material. When the block is rezoned, stresses in the block are set to zero. All gridpoints along the edges of blocks are retained and used for the new zoning. The zone edge lengths used with **rezone** can be specified to generate smaller zoning.

GIIC

GIIC Invoke graphical user interface.

GRAVITY

GRAVITY *gx gy*

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

GROUP

GROUP <keyword> <**delete**> name <range>

Groups are sets of objects identified by a group name. Groups cannot overlap; each object may only belong to one group. A new group name will replace an old name. The **GROUP** command assigns a name to all objects within a selected range. The range can be given in several forms (see [Section 1.1.3](#)). If no range is specified, the command applies to the entire model. The objects named to groups may be blocks, zones or contacts:

block specify groups for blocks

joint specify groups for joints

zone specify groups for zones

If the name *name* does not exist, it is created. The name *name* may contain spaces but, in that case, the whole name must be enclosed by single quotation marks. For example, the following commands are valid.

```
group zone fred
group zone 'have a nice day'
```

The optional keyword **delete** causes the named group to be deleted (and any range ignored). Note that **GROUP** assignments are independent of models existing in zones or not (i.e., **GROUP** names can be given to ranges containing null models). An object may only be assigned to one group; new assignments overwrite existing assignments.

The command **PRINT groups** lists the currently available user-defined group names.

The **PLOT group zone** command plots the groups assigned to zones as colored blocks.

HEADING

HEADING <'string'>

The title for a *UDEC* model is printed on subsequent output plots and recorded in the save files. (This command is the same as the **TITLE** command.)

If 'string' is present, then that token is parsed as a string and used as the current title. This token can be a *FISH* string variable; if it is, do not enter the token in single quotes. If 'string' is not present, then a `head>` prompt appears, and the next line input is taken as the title. If in interactive mode while this line is being read, simply hitting <ENTER> will retain the old title. To clear the title (causing the title window to disappear on output plots), give a null string (two consecutive single quotes) with the **HEADING** command,

```
heading ''
```

HIDE

HIDE <range phrase> <keyword>

All blocks with centroids in the range defined by the range phrase are made invisible; they can be made visible again with the **SHOW** command. When blocks are invisible, they are not split by the **JSET**, **CRACK** or **SPLIT** command; in this way, discontinuous joints may be made. However, the invisible blocks interact normally with other blocks. In general, invisible blocks are unaffected by the **BOUNDARY**, **CHANGE**, **CRACK**, **DELETE**, **FIX**, **FREE**, **GENERATE**, **GROUP**, **JMODEL**, **JOINT**, **JSET**, **SPLIT** and **ZONE** commands.

The range types are described in [Section 1.1.3](#). If no range is specified, all blocks are hidden.

The following keywords are available.

area *v*

All blocks with area smaller than the specified value will be hidden.

deformable hides blocks that are fully deformable.

rigid hides blocks that are rigid.

HISTORY

HISTORY **<ncyc = *n*>** keyword **<keyword>** ...
FISH symbol

A time history of selected variables is stored every **ncyc** timesteps. Variables are accessed by their history index number, which corresponds to the order in which they were defined using the **HISTORY** command. History index numbers can be listed by using the **PRINT history** command. **ncyc** is the same for all histories, and need only be given once prior to timestepping; *n* defaults to 10 if not defined explicitly. A time history of as many as 500 variables can be made in one run. Several history keywords are defined for block or contact numbers. Block numbers can be found via the **PLOT blocks num** command, and contact numbers via the **PLOT contacts num** command. The following keywords are available.

bfm	<i>n</i>	block centroid moment for rigid block number <i>n</i>
bfx	<i>n</i>	block <i>x</i> -direction centroid force for rigid block number <i>n</i>
bfy	<i>n</i>	block <i>y</i> -direction centroid force for rigid block number <i>n</i>
crt del		creep timestep
crt time		creep time
damp		auto damping parameter (see Section 1.2.7 in Theory and Background)
edxx	<i>x y</i>	<i>xx</i> -strain rate at location nearest to (<i>x,y</i>) (deformable blocks only)
edxy	<i>x y</i>	<i>xy</i> -strain rate at location nearest to (<i>x,y</i>) (deformable blocks only)
edyy	<i>x y</i>	<i>yy</i> -strain rate at location nearest to (<i>x,y</i>) (deformable blocks only)
energy		energy components. The histories of the 20 components of stored and dissipated energy are recorded. (See PRINT energy for a listing of the components.)
exx	<i>x y</i>	<i>xx</i> -strain at location nearest to (<i>x,y</i>) (deformable blocks only)

HISTORY

exy

exy

 $x \ y$ xy -strain at location nearest to (x,y) (deformable blocks only)

eyy

 $x \ y$ yy -strain at location nearest to (x,y) (deformable blocks only)

ffsxx

 $y \ i$ free-field xx -stress at y for side i (1 = left, 2 = right)

ffsxy

 $y \ i$ free-field xy -stress at y for side i (1 = left, 2 = right)

ffsyy

 $y \ i$ free-field yy -stress at y for side i (1 = left, 2 = right)

ffxa

 $y \ i$ free-field x -acceleration at y for side i (1 = left, 2 = right)

ffxd

 $y \ i$ free-field x -displacement at y for side i (1 = left, 2 = right)

ffxv

 $y \ i$ free-field x -velocity at y for side i (1 = left, 2 = right)

ffya

 $y \ i$ free-field y -acceleration at y for side i (1 = left, 2 = right)

ffyd

 $y \ i$ free-field y -displacement at y for side i (1 = left, 2 = right)

ffyv

 $y \ i$ free-field y -velocity at y for side i (1 = left, 2 = right)

flowrate

 $x \ y$ wetting fluid flow rate at contact nearest (x, y) (see [Section 2](#) in **Special Features**)

flowtime

fluid flow time

increment

rate of change in energy components. Histories of the rate of change for 20 energy components are recorded.

HISTORY ndis

ndis	<i>x y</i>	normal displacement at contact nearest to (<i>x,y</i>)
nstr	<i>x y</i>	normal stress at contact nearest to (<i>x,y</i>)
nvel	<i>x y</i>	normal velocity at contact nearest to (<i>x,y</i>)
nwflow	<i>x y</i>	non-wetting fluid flow rate at contact nearest to (<i>x,y</i>) (see Section 2 in Special Features)
nwpp	<i>x y</i>	non-wetting fluid domain pressure at location nearest to (<i>x,y</i>)
pp	<i>x y</i>	wetting fluid domain pressure at location nearest to (<i>x,y</i>)
sat	<i>x y</i>	saturation at location nearest to (<i>x,y</i>)
sdis	<i>x y</i>	shear displacement at contact nearest to (<i>x,y</i>)
solve_local		local mechanical force ratio in model (as defined by the SOLVE r_type command)
solve_ratio		maximum mechanical force ratio in model (as defined by the SOLVE r_type command)
srat	<i>x y</i>	ratio of shear stress to shear strength at contact nearest to (<i>x,y</i>)
sstr	<i>x y</i>	shear stress at contact nearest to (<i>x,y</i>)
svel	<i>x y</i>	shear velocity at contact nearest to (<i>x,y</i>)

HISTORY

sxx

sxx

 $x \ y$ xx -stress at location nearest to (x,y) (deformable blocks only)

sxy

 $x \ y$ xy -stress at location nearest to (x,y) (deformable blocks only)

syy

 $x \ y$ yy -stress at location nearest to (x,y) (deformable blocks only)

szz

 $x \ y$ zz -stress at location nearest to (x,y) (deformable blocks only)

temperature

 $x \ y$ history of temperature of gridpoint nearest to (x,y) (deformable blocks only)

thtime

thermal time

unbal

maximum unbalanced gridpoint force in model

unbvol

maximum unbalanced fluid volume (for **SET flow incompressible**; see [Section 2 in Special Features](#))

vmax

maximum gridpoint velocity in model

xacc

 $x \ y$ x -acceleration at location nearest to (x,y)

xdis

 $x \ y$ x -displacement at location nearest to (x,y)

xhis

history of applied x -force history function

xvel

 $x \ y$ x -velocity at location nearest to (x,y)

yacc

 $x \ y$ y -acceleration at location nearest to (x,y)

ydis

 $x \ y$ y -displacement at location nearest to (x,y)

yhis

history of applied y -force history function

HISTORY **yvel****yvel** *x y*y-velocity at location nearest to (*x,y*)Histories of Real Time

In addition, special keywords are provided to allow the user to plot histories for transient calculations against real time:

crttime creep time**flowtime** real time for fluid-flow analysis (see [Section 2](#) in **Special Features**)**thtime** real time for thermal analysis (see [Section 3](#) in **Special Features**)User-Defined *FISH* Variables

Using *FISH*, histories of user-defined variables may be collected (see [Section 1](#) in the ***FISH* volume**).

Manipulating History Data

The following keywords for the **HISTORY** command allow the user to write histories to the screen or file, or manipulate the history data.

list prints lists of all history types and locations. (This command is the same as the **PRINT history** command.)

table *itab*

write history *n* to table *itab*. For example, **HISTORY write 1 Table 1** will put history number 1 into table number 1.

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.

write *n* <keyword> <fname>

writes variable (with history number *n*) time history to a file on the hard drive. The default file name is "UDEEC.HIS." The output can be limited to a specific range of steps by using the following keywords.

begin *ncb*

Histories will be output beginning with step number *ncb*.

HISTORY write end

end *nce*

Histories will be output ending with step number *nce*.

skip *nc*

Only one point for every *nc* history points recorded is actually output. For example, **skip 10** means that every 10th recorded point (starting with the first) will be displayed.

vs. *nhis*

History values will be written with another history (*nhis*) instead of the cycle number.

INITEM

INITEM *value* <**add_stress**> <*range* *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 unless the **add_stress** keyword is included.

INITIAL

INITIAL keyword <keyword> *value* <**grad** *gx gy*> <**range**...>

Block, gridpoint, zone and contact variables are assigned initial values. Loads and velocities apply to rigid block centroids only. Gridpoint and zone values apply to deformable blocks only. Initialized values for rigid blocks can be made permanent by using the **FIX** command. The following keywords are available.

aperture *value*

contact hydraulic aperture. This also sets the **azero** parameter for each contact so that $aperture = azero + u_n$, where u_n is the normal displacement.

brvel *value*

rigid block θ velocity

bxload *value*

rigid block x -load

bxvel *value*

rigid block x -velocity

byload *value*

rigid block y -load

byvel *value*

rigid block y -velocity

ftemp *value*

fluid temperature

ndis *value*

normal displacement for contacts

nforce *value*

normal force for contacts

nstress *value*

normal stress for contacts

nwpp***value***

domain non-wetting fluid pressures if **SET twophase on**. Note that wetting fluid pressure, non-wetting fluid pressure and saturation are dependent variables in the case of two-phase flow. Only two of those field variables can be specified independently; the third one is calculated. Therefore, the user can specify saturation, and either wetting fluid pressure or non-wetting fluid pressure. (If both pressures are specified, then the one that is specified later will be initialized as specified, and the other will be calculated.) Also note that at least one mechanical timestep should be executed, and the capillary pressure curve defined, before initialization of wetting or non-wetting fluid pressure in the case of two-phase flow.

pp***value***

domain wetting fluid pressures. Note that wetting fluid pressure, non-wetting fluid pressure and saturation are dependent variables in the case of two-phase flow. Only two of those field variables can be specified independently; the third one is calculated. Therefore, the user can specify saturation, and either wetting fluid pressure or non-wetting fluid pressure. (If both pressures are specified, then the one that is specified later will be initialized as specified and the other will be calculated.) Also note that at least one mechanical timestep should be executed, and the capillary pressure curve defined, before initialization of wetting or non-wetting fluid pressure in the case of two-phase flow.

sat***value***

domain saturation for fluid flow (**SET flow compressible** and **SET flow incompressible**). The default is 1.0 (saturated).

sdis***value***

shear displacement for contacts

sforce***value***

shear force for contacts

sstress***value***

shear stress for contacts

sxx***value***

zone *xx*-stress

INITIAL**sxy****sxy***value*zone *xy*-stress**syy***value*zone *yy*-stress**szz***value*zone *zz*-stress**xdis***value*gridpoint *x*-displacement**xvel***value*gridpoint *x*-velocity**ydis***value*gridpoint *y*-displacement**yvel***value*gridpoint *y*-velocity**zone_pp***value*

zone pore pressure. NOTE: Fluid flow is only in the joints.

WARNING: To initialize zone stresses and contact stresses consistently, the **INSITU** command should be used.

The following optional keywords may also be given *immediately* following the variable keyword.

add *value*

The keyword **add** adds *value* (including any specified variations) to existing values of the given variable. The default is to replace the existing value.

mult *value*

The keyword **mult** multiplies the existing values of the given variable by *value* (including any specified variations).

An optional keyword phrase **grad gx gy** can be used to specify an *x*- and *y*-gradient for any keyword when calculating the value *vtot*. The following equation is used.

$$vtot = value + (gx \cdot x) + (gy \cdot y)$$

INSITU

INSITU keyword ... <**range**...>

The **INSITU** command initializes block and contact stresses, domain pressures, zone pore pressure and hydraulic apertures. Initial zone stresses in all deformable blocks and stresses along joints between rigid blocks and/or deformable blocks in the optional **range** are set. If no range is specified, stresses are initialized for the entire model. Remember that compressive stresses are negative. The following keywords apply.

stress **sxxo sxyo syyo**

The *xx*-stress, *xy*-stress and *yy*-stress components are initialized to the values **sxxo**, **sxyo** and **syyo**.

szz **szzo**

zz-component of stress. The initial out-of-plane stresses in all deformable blocks are set to zero. These stresses can be prescribed with this keyword.

zone_pp switch to also set zone pore pressure based on the **pp**, **pxgrad**, **pygrad** or **ywtable** keyword. NOTE: Fluid flow is only in joints.

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

$$sxx = sxxo + (sxxx \cdot x) + (sxxy \cdot y)$$

$$sxy = sxyo + (sxyx \cdot x) + (sxyy \cdot y)$$

$$syy = syyo + (syyx \cdot x) + (syyy \cdot y)$$

xgrad and **ygrad** must be given in the same command as the **stress** keyword. (Use & for line continuation if input is too long.)

zgrad **szzx szzy**

where

$$szz = szzo + (szzx \cdot x) + (szzy \cdot y)$$

zgrad must be given in the same command as the **szz** keyword.

Optional keywords are available to control the initialization of stresses:

- block** If the **block** keyword is given, stresses are initialized only in blocks, not in contacts.
- joint** If the **joint** keyword is given, stresses are initialized only in contacts, not in blocks.

Domain pressures may be initialized with the following pore-pressure keywords. These keywords should be given by the same **INSITU** command that initializes stresses (see Note 1, below).

- pp** p_o
 fluid pressure in domains (default is $p_o = 0.0$)
- pxgrad** xg
 fluid pressure gradient in the x -direction (default is $xg = 0.0$)
- pygrad** yg
 fluid pressure gradient in the y -direction (default is $yg = 0.0$)

Pressures in all domains except the outer domain will be initialized to

$$p = p_o + x \cdot xg + y \cdot yg$$

where (x,y) are the coordinates of the domain center.

Alternatively, a domain pressure distribution corresponding to a water table may be created with the keyword

- ywtable** y_w
 Domains with a y -coordinate below y_w are assigned pressure, p , in which

$$p = -\rho_w g_y (y_w - y)$$

Alternatively, a varying water table can be specified with

table ***n***

n is the table number that defines the water table line. The command provides a pore-pressure distribution corresponding to a static head below a specified phreatic surface.

The fluid density, ρ_w , and the vertical component of gravity, g_y , must have been previously defined by the **FLUID** and **SET gravity** commands, respectively.

Contact hydraulic apertures may be initialized with the following keyword.

aperture ***value***

contact hydraulic aperture. This also sets the **azero** parameter for each contact so that $aperture = azero + u_n$, where u_n is the normal displacement.

An additional optional keyword:

nodis inhibits calculation of joint normal displacements (i.e., joint initial closure).

NOTES:

1. *UDEC* assumes block stresses to be *total* stresses, and joint stresses to be *effective* stresses. Therefore, the initialization of domain pressures and block/zone stresses must be done by a single **INSITU** command. Stresses specified by the **stress** keyword are assumed to be *total* stresses. Joint stresses are calculated by adding the specified domain pressures to these stresses.
2. If only stress keywords are given, only zone and contact stresses are set; pore pressures are not changed.
3. If only pore pressure keywords are given, then only domain pore-pressure conditions are changed, without changing contact/zone effective stresses. If the keyword **zone_pp** is given, then zone pore pressures are set. Zone stresses are added to the zone pore-pressure increment, so effective stresses are unchanged. Immediately after the command, the system will not be balanced (e.g., if the water table rises, domain pore pressure will increase, and joints will have to open to accommodate the change). Zones also have to expand as a result of the pore pressure increase.
4. If stress and pore pressure keywords are given, then stresses are total stresses. Domain pressures, zone total stresses and contact effective stresses are set. If the **zone_pp** keyword is present, then zone pore pressures are also set.

5. **jmat** and **jcons** must be specified prior to the **INSITU** command in order to calculate joint normal displacement, u_n , due to in-situ stresses.
6. For **jcons = 3**, joint initial closure is only calculated for the **INSITU** command if joint normal stiffness is constant (i.e., **jen** = 0).

JDELETE

JDELETE Joints that do not completely intersect a block are deleted. (This command is performed automatically when the **GENERATE** command and **STEP**, **CYCLE** or **SOLVE** command are issued.)

JMODEL

JMODEL <model name> <keyword *value* ... > ... <range... >

This command is used to assign user-defined joint constitutive models (UDM) to contacts. This command is also used to assign properties to these contacts. The UDM option is required to use this command. Also, **CONFIG cppudm** must be used for the command to be accepted. Models exist as dynamic linked libraries, and must be loaded prior to assignment. Models that are placed in the “plugins\jmodels” folder will be automatically loaded at runtime. Otherwise, this is done with the **JMODEL load filename** command. The models may then be assigned to contacts via the **JMODEL** model name command.

At present, 1 example joint constitutive model is available through use of this command. The model is described in writing new constitutive models in [Section 4.2.2](#) in **Constitutive Models**. Built-in joint models that already exist for some or all of the contacts in the given range (as specified by **CHANGE cons**) are ignored. The **model** keyword assigns joint constitutive models via the following names.

Model name	Filename	Description
jmohr	jmodelmohr003.dll	Mohr-Coulomb joint model

The other keywords assign material properties and are model-specific. Other contact properties entered via the **PROPERTY** command are ignored if the range of contacts falls within that specified by a **JMODEL** command.

The model must be assigned *before* assigning properties. If properties that are not consistent with the chosen model are given, a warning message will be issued to inform the user that the unneeded properties were not accepted. If a required property is *not* specified, the default will be used.

WARNING: When a **model** is invoked over a specified range, all properties associated with that model are initialized to zero in that range. Properties previously assigned to that range must be specified again, even if their values have not changed.

The **SET jcondf** command may be used to define a user-defined model to be used for new contacts.

The following properties are for the joint models.

jmohr (Coulomb slip with residual strength)

(1)	ares	residual aperture at high stress (used in <i>UDEC</i> only for aperture calculation) [length]
(2)	azero	aperture for zero normal stress (used in <i>UDEC</i> only for aperture calculation) [length]
(3)	cohesion	joint cohesion [stress]
(4)	dilation	joint dilation angle [degrees]
(5)	empb	empirical multiplier for fluid flow law (used in <i>UDEC</i> only for fluid flow) (default = 1)
(6)	expa	exponent of joint hydraulic aperture (used in <i>UDEC</i> only for fluid flow) (default = 3)
(7)	friction	joint friction angle [degrees]
(8)	grdexp	gradient exponent
(9)	jkn	joint normal stiffness [stress/displacement]
(10)	jks	joint shear stiffness [stress/displacement]
(11)	jperm	joint permeability constant (see Eq. (2.3) in Special Features , also called joint permeability factor) [1/(stress · time)] – used only in <i>UDEC</i> for fluid flow calculations
(12)	jtension	joint tensile strength [stress]
(13)	nstable	normal stiffness table
(14)	rescohesion	joint residual cohesion [stress]
(15)	resdilation	joint residual dilation [degrees]
(16)	resfriction	joint residual friction angle (default = friction) [degrees]
(17)	restension	joint residual tensile strength [stress]
(18)	sstable	shear stiffness table
(19)	zerdilation	shear displacement for zero dilation (default: dilation not affected if zerdilation = 0.0) [displacement]

The default value for these properties (with the exception, as noted, of **empb**, **expa** and **resfriction**) is zero.

In this model, an internal flag is set for each joint segment when the joint tensile or shear strength is exceeded. If a joint is fractured (i.e., flag is set), then joint tensile strength, joint friction angle and joint cohesion are set to residual values.

JOIN_BLOCK

JOIN_BLOCK <off> <range>

This command is similar to the **JOIN_CONTACT** command. It joins or unjoins all contacts between blocks with centroids in a specified range. Contacts will be listed with a constitutive model of 10. Contacts that are joined cannot slide or open.

This command works best if used after all joints have been formed and the model has been zoned. Otherwise, the joining of the block edges may not be continuous.

UDEC will slave the gridpoints from blocks on either side of a contact. If there are no matching gridpoints, then a special contact constitutive model is used. Properties of the special constitutive model are determined automatically by *UDEC*.

The stiffnesses of the joined contacts are calculated automatically. The normal stiffness is based on an average zone stiffness multiplied by a factor (**join_ratio**). The default factor is 100. *UDEC* will use the calculated stiffnesses or the largest joint stiffness assigned in the model, whichever is greater. The shear stiffness is half the normal stiffness. The user may define stiffnesses for the joined contacts by using the command **SET join_jkn**, **SET join_jks** or **SET join_ratio**. Edges of blocks that have been joined will not be plotted by the **PLOT blocks** command. To plot joined contacts, use the **PLOT joined** command.

JOIN_CONT

JOIN_CONTACT <**off**> <**range**>

joins or unjoins all contacts in a specified range. Contacts will be displayed with a constitutive model of 10. Contacts that are joined cannot slide or open.

This command works best if used after all joints have been formed and the model has been zoned. Otherwise, the joining of the block edges may not be continuous.

UDEC will slave the gridpoints from blocks on either side of a contact. If there are no matching gridpoints, then a special contact constitutive model is used. Properties of the special constitutive model are determined automatically by *UDEC*.

The stiffnesses of the joined contacts are calculated automatically. The normal stiffness is based on an average zone stiffness multiplied by a factor (**join_ratio**). The default factor is 100. *UDEC* will use the calculated stiffnesses or the largest joint stiffness assigned in the model, whichever is greater. The shear stiffness is half the normal stiffness. The user may define stiffnesses for the joined contacts by using the command **SET join_jkn**, **SET join_jks** or **SET join_ratio**. Edges of blocks that have been joined will not be plotted by the **PLOT blocks** command. To plot joined contacts, use the **PLOT joined** command.

JOINT

JOINT <model name> <keyword *value*...> ... <range...>

This command associates a constitutive model and/or one or more properties with all contacts located in the given range. At present, five joint constitutive models are available through the use of this command:

- Coulomb slip (point contact);
- Coulomb slip (area contact);
- Coulomb slip with residual strength (area contact);
- continuously yielding; and
- Barton-Bandis (optional model).

The Coulomb slip models are described in [Section 1.2.4](#) in **Theory and Background**, the continuously yielding model is described in [Section 2](#) in **Constitutive Models**, and the Barton-Bandis model is described in [Section 3](#) in **Constitutive Models**. Any joint constitutive model that already exists for some or all of the contacts in the given range (as specified by the **CHANGE jcons** command) is ignored. The **SET jcondf** command may be used to specify a model and properties to be used for new contacts.

The **model** keyword assigns constitutive models via the following names.

model	keyword
--------------	---------

The following names are available.

area	Coulomb slip (area contact)
bb	Barton-Bandis joint model (optional model)
cy	continuously yielding model
point	Coulomb slip (point contact)
residual	Coulomb slip with residual strength (area contact)

The other keywords assign material properties, and are model-specific. The property keywords and associated model type for the Coulomb slip and the continuously yielding models are presented in [Table 1.4](#). The keywords associated with the Barton-Bandis model are described in [Section 3](#) in **Constitutive Models**.

If properties that are not consistent with the chosen model are given, a warning message will be given to inform the user that the unneeded properties were not accepted. If a required property is *not* specified, the default given in [Table 1.4](#) will be used. The required keywords for each model are listed following the table. Joint properties can be printed with the **PRINT property** keyword command, where keyword

JOINT

is the property name. Joint properties can be plotted with the **PLOT property** keyword command.

Table 1.4 Property keywords for the JOINT model command

Keyword	Description	Default Value	P	Model*		
				A	R	C
ares	residual aperture at high stress (m)	0		x	x	x
azero	aperture for zero normal stress (m)	0		x	x	x
ccohesion	contact cohesion (N)	0	x			
cdilation	contact dilation (degrees)	0	x			
cfriiction	contact friction angle (degrees)	0	x			
cperm	contact permeability ($m^2 Pa^{-1} sec^{-1}$)	0	x			
ctension	contact tensile strength (N)	0	x			
empb	empirical multiplier for fluid flow	1.0		x	x	x
expa	exponent joint hydraulic aperture	3.0		x	x	x
grdexp	gradient exponent for fluid flow	0		x	x	x
jcohesion	joint cohesion (Pa)	0		x	x	
jdilation	joint dilation (degrees)	0		x	x	
jen	exponent of joint normal stiffness	0				x
jes	exponent of joint shear stiffness	0				x
jfriction	joint friction angle (degrees)	0		x	x	x
jif	joint initial friction angle (degrees)	0				x
jkn	joint normal stiffness (Pa/m)	0		x	x	x
jks	joint shear stiffness (Pa/m)	0		x	x	x
jperm	joint permeability ($Pa^{-1} sec^{-1}$)	0		x	x	x
jrescoh	joint residual cohesion (Pa)	0			x	
jrfric	joint residual friction angle (degrees)	0			x	
jrough	joint roughness parameter (m)	0				x
jrtension	joint residual tensile strength (Pa)	0			x	
jtension	joint tensile strength (Pa)	0		x	x	
kn	contact normal stiffness (N/m)	0	x			
ks	contact shear stiffness (N/m)	0	x			
maxjkn	max value of joint normal stiffness (Pa/m)	0				x
maxjks	max value of joint shear stiffness (Pa/m)	0				x
minjkn	min value of joint normal stiffness (Pa/m)	0				x
minjks	min value of joint shear stiffness (Pa/m)	0				x
nstable	normal stiffness table	—		x	x	
nwcperm	non-wetting fluid contact permeability	0	x			
nwjperm	non-wetting fluid joint permeability	0		x	x	x
zdilation	shear displacement for zero dilation (m)	0		x	x	

* P: Coulomb slip (point contact); R: Coulomb slip (with residual strength)

A: Coulomb slip (area contact); C: continuously yielding

Note: Listed units are only examples; any consistent set of units may be used.

Coulomb slip (point contact)

- | | | |
|-----|-------------------|--|
| (1) | ccohesion | contact cohesion |
| (2) | cdilation | contact dilation angle |
| (3) | cfriiction | contact friction angle |
| (4) | cperm | wetting fluid contact permeability |
| (5) | ctension | contact tensile strength |
| (6) | kn | contact normal stiffness |
| (7) | ks | contact shear stiffness |
| (8) | nwcperm | non-wetting fluid contact permeability |

Coulomb slip (area contact)

- | | | |
|------|------------------|---|
| (1) | ares | residual aperture at high stress |
| (2) | azero | aperture for zero normal stress |
| (3) | empb | empirical multiplier for fluid flow law |
| (4) | expa | exponent for joint hydraulic aperture |
| (5) | grdexp | gradient exponent for fluid flow |
| (6) | jcohesion | joint cohesion |
| (7) | jdilation | joint dilation angle |
| (8) | jfriction | joint friction angle |
| (9) | jkn | joint normal stiffness |
| (10) | jks | joint shear stiffness |
| (11) | jperm | wetting fluid joint permeability |
| (12) | jtension | joint tensile strength |
| (13) | nstable | normal stiffness table |
| (14) | nwjperm | non-wetting fluid joint permeability |
| (15) | zdilation | shear displacement for zero dilation |

Coulomb slip with residual strength (area contact)

- | | | |
|-----|------------------|---|
| (1) | ares | residual aperture at high stress |
| (2) | azero | aperture for zero normal stress |
| (3) | empb | empirical multiplier for fluid flow law |
| (4) | expa | exponent of joint hydraulic aperture |
| (5) | jcohesion | joint cohesion |
| (6) | jdilation | joint dilation angle |
| (7) | jfriction | joint friction angle |
| (8) | jkn | joint normal stiffness |

JOINT

- (9) **jks** joint shear stiffness
- (10) **jperm** wetting fluid joint permeability
- (11) **jrescoh** joint residual cohesion
- (12) **jrfric** joint residual friction angle
- (13) **jrtension** joint residual tensile strength
- (14) **jtension** joint tensile strength
- (15) **nstable** normal stiffness table
- (16) **nwjperm** non-wetting joint permeability
- (17) **zdilation** shear displacement at zero dilation

Notes:

- (1) An internal fracture flag is set for each joint segment where the joint friction, cohesion or tensile strength is exceeded. If a joint is fractured (i.e., fracture flag is set), then the residual friction, cohesion and tensile strength values are used.
- (2) No fluid flow occurs in unfractured joint segments. Fluid flow occurs in fractured segments if the command **SET j5flow on** is given.

Continuously yielding model

- (1) **ares** residual aperture at high stress
- (2) **azero** aperture for zero normal stress
- (3) **empb** empirical multiplier for fluid flow law
- (4) **expa** exponent of joint hydraulic aperture
- (5) **jen** exponent of joint normal stiffness
- (6) **jes** exponent of joint shear stiffness
- (7) **jfriction** joint intrinsic friction angle
- (8) **jif** joint initial friction angle
- (9) **jkn** joint normal stiffness
- (10) **jks** joint shear stiffness
- (11) **jperm** wetting fluid joint permeability
- (12) **jrough** joint roughness parameter
- (13) **maxjkn** maximum value of joint normal stiffness
- (14) **maxjks** maximum value of joint shear stiffness
- (15) **minjkn** minimum value of joint normal stiffness
- (16) **minjks** minimum value of joint shear stiffness
- (17) **nwjperm** non-wetting fluid joint permeability

NOTES:

- (1) If neither **maxjkn** nor **minjkn** is defined, joint normal stiffness will be constant (i.e., **jen** will have no effect). The same condition holds for joint shear stiffness.
- (2) Timestep calculation is based on the values for **maxjkn** and **maxjks**. The user should avoid very large values for **maxjkn** and **maxjks**, as these will result in very small timesteps.

For the **area** and **residual** models, the following conditions apply.

- (1.) A nonlinear normal stress-normal displacement relation may be assigned, by means of a table, with the keyword

nstable **n**

Table **n** (see the **TABLE** command) contains a list of pairs (u_n, σ_n) , in which u_n is normal displacement and σ_n is normal stress, defining the joint stress-displacement behavior in the compressible range. (u_n must be negative or zero, and σ_n must be positive or zero.) If the point (0,0) is not given, it will be added to the table. In the tensile range, constant normal stiffness equal to the first table segment is used. For unloading and reloading, the maximum stiffness attained in the past is used.

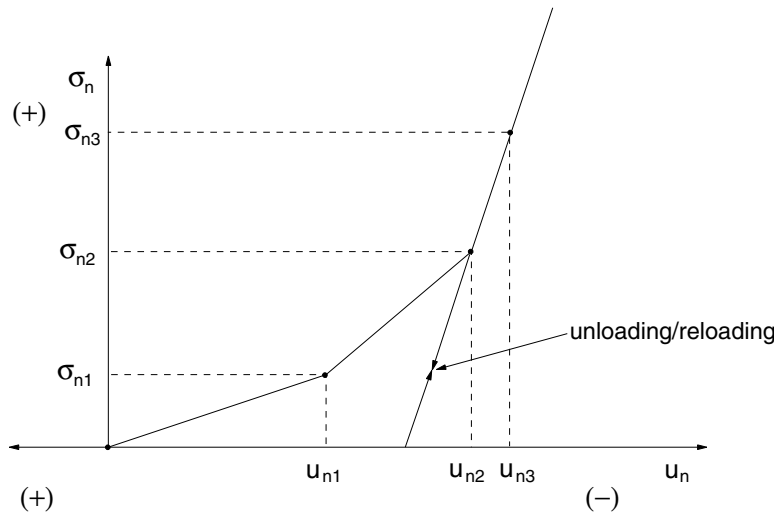


Figure 1.1 **TABLE 1** (0,0) (u_{n1}, σ_{n1}) (u_{n2}, σ_{n2}) (u_{n3}, σ_{n3})

- (2.) Joint dilation is effective when the joint is at slip. The normal displacement increment, Δu_n^d , due to dilation is calculated as

$$\begin{aligned} \Delta u_n^d &= \tan(jdil) \Delta u_s \text{ sign}(u_s) & \text{if } |u_s| < zdil \\ &= 0 & \text{if } |u_s| > zdil \end{aligned}$$

where **zdil** is shear displacement for zero dilation as prescribed by **zdilation**.

Note that dilation is positive if the shear displacement increment (Δu_s) is in the direction of the total shear displacement (u_s), and negative otherwise.

JOINT

If **SET add_dil off** (default), the effective friction is equal to **jfric**. If **SET add_dil on**, the effective friction angle is obtained by adding the dilation angle (**jdil**) to **jfric**, if the dilation is positive, or subtracting it from **jfric** if the dilation is negative.

JREGION

JREGION **id n x1 y1 x2 y2 x3 y3 x4 y4 <delete>**

A convex quadrilateral region is defined for generation of a joint set (or sets) and assigned **id n**. The coordinates of the four corners (in a *clockwise* direction) delimit the boundary of the region. The region boundary must be convex. 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.

The optional keyword **delete** will cause stored joints from previous **JSET** or **CRACK** commands to be deleted within the region. This is useful when generating joints in multiple **JREGION** regions to avoid cuts by joints generated in adjacent regions.

NOTE: **JREGION** may also be used to define quadrilateral regions to be used in the range phrase of any command (e.g., . . . **range jregion n**).

JSET

JSET keyword <**range**. . . >

A joint set is generated in the **range** with characteristics defined by the following keywords (see [Figure 1.2](#) for illustration of parameters).

angle *am* <*ad*>
angle of joint track to the global *x*-axis (default = 0)

gap *gm* <*gd*>
gap length between joint segments (default = 0)

spacing *sm* <*sd*>
spacing normal to joint tracks (default = model size)

trace *tm* <*td*>
trace length of joint segment (default = model size)

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

adev *ad0*
local deviation of angle of all joints (the deviation is around the center of the segment) from the direction given for the global joint track, *a*

id *n*
joint ID number

join Joints will be created as joined or glued joints needed for geometric construction. Contacts that have been joined will be listed with a constitutive model 10.

origin *x0 y0*
coordinates (global axis) of the start of one joint trace. A joint will be generated starting at (*x0*, *y0*); additional joints will be generated to fill the region defined by **JREGION**. (*x0*, *y0*) can be anywhere inside or outside the joint region.

NOTE: If no range is given, the entire model is used for generation. In most cases, the range is defined by keyword **jregion** *n*, where *n* refers to the **id** in a previous **JREGION** command. Other **range** keywords (e.g., **mat** *n*) may also be used to restrict the blocks to be cut (see [Section 1.1.3](#)). Joint segments that do not cut blocks are stored to be used in conjunction with succeeding **JSET** commands. The area of block cutting is controlled by the **range** keywords. The area of storage is not affected by the **range** keywords. Only totally unsuccessful joint segments are stored. The remainder ends

of joint segments that do not cut blocks are not stored. The stored joint segments can be removed by the **JDELETE** command. All incomplete joint segments are discarded automatically before zone generation or cycling. Partial cracks are not modeled in *UDEC* (only completed blocks). A block that wraps completely around another block will not behave correctly. This is because a block cannot form contacts with itself.

The optional ID is provided to allow joints created with this command to be identified. An ID number is automatically assigned to all joints. This allows the user to specify that number.

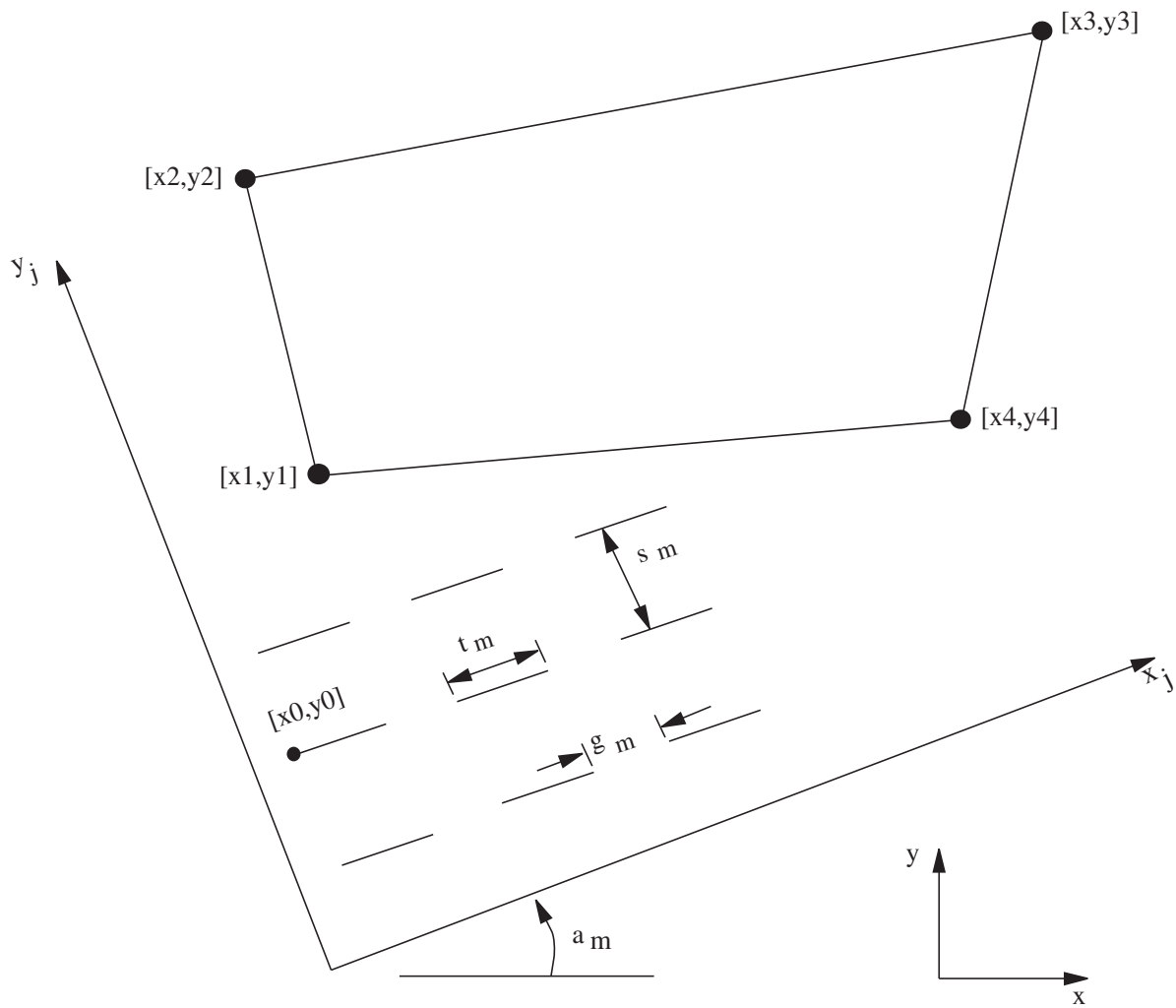


Figure 1.2 Joint set parameters (joint region is defined by JREGION id 1 $x1,y1$ $x2,y2$ $x3,y3$ $x4,y4$)

LABEL

LABEL keyword *value* . . .

This command allows user-defined labels to be added to *UDEEC* plots. The *next* input line following the **LABEL** command for the **arrow**, **history**, **plot** and **table** labels is taken as text for the label. The following labels can be defined.

arrow *id x1 y1 x2 y2*

defines a label with an arrow that may be placed on a plot. The arrow is assigned an *id* number, and extends from position (*x1*, *y1*) to (*x2*, *y2*).

history *nh*

creates a plotting label alias for history number *nh*.

line *id x1 y1 x2 y2*

defines a line that may be placed on a plot. The line is assigned an *id* number, and extends from position (*x1*, *y1*) to (*x2*, *y2*). A label text is not specified.

plot *id x y*

defines a label text that can be added to a plot. The label is assigned an *id* number, and is positioned at (*x*,*y*).

table *nt*

creates a plotting label alias for table number *nt*.

NOTE:

1. All arrow, line and plot labels are added to a plot by specifying the **label** keyword with the **PLOT** command.
2. Individual labels can be added to a plot by specifying the *id* number following the **label** keyword.

The following keyword can be given to delete a label.

delete	keyword <i>id</i>	
	arrow	removes arrow label <i>id</i> .
	history	removes history label <i>id</i> .
	line	removes line label <i>id</i> .
	plot	removes plot label <i>id</i> .
	table	removes table label <i>id</i> .

LINK

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 that will provide the link to the flying block. This block should be the one that 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. This command is not required if **CONFIG cell** is used.

LOAD

LOAD keyword *v* <keyword *v* > ... <**range**...>

All rigid blocks with centroids within the optional **range** are loaded with forces applied at block centroids. (See the **INITIAL** command.)

The following keywords are used to apply loads.

xload *xl*
static load applied in *x*-direction

yload *yl*
static load applied in *y*-direction

NOTE: Time-varying loads can be applied by *FISH* functions.

MOVIE**MOVIE** keyword

This command controls the capture of a sequence of screen plots. The **MOVIE** command controls the creation of a set of PNG images that may be strung together. The images are generated as screen dumps. The size of the movie images can be set by the **size** keyword. The size cannot be larger than the user's screen. Also, the plot window must be entirely on the screen or a truncated image will result. The following keywords are available.

file *fname*

sets the base of the file name. Frames will be numbered incrementally.

off turns off screen-capture mode.

on turns on capturing mode; any screen plot will be saved in the specified (or default) file. If a data file is being processed and **MOVIE** is **on**, all plots will be saved. In interactive mode, the user must use the **MOVIE on** command and press <F2> to save a frame.

size *ix, iy*

specifies the size of the movie image. The default size is 750×563 . This will result in a movie slightly smaller than an 800×600 screen, and will maintain a square aspect ratio in the image. The size specified should not be larger than the actual screen.

step <**off on**> *n*

A new screen plot is saved to the movie file every *n* steps (default *n* = 1000). This allows frames to be saved without interrupting a **CYCLE** or **SOLVE** command. The plot generated is the same as the most recent **PLOT** command issued.

MSCALE

MSCALE keyword

max *rval*

The ratio of maximum-to-minimum mass scaling factor is limited to *rval* for zone mass scaling. By default, the maximum and minimum factors are not limited.

off
on<*zcmass bcmass*>

Mass scaling for all blocks is turned on to speed up convergence of static problems involving very nonuniform blocks or zone meshes, or very nonuniform elastic properties. The scaling is based upon the *average* block mass or zone mass in the model. The average value generally leads to the fastest convergence. The mass used for scaling may be changed via the optional values *zcmass* and *bcmass*. For rigid blocks, the user must supply a value for *bcmass* to change the block mass used for scaling. (A dummy place for *zcmass* is therefore required. For deformable blocks, *zcmass* must be given; *bcmass* need not be supplied.)

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

NOTE: Mass scaling is **on** by default, and is turned on automatically when the **DAMPING local** or **DAMPING auto** command is issued. Mass scaling is turned off automatically when **DAMPING** is invoked without the **local** or **auto** keyword.

part *tdel*

experimental partial density scaling for deformable blocks. This command applies mass scaling only to those zones necessary to achieve timestep *tdel*. This command is useful for dynamic analysis. The number of zones affected by this command can be determined via the **PRINT zone state** command.

NEW

NEW

This command allows the user to begin a new problem without leaving *UDEC*. The model becomes undefined, and problem variables are reset to zero or their default values; the file “UDEC.INI” is consulted again for any start-up commands.

NOTE: When running several different problems from a **CALLed** file, the **NEW** command must be given before each problem.

Please note: The following files are *not* affected by the **NEW** command.

history file (see **HISTORY write**)

log file (see **SET log**)

movie file (see **MOVIE**)

plot file (see **SET output**)

These files remain open if they are open already, and their file names are not changed when a **NEW** command is given. New file names should be specified after the **NEW** command, if required.

The echo mode (**SET echo**) and message mode (**SET message**) are unaffected by **NEW**. These modes can be turned off and on as needed. Also, the random number generator seed (used, for example, in the **JSET** and **VORONOI** commands) is not initialized when **NEW** is given. It is only initialized when *UDEC* is first executed.

All other conditions and values are reset after **NEW** is given. In particular, *FISH* functions and histories will be lost. If certain problem variables are needed for different problems, the problem state can be **SAVEd** and then **RESTORED** when starting a new analysis.

PAUSE**PAUSE** <keyword> <*t*>

This command allows the user to pause reading of a **CALL**ed file. When **PAUSE** is encountered, *UDEC* will stop processing the data at that point and pass control back to the keyboard. Any commands can then be typed (e.g., **PLOT blocks**). When the **CONTINUE** command is typed, *UDEC* will resume reading the data file (unless a **NEW** or **CALL** command has been given).

Two options are available:

- | | |
|-----------------|--|
| key | <i>UDEC</i> will resume reading the data file when any key (except <Esc>) is pressed. The <Esc> key will abort all processing and abort reading the data file. |
| <i>t</i> | <i>UDEC</i> will pause <i>t</i> seconds, or until a key is pressed, and then resume processing. The <Esc> key will abort reading of the data file. |

PFIX

PFIX keyword *value* <keyword *value*> ... <**range**. . . >

This command allows fluid pressure to be specified within domains (i.e., “open” spaces between blocks) within the optional range. If the range is omitted, the command applies to all domains except the outer domain. Note that the specified fluid pressures do *not* change unless a different **PFIX** or **PFREE** command is executed, or new contacts are formed or deleted. To avoid forming or deleting contacts, use larger rounding lengths.

Also note that when domain pressures are changed, there will be an imbalance between block (total) stresses and joint (effective) stresses. Some cycling may be necessary to bring the model to equilibrium. **PFIX** does not change fluid pressure stored in zones (use **INSITU**).

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

density	ρ fluid density to be used with PFIX table
nonwetting	denotes that pressures are fixed in non-wetting fluid. If the pressure is also set using the nwpp , nwpgrad and nwpgrad keywords, then the nonwetting keyword is not necessary.
pp	pw wetting fluid pressure
pxgrad	dwx wetting fluid pressure gradient in the x -direction
pygrad	dwy wetting fluid pressure gradient in the y -direction
table	n table n defines the surface of the water table. The command provides fluid pressures corresponding to a static head below the water table surface. The fluid density, ρ , and gravity must be specified prior to the use of this command.
wetting	denotes that pressures are fixed in wetting fluid. If the pressure is also set using the pp , pxgrad and pygrad keywords, then the wetting keyword is not necessary.

Total wetting fluid pressure, **ppw**, in a domain is determined:

$$ppw = pw + dwx \cdot x + dwy \cdot y$$

PREFIX

nonwetting

nonwetting frees pressures in wetting flows.

nwpp *pnw*

non-wetting fluid pressure

nwp_xgrad *dnw_x*

non-wetting fluid pressure gradient in the *x*-direction

nwp_ygrad *dnw_y*

non-wetting fluid pressure gradient in the *y*-direction

Total non-wetting fluid pressure, *ppw*, in a domain is determined:

$$ppnw = pnw + dnw_x \cdot x + dnw_y \cdot y$$

NOTE: Positive pore pressures cause the domain to expand.

A time history multiplier can be assigned by the keyword

history keyword

The following keywords apply.

cosine *freq time*

cosine wave load with frequency *freq* (cycles/sec) applied for a problem time period of *time*

n History time record *n* (previously read into *UDEC* with the **BOUNDARY hread** command) is applied.

name The history multiplier is a user-defined *FISH* function, where *name* is the function name.

sine *freq time*

sine wave load with frequency *freq* (cycles/sec) applied for a problem time period of *time*

table *n*

history is input using the **TABLE** command. **TABLE n** consists of a list of pairs: *time*, *f(time)*. Linear interpolation is performed between the given discrete points.

PFIX **seepage**

seepage The seepage condition is prescribed on the boundaries of the domain. If a seepage condition is assigned to a domain, then (1) wetting and non-wetting fluid pressures are fixed (wetting fluid pressure is equal to non-wetting fluid pressure); and (2) wetting fluid can inflow into the domain only from saturated domains.

Use of the **PFIX** command is illustrated in [Example 1.1](#):

Example 1.1 Use of the PFIX command

```
pfix pressure=10 range domain 521
pfix pressure=10
pfix pressure=10 xgrad=1 range 0 10 25 50
pfix pressure=15 range region 0,0 0,10 10,10 10,0
pfix pressure=10 hist table 1 range domain 251
```

PFREE

PFREE keyword <**range**. . . >

The pressure is not controlled in the domains within the **range**. If the range is omitted, the command applies to all domains except the outer domain.

nonwetting denotes that pressures in non-wetting fluids are not controlled.

wetting denotes that pressures in wetting fluid are not controlled. (This option is the default.)

Two things will cause domain pressures to change:

- (1) fluid flow; or
- (2) domain volume changes (if **SET flow off** is specified).

In both cases, the fluid bulk modulus must be nonzero.

PLOT

PLOT <keyword> <switch <*value*> ... > <keyword ... > ...

This command requests that a plot be made on the screen, or directed to a hardcopy plotting device or a file. Note that the **WINDOW** command may be used to change the plotting area. Several variables may be plotted as overlays by giving several keywords on one line. The <Esc> key will terminate any plotting in progress. Note that if no keyword is given, only the blocks will be plotted (by default), or the previously issued **PLOT** command will be repeated.

Keywords for the **PLOT** command are grouped into five categories:

- (1) general plotting keywords;
- (2) switches to enhance or modify plots;
- (3) color switches;
- (4) the profile (line) plot switch; and
- (5) interactive (screen mode) plotting options.

Switch keywords modify (or enhance) the general plotting keyword; they are optional, but they *must* follow the keyword that is to be modified, unless specifically stated otherwise. The general plotting keywords and their switches are summarized in [Table 1.5](#). The keywords and interactive plotting are described separately for each category.

Table 1.5 Switches to modify plotting keywords

	color	sc	absolute	alias	back	bfill	diam	filcolor	fill	grid	hold	id	interval	inverse	label	magnify	max	min	nc	ndfix	nohead	noopen	noscale	number	pen	project	sampld	scale	thick	yrev	zero
aperture	x	x		x	x			x			x		x				x	x			x	x	x		x				x		
axial	X			X	X						X										X		X		X			X			
be	X			X	X						X										X		X		X						
blocks	X			X	X						X	X				X			X	X	X		X	X	X						
boundary	X			X	X						X										X		X	X	X						
c_extra	X	X		X	X		X				X		X				X	X			X	X	X	X	X						
cable	X			X	X						X						X				X		X	X	X			X		X	
ccdif		X		X			X				X						X	X			X		X	X	X						
ccfail		X		X			X				X						X	X			X		X	X	X						
ccmean		X		X			X				X						X	X			X		X	X	X						
ccs1		X		X			X				X						X	X			X		X	X	X						
ccs2		X		X			X				X						X	X			X		X	X	X						
cforce	X	X		X	X						X						X	X			X		X	X	X			X			
closure	X	X		X	X			X			X		X				X	X			X	X	X	X	X			X			
contacts	X			X	X							X	X								X		X	X	X						
disp	X	X		X	X			X			X										X		X	X	X			X			
dmagnitude	X			X	X			X	X	X	X		X	X	X		X	X			X		X	X	X						X
dnum	X			X	X						X										X		X	X	X						
dshe	X	X		X	X			X			X		X				X	X			X	X	X	X	X			X			
evol				X	X	X		X	X	X	X		X	X	X		X	X			X		X	X	X	X	X	X			X
exx				X	X	X		X	X	X	X		X	X	X		X	X			X		X	X	X	X	X	X			X
exy				X	X	X		X	X	X	X		X	X	X		X	X			X		X	X	X	X	X	X			X
eyy				X	X	X		X	X	X	X		X	X	X		X	X			X		X	X	X	X	X	X			X
fbcont				X	X			X	X	X	X		X	X	X		X	X			X		X		X						X
fbflow				X	X			X			X										X		X	X	X			X			
fboundary				X	X						X										X		X	X	X						
fdil		X		X							X										X		X	X	X						
flow	X	X		X	X			X			X		X				X	X			X	X	X	X	X			X			
fractures	X	X		X	X						X										X		X	X	X						
fstrain		X		X							X										X		X	X	X						
fvel		X		X	X			X			X		X				X	X			X	X	X	X	X			X			
gp_extra				X	X			X	X	X	X		X	X	X		X	X			X		X	X	X						X
gpforces	X	X		X	X			X			X										X		X	X	X			X			
group				X				X			X										X		X	X	X						
hist			X								X										X				X						
hoek	X			X	X			X	X	X	X		X	X	X		X	X			X		X	X	X						X
idil		X		X							X										X		X	X	X						
istrain		X		X							X										X		X	X	X						
j_nstress	X	X		X	X			X			X		X				X	X			X	X	X	X	X			X			
j_sstress	X	X		X	X			X			X		X				X	X			X	X	X	X	X			X			
jcons				X	X						X										X		X	X	X						
jline				X							X										X		X	X	X						
joined	X			X	X						X										X		X	X	X						
joint				X							X										X		X	X	X						
jprop				X				X			X										X		X	X	X						
label	X										X										X				X						
map				X	X						X		X				X	X			X	X	X	X	X						
mat				X							X		X				X	X			X	X	X	X	X						
model				X				X			X										X		X	X	X						
mohr	X			X	X			X	X	X	X		X	X	X		X	X			X		X	X	X						X
moment_thrust	X			X							X										X			X	X						
-nwpp	X			X	X			X			X		X				X	X			X	X	X	X	X			X			
nwpp	X			X				X			X		X				X	X			X	X	X	X	X			X			
open	X			X	X						X		X		X		X	X			X		X	X	X						

Table 1.5 Switches to modify plotting keywords (continued)

	color	sc	absolute	alias	back	bfill	diam	fillcolor	fill	grid	hold	id	interval	inverse	label	magnify	max	min	nc	ndfix	nohead	noopen	noscale	number	pen	project	sampld	scale	thick	yrev	zero
overlap	X			X	X						X										X		X		X						
plas				X							X										X		X		X						
pline	X			X	X						X										X		X		X						
-pp	X			X	X			X			X										X	X	X		X				X		
pp	X	X		X	X						X		X				X	X			X	X	X		X						
property				X				X			X										X		X		X						
reinforce	X			X	X						X										X		X	X	X						
rockbolt	X			X	X						X						X				X		X	X	X			X			
rotate	X			X	X						X										X		X		X						
rshear	X	X		X							X				X		X	X			X		X		X						
saturation	X	X		X							X		X		X		X	X			X		X		X				X		
sdif	X		X	X	X	X		X	X	X	X		X	X	X		X	X			X		X		X	X	X				X
separ	X	X		X	X						X		X				X	X			X	X	X		X					X	
shear	X	X		X	X						X		X				X	X			X	X	X		X				X		
sig1	X			X	X	X		X	X	X	X		X	X	X		X	X			X		X		X	X	X				X
sig2	X			X	X	X		X	X	X	X		X	X	X		X	X			X		X		X	X	X				X
slip	X			X	X						X										X		X		X						
smax	X			X	X	X		X	X	X	X		X	X	X		X	X			X		X		X	X	X				X
smin	X			X	X	X		X	X	X	X		X	X	X		X	X			X		X		X	X	X				X
ssi				X	X			X	X	X	X		X	X	X		X	X			X		X		X						X
ssr				X	X			X	X	X	X		X	X	X		X	X			X		X		X						X
stcon	X			X	X						X										X		X		X						
stresses	X	X		X	X		X				X										X		X		X						
struct	X			X	X						X						X				X		X	X			X				
support	X			X	X						X										X		X		X						
sxx				X	X	X		X	X	X	X		X	X	X		X	X			X		X		X	X	X				X
sxy				X	X	X		X	X	X	X		X	X	X		X	X			X		X		X	X	X				X
syy				X	X	X		X	X	X	X		X	X	X		X	X			X		X		X	X	X				X
szz				X	X	X		X	X	X	X		X	X	X		X	X			X		X		X	X	X				X
table			X								X										X				X						
tcont				X	X			X	X	X	X		X	X	X		X	X			X		X		X						X
temp	X			X	X			X	X	X	X		X	X	X		X	X			X		X		X						X
tfix	X			X	X						X										X		X		X						
thrust_shear	X			X	X						X										X				X						
ubiquitous	X			X	X						X										X		X		X			X			
unbalanced	X		X	X							X										X				X						
vel	X	X		X	X						X										X		X		X			X			
vflow	X	X		X	X						X		X								X		X		X			X			
vmagnitude	X			X	X			X	X	X	X		X	X	X		X	X			X		X		X						X
window	X			X							X										X				X						
xdis				X	X			X	X	X	X		X	X	X		X	X			X		X		X						X
xvel				X	X			X	X	X	X		X	X	X		X	X			X		X		X						X
ydis				X	X			X	X	X	X		X	X	X		X	X			X		X		X						X
yvel				X	X			X	X	X	X		X	X	X		X	X			X		X		X						X
z_extra				X	X			X	X	X	X		X	X	X		X	X			X		X		X						X
zone_pp	X			X				X	X		X		X		X		X	X			X		X		X						
zones	X			X	X						X	X									X		X	X	X						

1. General Plotting Keywords

aperture	joint hydraulic aperture
	Joint aperture is plotted as a line colored by magnitude. The maximum and minimum switches may be used to control the color scaling, and limit the range of magnitudes plotted. Magnitudes greater than the maximum and lower than the minimum are not plotted. If the thick keyword is used, the thickness of the line plotted is proportional to the joint hydraulic aperture. The maximum magnitude is automatically scaled to be five line thicknesses. The interval switch may be used to change the magnitude represented by each line thickness. The noopen switch may be used to omit plotting values from contacts that are “open.” Contacts that have zero normal force are defined as open.
axial	axial forces in reinforcement
be	boundary elements
blocks	assemblages of blocks; only visible blocks are plotted (see HIDE and SHOW)
boundary	<keyword> outer boundary (can only be plotted after execution of the BOUNDARY command). Boundaries are also plotted for voids inside the <i>UDEC</i> model. The void volume must be 50 times the minimum domain volume. (Also see SET dami .) The following keywords may be used to modify boundary plots to indicate boundary conditions. bvelocity plots boundary velocities fluid fluid-flow boundary conditions force applied boundary forces nwfluid non-wetting fluid-flow boundary condition reaction boundary reaction forces thermal thermal boundary conditions xcondition boundary conditions applied in the <i>x</i> -direction ycondition boundary conditions applied in the <i>y</i> -direction
c_extra	plots lines colored by values stored by the user in the contact data array. (See Section 1 in the <i>FISH</i> volume.)

PLOT **cable**

cable <keyword> <*n*>

plots location of cable elements. For most of the keywords (shown below), the identification number *n* for the cable (defined by the **CABLE** command) may be used to plot only the cable elements associated with the number. The corresponding value of *n* for the cable may be found by using the **PLOT cable number** command. The following keywords may be used.

afail cable element axial failure

axial <*n*> axial force

element cable element numbers

fail shows failure modes in color.

gfail cable node grout failure

node cable node numbers

number the identification number of the cable group used (for example, for line plots)

sdisp nodal point displacement vectors

shear <*n*> relative shear force between cable node and host material

strain <*n*> axial strain in cable elements

svel nodal point velocity vectors

xdisp <*n*> x-displacement of cable nodes

xvel <*n*> x-velocity of cable nodes

ydisp <*n*> y-displacement of cable nodes

yvel <*n*> y-velocity of cable nodes

NOTE:

- (1) The **line** plot switch can be used to plot a cable variable as a line plot (see plot category 4).
- (2) The sense of the cable variable plots can be reversed by giving the switch **yrev** after the cable variable keyword.

PLOT**ccdif**

ccdif Color-coded principal stress tensors (similar to the **PLOT stresses** command except that the colors of the plotted vectors are scaled based on the magnitude of the differential stress in the zone). This command works only for deformable blocks.

ccfail Color-coded principal stress tensors (similar to the **PLOT stresses** command except that the colors of the plotted vectors are scaled based on the fluid pressure increase required to cause failure). This plot requires that the properties of friction angle, cohesion and tensile strength be defined via the **PROPERTY** command for the blocks being plotted. This must be done even if the constitutive model is not Mohr-Coulomb. This command works only for deformable blocks.

ccmean Color-coded principal stress tensors (similar to the **PLOT stresses** command except that the colors of the plotted vectors are scaled based on the magnitude of the mean stress in the zone). This command works only for deformable blocks.

ccs1 Color-coded principal stress vectors (similar to the **PLOT stresses** command except that the colors of the plotted vectors are scaled based on the most-compressive principal stress). This command works only for deformable blocks.

ccs2 Color-coded principal stress vectors (similar to the **PLOT stresses** command except that the colors of the plotted vectors are scaled based on the least compressive principal stress). This command works only for deformable blocks.

cforce axial force vector along cable bolt elements

closure joint closure

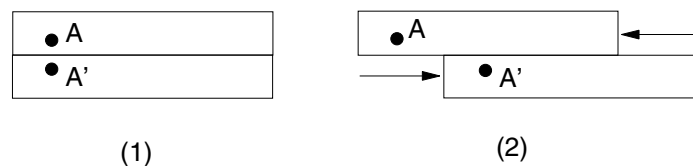
Joint closures (negative normal displacements) are plotted as a line colored by magnitude. The **maximum** and **minimum** switches may be used to control the color scaling, and to limit the range of magnitudes plotted. Magnitudes greater than the **maximum** and lower than the **minimum** are not plotted. If the **thick** keyword is used, the thickness of the line plotted is proportional to the closure of the joints. The **maximum** magnitude is automatically scaled to be five line thicknesses. The **interval** switch may be used to change the magnitude represented by each line thickness. The **noopen** switch may be used to omit plotting values from contacts that are “open.” Contacts that have zero normal force are defined as open.

contacts normal to contacts (colors: green – corner/corner contact, cyan – corner/edge contact, red – edge/corner contact)

disp gridpoint displacement vectors for deformable blocks and corner displacement vectors for rigid blocks.

PLOT **dmagnitude****dmagnitude** contour of displacement magnitude**dnum** domain numbers**dshe** directions of shear displacement. The terms “left-lateral” and “right-lateral” are used to describe the sense of relative shear displacements, as explained below.

Imagine two points, A and A', originally next to each other, before separation (1). After displacement, they are separated (2). Left-lateral displacements mean that, when crossing the separation plane, the corresponding point is to the left of its original position.

**Figure 1.3** *Definition for direction of shear displacement***evol** contours of volumetric strain in deformable block zones.***exx** contours of *x*-component of strain in deformable block zones.***exy** contours of shear strain in deformable block zones.***eyy** contours of *y*-component of strain in deformable block zones.***fbcont** fluid pressure contours in fluid boundary grid****fbflow** fluid boundary flow vectors****fboundary** fluid boundary grid****fdil** principal finite strains (similar to the **PLOT fstrain** command with the exception that the colors of the plotted tensors are scaled based on the magnitude of dilation in the zone). This command works only for deformable blocks.

* The strains are calculated using the current displacements of the zone gridpoints and the current zone volume. Note that the strains resulting from initializing the stresses in a zone using the **INSITU stress** command are not included in the calculation. The use of the **RESET disp** command will affect the calculated values.

** Use the **WINDOW** command to enlarge the viewing window for the fluid boundary grid.

PLOT flow

flow fluid flow quantity in joints

Joint fluid flow volume is plotted as a line colored by magnitude. The **maximum** and **minimum** switches may be used to control the color scaling, and limit the range of magnitudes plotted. Magnitudes greater than the **maximum** and lower than the **minimum** are not plotted. If the **thick** keyword is used, the thickness of the line plotted is proportional to the closure of the joints. The **maximum** magnitude is automatically scaled to be five line thicknesses. The **interval** switch may be used to change the magnitude represented by each line thickness. The **noopen** switch may be used to omit plotting values from contacts that are “open.” Contacts that have zero normal force are defined as open.

fractures joints with peak/residual strength that have fracture flag set. (See **CHANGE jcons=5** or **JOINT model residual**.)

fstrain principal finite strain tensors plotted in a manner similar to that in the **PLOT stresses** command. The finite strains are calculated using the current gridpoint displacements and the current zone volume. Note that the strains resulting from initializing the stresses in a zone using the **INSITU stress** command are not included in the calculation. The use of the **RESET disp** command will affect the calculated values. This command works only for deformable blocks.

fvel fluid velocities in joints (fluid flow volume divided by aperture)

Joint fluid flow velocity is plotted as a line colored by magnitude. The **maximum** and **minimum** switches may be used to control the color scaling, and to limit the range of magnitudes plotted. Magnitudes greater than the **maximum** and lower than the **minimum** are not plotted. If the **thick** keyword is used, the thickness of the line plotted is proportional to the closure of the joints. The **maximum** magnitude is automatically scaled to be five line thicknesses. The **interval** switch may be used to change the magnitude represented by each line thickness. The **noopen** switch may be used to omit plotting values from contacts that are “open.” Contacts that have zero normal force are defined as open.

gp_extra plots contours of values stored by user in gridpoint data structure (see [Section 1](#) in the **FISH** volume).

gpforces gridpoint force vectors for deformable blocks

PLOT **group****group** <keyword>

plots zones, blocks or contacts colored by group. The following keywords are available.

block block groups names**joint** joint groups names**zone** zone groups names**hist** <*n* ... > <keyword ... >

A plot of the time history of the variable recorded in history number *n* is made. Multiple histories can be plotted by giving several numbers in sequence. A minus sign in front of the history number *n* reverses the sign of the history plotted. Use **PRINT history** to identify history numbers.

This keyword can also be used to show (e.g., on a block plot) where histories are recorded. The command **PLOT hist** (without a history number) will show the location of all histories. See the **LABEL** command for instruction on defining a label for **HISTORY** plots.

Optional keywords allow the user to manipulate the history:

begin *ncb*

Histories will be plotted beginning with cycle number *ncb*.

both both a line and “x” symbols are used to plot the history**cross** the history points are plotted as “x” symbols**cycle** Use cycle count for *x*-axis (default is time).**end** *nce*

Histories will be plotted ending with cycle number *nce*.

line The history is plotted as a line. This is the default style, but if **line** is explicitly requested, a sample line will also be included on the legend.

skip *nc*

Only one point for every *nc* history points recorded is plotted. For example, **skip 10** means that every 10th recorded point (starting with the first) is plotted.

PLOT	hist	vs
		<p>vs</p> <p><i>n</i> time cycle</p> <p>enables plotting of one or more histories versus another. For example,</p> <pre>plot this 2 3 vs 4</pre> <p>plots histories 2 and 3 versus history 4. History 4 plots along the abscissa. (The default for the abscissa is problem time.) The time and cycle keywords may be used to set the abscissa to mechanical time or cycle number (default is cycle number).</p> <p>xrev reverses sign of history plotted to abscissa axis.</p> <p>xwindow <i>xl xu</i></p> <p>specifies range for abscissa axis.</p> <p>yrev reverses sign of history plotted to ordinate axis.</p> <p>ywindow <i>yl yu</i></p> <p>specifies range for ordinate axis.</p>
	hoek	allows the user to plot contours of strength/stress ratios for zones in deformable blocks based upon a Hoek-Brown (H-B) failure criterion (see Section 3.9 in the User's Guide). The H-B strength parameters (unconfined compressive strength, <i>m</i> and <i>s</i>) and tensile strength must be previously defined with the PROPERTY command and ucs , hbm , hbs and tension keywords.
	idil	principal infinitesimal strains (similar to PLOT istrain command, except that the colors of the plotted tensors are scaled based on the magnitude of dilation in the zone). This command works only for deformable blocks.
	istrain	principal infinitesimal strain tensors plotted in a manner similar to that in the PLOT stresses command. The infinitesimal strains are calculated using the gridpoint velocities and the current zone volume. Note that the strains resulting from initializing the stresses in a zone using the INSITU stress command are not included in the calculation. The use of the RESET disp command will affect the calculated values. This command works only for deformable blocks.

PLOT **j_nstress****j_nstress** joint normal stress

Joint normal stress is plotted as a line colored by magnitude. The **maximum** and **minimum** switches may be used to control the color scaling, and to limit the range of magnitudes plotted. Magnitudes greater than the **maximum** and lower than the **minimum** are not plotted. If the **thick** keyword is used, the thickness of the line plotted is proportional to the closure of the joints. The **maximum** magnitude is automatically scaled to be five line thicknesses. The **interval** switch may be used to change the magnitude represented by each line thickness.

j_sstress joint shear stress

Joint shear stress is plotted as a line colored by magnitude. The **maximum** and **minimum** switches may be used to control the color scaling, and to limit the range of magnitudes plotted. Magnitudes greater than the **maximum** and lower than the **minimum** are not plotted. If the **thick** keyword is used, the thickness of the line plotted is proportional to the closure of the joints. The **maximum** magnitude is automatically scaled to be five line thicknesses. The **interval** switch may be used to change the magnitude represented by each line thickness.

jcons joints plotted with color representing the assigned joint constitutive model.**jline** *x1 y1 x2 y2* keyword

A linear variation of a joint parameter is plotted, for all joints that are intersected by the line from (*x1,y1*) to (*x2,y2*) at the location where the line intersects the joint. The following keywords are used to define a joint parameter.

ape	aperture
flow	flow rate
ndisp	normal displacement
nstr	normal stress
pp	pore pressure
sdisp	shear displacement
sstr	shear stress
vel	fluid flow velocity

joined plots edges of blocks that have been joined (i.e., construction joints)

PLOT joint

joint *x1,y1 x2,y2* keyword *<tolc>*

A linear variation of the joint parameter is plotted for a single joint along the line from (*x1,y1*) to (*x2,y2*) within a band \pm *tolc* perpendicular to the line (*x1,y1*), (*x2,y2*). The default value for *tolc* is twice the rounding length. The following keywords are used to define a joint parameter.

ape	aperture
flow	flow rate
ndisp	normal displacement
nstr	normal stress
pp	pore pressure
sdisp	shear displacement
ssir	shear stress
vel	fluid flow velocity

jprop name
plots values of joint properties.

label *<id1 ... idn>*
places arrow, plot and line labels defined with the **LABEL** command on the current plot. The ID numbers may be specified to limit which items are included.

mape joint mechanical aperture

Joint mechanical aperture is plotted as a line colored by magnitude. For non-Barton-Bandis joints, this plot will be the same as that for the hydraulic aperture (**PLOT aperture**). The **maximum** and **minimum** switches may be used to control the color scaling, and to limit the range of magnitudes plotted. Magnitudes greater than the **maximum** and lower than the **minimum** are not plotted. If the **thick** keyword is used, the thickness of the line plotted is proportional to the joint hydraulic aperture. The **maximum** magnitude is automatically scaled to be five line thicknesses. The **interval** switch may be used to change the magnitude represented by each line thickness. The **noopen** switch may be used to omit plotting values from contacts that are “open.” Contacts that have zero normal force are defined as open.

PLOT **mat****mat** <keyword>

material property number assigned to blocks, joints, cables and structural elements (designated by color). The following keywords may be used to plot block, joint, cable, rockbolt and structural element property numbers separately.

block block material numbers**cable** cable material numbers**joint** joint material numbers**reinf** local reinforcement material numbers**rockbolt** rockbolt element material number**struct** structural element material numbers**model** creates a filled plot of the various mechanical material models in the blocks (screen or PostScript plot only).**mohr** allows the user to plot contours of strength/stress ratios for zones in deformable blocks based on a Mohr-Coulomb (M-C) failure criterion (see [Section 3.9](#) in the **User's Guide**). The M-C strength parameters (i.e., cohesion, friction coefficient and tensile strength) must be previously defined with the **PROPERTY** command and **cohesion**, **friction** and **tension** keywords.**moment_thrust** <**smat** = *n*>

generates a moment-thrust yield envelope, and plots the current status of the structural elements in the model. If no **smat** is specified, *UDEC* uses the material properties of the last structural element defined to construct the yield envelope, and then includes the status for all structural elements. If **smat** is specified, then material **smat** is used to construct the yield envelope, and only structural elements with that material number are included.

nwpp non-wetting fluid domain pore pressure

The non-wetting fluid domain pore pressure is plotted as a line colored by magnitude. This plot is only meaningful in the case of two-phase flow. The **maximum** and **minimum** switches may be used to control the color scaling, and limit the range of magnitudes plotted. Magnitudes greater than the **maximum** and lower than the **minimum** are not plotted. If the **thick** keyword is used, the thickness of the line plotted is proportional to the closure of the joints. The **maximum** magnitude is automatically scaled to be five line thicknesses. The **interval** switch

may be used to change the magnitude represented by each line thickness. The **noopen** switch may be used to omit plotting values from contacts that are “open.” Contacts that have zero normal force are defined as open.

-nwpp

negative non-wetting fluid domain pore pressure

The negative non-wetting fluid domain pore pressure is plotted as a line colored by magnitude. This plot is only meaningful in the case of two-phase flow. The **maximum** and **minimum** switches may be used to control the color scaling, and to limit the range of magnitudes plotted. Magnitudes greater than the **maximum** and lower than the **minimum** are not plotted. If the **thick** keyword is used, the thickness of the line plotted is proportional to the closure of the joints. The **maximum** magnitude is automatically scaled to be five line thicknesses. The **interval** switch may be used to change the magnitude represented by each line thickness. The **noopen** switch may be used to omit plotting values from contacts that are “open.” Contacts that have zero normal force are defined as open.

open

joints with zero normal force or stress.

overlap

blocks that are identified during cycling as having “contact overlap too great”

plas

the plastic state (i.e., elastic, at yield, yield in past or tensile failure) of zones in deformable blocks

pline

<*n*> <keyword> <line>

A profile or line plot of a selected zone or gridpoint variable of deformable blocks may be plotted versus distance along a line. The line is defined by the **SET pline** command. The line is identified by its line number, *n*. If *n* is omitted, all lines will be plotted. If no keyword is specified, only the line location is plotted. The lines are oriented within the model window. If the optional keyword **line** is used, the line plot is made separately, with the distance axis horizontal and the variable axis vertical. A line number, *n*, *must* be specified when the **line** keyword is used.

The following keywords are available to plot selected variables.

disp

gridpoint displacement

nstress

magnitude of zone stress normal to the line plot

sig1

zone major principal stress

sig2

zone minor principal stress

PLOT **pline** **sstress**

sstress magnitude of zone shear stress in direction parallel to the line plot

sxx zone *xx*-stress component

sxy zone *xy*-stress component

syy zone *yy*-stress component

szz zone *zz*-stress component

velocity magnitude of gridpoint velocity

xdisp gridpoint *x*-displacement

xvelocity gridpoint *x*-velocity

ydisp gridpoint *y*-displacement

yvelocity gridpoint *y*-velocity

NOTE: No interpolation is made – the value of the nearest data point to the search point on the line is plotted.

pp wetting fluid domain pore pressure in domains (see **PLOT zone_pp** for zone pore pressure)

The wetting fluid domain pore pressure is plotted as a line colored by magnitude. The **maximum** and **minimum** switches may be used to control the color scaling, and to limit the range of magnitudes plotted. Magnitudes greater than the **maximum** and lower than the **minimum** are not plotted. If the **thick** keyword is used, the thickness of the line plotted is proportional to the closure of the joints. The **maximum** magnitude is automatically scaled to be five line thicknesses. The **interval** switch may be used to change the magnitude represented by each line thickness. The **noopen** switch may be used to omit plotting values from contacts that are “open.” Contacts that have zero normal force are defined as open.

-pp negative wetting fluid domain pore pressure

The negative wetting fluid domain pore pressure is plotted as a line colored by magnitude. The **maximum** and **minimum** switches may be used to control the color scaling, and to limit the range of magnitudes plotted. Magnitudes greater than the **maximum** and lower than the **minimum** are not plotted. If the **thick** keyword is used, the thickness of the line plotted is proportional to the closure of the joints. The **maximum** magnitude is automatically scaled to be five line thicknesses. The **interval** switch may be used to change the magnitude represented

by each line thickness. The **noopen** switch may be used to omit plotting values from contacts that are “open.” Contacts that have zero normal force are defined as open.

property

name

creates a filled plot of the named zone material property, defined by the **ZONE** command, or a color line plot of the named joint property defined by the **JOINT** command. (See [Table 1.4](#) for the joint property names and the **ZONE** command for the zone property names.)

reinforce

location of reinforcing elements crossing joints (see the **REINFORCE** command and [Section 1](#) in **Special Features**)

rockbolt

<keyword> <*n*>

plots location of rockbolt elements and associated variables. (See the **STRUCT rockbolt** command and [Section 1](#) in **Special Features**.) For certain keywords (shown below), the identification number, *n*, for the rockbolt (defined by the **STRUCT rockbolt** command) may be used to plot only the rockbolt elements associated with that number. The corresponding value of *n* for the rockbolt may be found by using the **PLOT rockbolt number** command. The following keywords may be used.

afail rockbolt element axial failure

avel <*n*> angular velocity of element

axial <*n*> axial force

ifail interface failure

inormal <*n*> normal force at rockbolt interface

interface rockbolt interface location

ishear <*n*> shear force at rockbolt interface

moment <*n*> moment

number ID number of the element group used (for example, for line plots)

sdisp nodal point displacement vectors

shear <*n*> shear force

svel velocity vectors of rockbolt nodes

xdisp <*n*> x-displacement of rockbolt nodes

PLOT **rockbolt** **xvel** <*n*>
xvel <*n*> *x*-velocity of rockbolt nodes**ydisp** <*n*> *y*-displacement of rockbolt nodes**yvel** <*n*> *y*-velocity of rockbolt nodes

NOTE:

- (1) The **line** plot switch can be used to plot a rockbolt element variable as a line plot (see plot category 4).
- (2) The sense of rockbolt element variable plots can be reversed by giving the switch **yrev** after the rockbolt element keyword.

rotate **block**
zone

rotation angle (in degrees) of blocks or zones. The rotation “arc” is scaled to a maximum arc of 45° (deformable blocks only).

rshear shear force in local reinforcement**saturation** joint saturation

The joint saturation is plotted as a line colored by magnitude. This plot is only meaningful in the case of two-phase flow. The **maximum** and **minimum** switches may be used to control the color scaling, and to limit the range of magnitudes plotted. Magnitudes greater than the **maximum** and lower than the **minimum** are not plotted. If the **thick** keyword is used, the thickness of the line plotted is proportional to the closure of the joints. The **maximum** magnitude is automatically scaled to be five line thicknesses. The **interval** switch may be used to change the magnitude represented by each line thickness. The **noopen** switch may be used to omit plotting values from contacts that are “open.” Contacts that have zero normal force are defined as open.

sdif contours of principal stress difference (i.e., **smax-smin** – deformable blocks only)**separ** joint separation

Joint separation (positive normal displacements) is plotted as a line colored by magnitude. The **maximum** and **minimum** switches may be used to control the color scaling, and to limit the range of magnitudes plotted. Magnitudes greater than the **maximum** and lower than the **minimum** are not plotted. If the **thick** keyword is used, the thickness of the line plotted is proportional to the closure of the joints. The **maximum** magnitude is automatically scaled to be five line thicknesses. The **interval** switch may be used to change the magnitude

PLOT

separ

represented by each line thickness. The **noopen** switch may be used to omit plotting values from contacts that are “open.” Contacts that have zero normal force are defined as open. (NOTE: Only joints with zero tensile normal stress are plotted.)

shear

joint shear displacement

Joint shear displacement is plotted as a line colored by magnitude. The **maximum** and **minimum** switches may be used to control the color scaling, and to limit the range of magnitudes plotted. Magnitudes greater than the **maximum** and lower than the **minimum** are not plotted. If the **thick** keyword is used, the thickness of the line plotted is proportional to the closure of the joints. The **maximum** magnitude is automatically scaled to be five line thicknesses. The **interval** switch may be used to change the magnitude represented by each line thickness. The **noopen** switch may be used to omit plotting values from contacts that are “open.” Contacts that have zero normal force are defined as open.

sig1

contours of major principal stress in the xy -plane (deformable blocks only)

sig2

contours of minor principal stress in the xy -plane (deformable blocks only)

slip

joints that are at limiting friction

smax

contours of maximum principal stress in the xy -plane (deformable blocks only). Because compressive stresses are negative, this is the minor principal stress (**smax** = **sig2**).

smin

contours of minimum principal stress in the xy -plane (deformable blocks only). Because compressive stresses are negative, this is the major principal stress (**smin** = **sig1**).

ssi

contour of maximum shear strain (strains are calculated based on gridpoint displacements)

ssr

contour of maximum shear strain rate

stcon

normals of structural element interface contacts

stresses

principal stress tensors in zones of deformable blocks

PLOT struct

struct <keyword> <*n*>

plots location of structural (beam) elements and associated variables. (See the **STRUCT** command and [Section 1](#) in **Special Features**.) For certain keywords (shown below), the identification number, *n*, for the beam (defined by the **STRUCT** command) may be used to plot only the structural elements associated with that number. The corresponding value of *n* for the beam may be found by using the **PLOT struct number** command. The following keywords may be used.

afail	structural element axial failure
avel < <i>n</i> >	angular velocity of element
axial < <i>n</i> >	axial force
ifail	interface failure
inormal < <i>n</i> >	normal force at structural interface
interface	structural interface location
ishear < <i>n</i> >	shear force at structural interface
moment	moment
node	structural node ID numbers
number	ID number of the element group used (for example, for line plots)
sdisp	nodal point displacement vectors
shear < <i>n</i> >	shear force
svel	velocity vectors of structural nodes
thick	structural element is plotted with thickness rather than a single line
xdisp < <i>n</i> >	x-displacement of structural nodes
xvel < <i>n</i> >	x-velocity of structural nodes
ydisp < <i>n</i> >	y-displacement of structural nodes

PLOT**struct****yvel** <*n*>**yvel** <*n*> y-velocity of structural nodes

NOTE:

- (1) The **line** plot switch can be used to plot a structural element variable as a line plot (see plot category 4).
- (2) The sense of structural element variable plots can be reversed by giving the switch **yrev** after the structural element keyword.

support

structural support elements

sxxcontours of *xx*-stress (deformable blocks only)**sxy**contours of *xy*-stress (deformable blocks only)**syy**contours of *yy*-stress (deformable blocks only)**szz**contours of *zz*-stress (out-of-plane stress) (deformable blocks only)**table***n* ... <keyword ... >

A plot of table number *n* is made (see the **TABLE** command). Multiple tables can be plotted by giving several numbers in sequence. See the **LABEL** command for instructions on defining alternative labeling for table plots. Optional keywords allow the user to manipulate the table:

begin*n*start with data point number *n***both**Both a line and an *x* will be plotted.**cross***xs* will be plotted at table data points.**end***n*end with data point number *n***line**

table will be plotted as a line

skip*n*plot every *n*th data point**xrev**

reverses sign of table plotted to abscissa axis.

xwindow*xl xu*

specifies range for abscissa axis.

yrev

reverses sign of table plotted to ordinate axis.

PLOT	table	ywindow
		ywindow <i>yl yu</i> specifies range for ordinate axis.
tcont	A single contour that separates regions of tension and compression (deformable blocks only) is plotted.	
temp	temperature contours (see Section 3 in Special Features)	
tfix	plots locations where temperature is fixed.	
thrust_shear	<smat = n> generates a thrust-shear yield envelope, and plots the current status of the structural elements in the model. If no smat is specified, <i>UDEC</i> uses the material properties of the last structural element defined to construct the yield envelope, and then includes the status for all structural elements. If smat is specified, then material smat is used to construct the yield envelope, and only structural elements with that material number are included.	
ubiquitous	vectors showing ubiquitous joint angles. This works only for the ubiquitous joint zone model (ZONE model ubiquitous).	
unbalanced	plots graph of unbalanced force history	
vel	block and/or gridpoint velocity vectors	
vflow	flow vectors indicating fluid flow direction	
vmagnitude	contour of velocity magnitude	
window	plots box around plot; used with nohead and SET legend off .	
xdis	contours of gridpoint <i>x</i> -displacement (deformable blocks only)	
xvel	contours of gridpoint <i>x</i> -velocity (deformable blocks only)	
ydis	contours of gridpoint <i>y</i> -displacement (deformable blocks only)	
yvel	contours of gridpoint <i>y</i> -velocity (deformable blocks only)	
z_extra	plots values stored by user in zone data structure (see Section 1 in the FISH volume).	
zone_pp	pore pressures in zones	
zones	zones in deformable blocks	

2. Switches to Modify Plots

If no additional parameters are given, scale factors, a window and a default line color will be chosen automatically. However, any keyword may be followed by any number of “switches,” which are themselves keywords that set certain characteristics of the plot. Each switch operates on the keyword that precedes it. The switches described below can be used to control scaling of the vectors, contour interval and drawing of geometric scales on the plot boundaries. [Table 1.5](#) summarizes the plotting keywords that are affected by these switches.

Note that an enlarged or contracted plot may be made by giving a **WINDOW** command prior to giving the **PLOT** command. The window will then remain set, and will be remembered on restart.

The following keywords may be used to set switches. (Remember that the switch keyword *must* follow the plot keyword that is to be modified, unless specifically stated otherwise.)

sc	scaling factor for scalar and vector quantities. If the parameter sc is specified, the magnitudes are multiplied by sc (i.e., .5 will make vectors half the default size).
absolute	The absolute magnitude of the quantity is plotted.
alias	name changes the name of the variable in the legend. A sequence of words can be given for name if they are contained within single quotes.
back	sets pen color to current background color.
bfill	specifies that stress contours are to be constructed by filling each zone with a color that represents the stress in the zone. For constant strain zones, this is the most precise plot, but it will be blocky in appearance. The default setting is sampled .
diam	plots diamond on principal stress plots if any principal stress in the <i>xy</i> -plane is in tension. Out-of-plane stress (<i>szz</i>) is ignored.
filcolor	n use fill color table number n . Valid numbers are 1-3. See SET filcolor for more information on color tables.
fill	is used to create filled contour plots, as opposed to the default line-contour plots (for PostScript or screen plots). It also fills when used with the mat keyword for deformable blocks.
fos	adds the current fos value to the plot legend. This value is calculated using the SOLVE fos command.

PLOT **grid****grid** *nx ny*

partitions the window into *nx* by *ny* contouring points to be used to determine contours. (The default is *nx* = *ny* = 20.) For higher contour detail, zone sizes should be of the same order of magnitude as the grid squares (see ‘503S1052’ in the **User’s Guide**). This command changes the grid only for the current plot. To change the global values, use **SET sgrid nx ny**.

hold A **PLOT** command issued from a **CALLED** file can be paused for viewing with the **hold** keyword. The **CALL** file will continue when the user presses <ENTER>. The **hold** keyword should immediately follow the **PLOT** command.

id prints ID number on plots; this works for blocks, zones and contacts. Only IDs in contacts are currently set automatically by *UDEC*.

interval *val*

Contours will be plotted at intervals of *val*. The contour values may be written on the screen or to a hardcopy device by issuing the **label** keyword immediately following the **PLOT** command (see the **label** keyword). The maximum limit for contour intervals is 10 for **filled** plots.

This switch is also used to set the interval for the **shear**, **closure**, **separation**, **flow**, **fvel**, **vflow**, **pp** and **aperture** plots.

inverse reverses the grayscale in filled PostScript plots

magnify *<scale>*

If the switch **magnify** is used with **PLOT blocks** for deformable blocks, the deformed blocks are plotted. The distortion of the blocks is magnified by a factor applied to the gridpoint displacements of the deformable blocks. The *scale* value defines the magnification factor. For example, **PLOT blocks magnify 10** plots blocks with gridpoint displacements magnified ten times. If no scale value is given, the factor is based on the maximum displacement within the current window. The magnification factor is displayed in the legend.

max *v*

sets value corresponding to the maximum length of arrow to *v* for vector plots. All other vectors will be scaled to this maximum length. On contour plots, the maximum contour value can be set.

This switch also may be used to limit the range of plotted values in the **shear**, **closure**, **separation**, **flow**, **fvel**, **pp** and **aperture** plots.

PLOT min

min	<p><i>v</i></p> <p>sets the minimum value to be used on contour plots.</p> <p>This switch also may be used to limit the range of plotted values in the shear, closure, separation, flow, fvel, pp and aperture plots.</p>
nc	corner-rounding deleted for plotting blocks. Deleting rounded corners speeds plotting.
nofix	inhibits plotting of fixed blocks (for rigid blocks only). This keyword must precede the blocks keyword.
nohead	removes the border and heading from the plot. This keyword should immediately follow the PLOT command.
noopen	inhibits plotting of open joints when creating the shear , closure , separation , flow , fvel , pp and aperture plots.
noscale	inhibits the plotting of the geometric scales outside of the model.
number	sets flag to plot address numbers for blocks, contacts, zones and reinforcement. This also works to plot material numbers on material plots.
pen	routes the plot according to the SET output and SET plot commands. This keyword must immediately follow the PLOT command. (Also see COPY .)
projected	specifies that stress contour plots will be constructed by projecting the zone stresses to the gridpoints. This allows stresses to be plotted up to the model edge. This is the default.
sampld	specifies that stress contour plots will be constructed using a stress sampling based on the pattern specified in the grid switch. Currently this will produce the smoothest contours, but cannot plot up to the edge of the model for non-orthogonal surfaces. The default setting is projected .
scale	<p><i>v</i></p> <p>sets scale factor for vector plots. In vector plots, the vector magnitude is multiplied by the scale value (i.e., scale .5 will make the vectors half as long). Use the max keyword to set maximum vector value.</p> <p>The usage of the scale value changes if the max and min keywords are used to set the contour range on a colored vector plot (i.e., PLOT ccs1). In this case, the specified value is used as a maximum vector value.</p>

PLOT **sclin**

sclin *x1, y1 x2, y2*

specifies a “scan line,” along which the values of contours are to be displayed. The labels are placed along a line from *x1,y1* to *x2,y2*. This keyword must immediately follow the **PLOT** command.

thick The joint shear, open, closure, pp, aperture, flow, flow velocity, separation, saturation, normal and shear stress plots are no longer automatically scaled by line thickness; they are now scaled by color. To scale by thickness, include the **thick** keyword for these plots.































yrev reverses the sense of the cable axial force plot and structural element axial force and moment plots.

zero suppresses zero contour line.

3. Color Switches

Switches can also be used to change screen color, or fill shade, color or line style on hardcopy plots. (Note that line style applies for PostScript plots only. The line style for Windows printer plots may vary.) The desired line color is typed literally on the command line (for example, **PLOT blocks red**). Note that the background color may be adjusted with the **SET back** command. The following table of keywords may be used to select a color line type for a specified plot keyword.

Table 1.6 *Color switch keywords*

Keyword	Color #	Screen Color	Printer Color	Fill Shade	Line-style
b lack	0	Black	White		
blu e	1	Blue	Blue		
g reen	2	Green	Green		
c yan	3	Cyan	Cyan		
r ed	4	Red	Red		
m agenta	5	Magenta	Magenta		
br own	6	Brown	Orange		
w hite	7	White	Light Gray		
g ray	8	Gray	Dark Gray		
l blue	9	Light Blue	Light Blue		
lg reen	10	Light Green	Light Green		
lc yan	11	Light Cyan	Light Cyan		
lr ed	12	Light Red	Light Red		
lm agenta	13	Light Magenta	Light Magenta		
y ellow	14	Yellow	Yellow		
i white	15	Intense White	Black		

4. Profile (**line**) Plot Switch

A profile or line plot of a cable or beam structural element variable may be plotted versus distance along a line. For the **cable** and **structure** keywords with associated keywords, only the switch keyword **line** need be given. The profile plot will be made versus the distance along the cable or beam, and the number of points will correspond to the number of structural nodes.

The cable or beam number must be specified (type **PLOT cable number** or **PLOT struct number**) with the **line** keyword. The command is **PLOT cable** keyword *n* **line** or **PLOT struct** keyword *n* **line**, where *n* is the cable or beam number. [Table 1.7](#) lists the cable and structural element plotting keywords affected by the **line** switch.

Table 1.7 Keywords affected by the line switch

cable	axial
	shear
	strain
	xdisp
	xvel
	ydisp
	yvel
struct	avel
	axial
	inormal
	ishear
	moment
	shear
	xdisp
	xvel
	ydisp
	yvel

The profile plot switch, **line**, can also be specified with the **PLOT pline** command. Profile plots are made automatically with the **PLOT jline** and **PLOT joint** commands.

5. Interactive Plotting

When the graphics plot of a *UDEC* model is displayed, certain keystrokes may be used to manipulate the model. (It is *not* necessary to follow the keystroke with the <ENTER> key.)

<F2> A PCX output file of the graphics screen will be created. The default file name is "UDEc.PcX." See **SET pcxfile** to change the file name.

<Z> "zoom" (NOTE: This must be an uppercase "Z.")

A cross-hair cursor is activated to modify the viewing window. The cursor is moved by use of the mouse. Press and hold the left mouse button to define one corner of the new viewing window. Drag the mouse to define the diagonally opposite corner of the viewing window. The screen will redraw the current plot using the new window limits. To expand the window again, press <Z><F1><F1>, or double click the left mouse button, or use the **WINDOW** command in the command mode. To inhibit the automatic re-plotting, use the command **SET replot off**.

The zoom keystroke also facilitates making changes to model conditions because **window** is an optional range keyword for the **CHANGE**, **DELETE**, **GENERATE**, **INSITU**, **LOAD**, **PfIX**, **PfREE**, **PRINT** and **ZONE** commands. Therefore, the window defined in the interactive mode will apply as a range for any of the above commands.

PRINT

PRINT keyword <keyword ... > <range... >
FISH symbol name

Printed output is produced for all variables defined by the keywords described below. The printed output for certain keywords (designated by *) can be restricted to an optional range.

If no range is given, then all output for the prescribed keyword will be printed. Printing can be interrupted at any time by pressing the <Esc> key.

UDEC model variables (e.g., blocks, contacts, domains and gridpoints) are identified by address numbers in the “linked-list” data structure. These addresses uniquely define each variable in the model; the addresses are given with the **PRINT** command.

The following keywords may be used.

at *x y*

Model information is printed at location (*x, y*). Data are printed on the block and zone whose centroid is closest to (*x,y*), and on the contact and gridpoint whose locations are closest to (*x,y*). See keywords **blocks**, **zone**, **contact** and **gridpoint** for a description of the output.

bebound <keyword>

induced stress and displacements in boundary elements. The following keywords may be used to separate displacement and stress output.

disp displacement at boundary nodes. The column headings are:

- (1) boundary element node number
- (2) *x*-coordinate of node
- (3) *y*-coordinate of node
- (4) *x*-displacement of node
- (5) *y*-displacement of node
- (6) *x*-direction traction at node
- (7) *y*-direction traction at node

stress stresses at boundary nodes. The column headings are:

- (1) boundary element node number
- (2) *x*-coordinate of node
- (3) *y*-coordinate of node
- (4) tangential stress component in element
- (5) normal stress component in element
- (6) shear stress component in element

beint *n x1 y1 x2 y2* <keyword>

induced stress and displacements at *n* points along line between *x1,y1* and *x2,y2* in boundary element domain. The following keywords may be used to separate displacement and stress output.

disp displacements at interior boundary element domain points. The column headings are:

- (1) interior point number
- (2) *x*-coordinate of point
- (3) *y*-coordinate of point
- (4) *x*-displacement of point
- (5) *y*-displacement of point

stress stresses at interior boundary element domain points. The column headings are:

- (1) interior point number
- (2) *x*-coordinate of point
- (3) *y*-coordinate of point
- (4) *xx*-stress component of point
- (5) *xy*-stress component of point
- (6) *yy*-stress component of point
- (7) major principal stress of point
- (8) minor principal stress of point
- (9) orientation of major principal stress from *x*-axis

blocks* <*n*>

general block data for all block types. The column headings are:

- (1) block address number
- (2) material number for block
- (3) constitutive model number for block (negative for backfill in energy calculation)
- (4) fixity condition (rigid blocks only):

0	free
1	fixed
- (5) block centroid coordinates
- (6) block mass
- (7) block moment of inertia

If the block address number *n* is given, only the data for that block will be printed.

PRINT **boundary*****boundary*** <keyword>

boundary condition information. The following keywords may be used to print the information separately.

be boundary element conditions. The column headings are:

- (1) boundary corner address
- (2) associated block corner address
- (3) associated gridpoint address
- (4) *x*-coordinate of boundary corner
- (5) *y*-coordinate of boundary corner
- (6) associated boundary element node
- (7) second boundary element node
- (8) boundary element load-distribution factor

disp displacement at boundary. The column headings are:

- (1) boundary corner address
- (2) *x*-coordinate of boundary corner
- (3) *y*-coordinate of boundary corner
- (4) *x*-displacement of boundary corner
- (5) *y*-displacement of boundary corner

fluid wetting fluid pressure and permeability condition. The column headings are:

- (1) boundary corner address
- (2) *x*-coordinate of boundary corner
- (3) *y*-coordinate of boundary corner
- (4) permeability conditions at boundary corner:
 0 permeable
 1 impermeable
- (5) pore pressure at boundary corner

PRINT

boundary*

forces

forces

forces at boundary. The column headings are:

- (1) boundary corner address
- (2) x -coordinate of boundary corner
- (3) y -coordinate of boundary corner
- (4) total x -component of force at boundary
- (5) total y -component of force at boundary
- (6) x -component of force added for transient loading
- (7) x -history applied type*
- (8) y -component of force added for transient loading
- (9) y -history applied type*

NOTE: The total forces also include the force induced by displacement boundary conditions.

history

histories* used to apply transient loads. The column headings are:

- (1) history address
- (2) history type*
- (3) current value of history

interior

interior boundary loads. The column headings are:

- (1) boundary corner address
- (2) x -coordinate of boundary corner
- (3) y -coordinate of boundary corner
- (4) total x -component of force at boundary
- (5) total y -component of force at boundary
- (6) x -component of force added for transient loading
- (7) x -history type*
- (8) y -component of force added for transient loading
- (9) y -history type*

* History types are: sine function, cosine function, history record, table and *FISH* function.

PRINT **boundary*** **nwfluid**

nwfluid non-wetting fluid pressure and permeability condition.
The column headings are:

- (1) boundary corner address
- (2) x -coordinate of boundary corner
- (3) y -coordinate of boundary corner
- (4) permeability conditions at boundary corner:
 - 0 permeable
 - 1 impermeable
- (5) pore pressure at boundary corner

State boundary condition state. The column headings are:

- (1) boundary corner address
- (2) associated block corner address
- (3) associated block address
- (4) associated gridpoint address
- (5) x -coordinate of boundary corner
- (6) y -coordinate of boundary corner
- (7) boundary condition in the x -direction
- (8) boundary condition in the y -direction
- (9) free-field boundary condition
- (10) boundary condition in the shear direction
- (11) boundary condition in the normal direction

NOTES:

1. The condition numbers for x -, y -, normal and shear directions are:

- 0 free
- 1 stress (or force)
- 2 boundary element
- 3 viscous
- 4 velocity

2. The free-field indicator numbers are:

- 0 free field turned off
- 1 or 2 free field turned on

PRINT

boundary*

thermal

thermal

temperature at boundary. The column headings are:

- (1) boundary corner address
- (2) *x*-coordinate of boundary corner
- (3) *y*-coordinate of boundary corner
- (4) temperature at boundary corner

vel

velocities* at boundary corners.

- (1) boundary corner address
- (2) *x*-coordinate of boundary corner
- (3) *y*-coordinate of boundary corner
- (4) *x*-velocity of boundary corner
- (5) *x*-history type*
- (6) *y*-velocity of boundary corner
- (7) *y*-history type*

cable

information on cable-reinforcing elements. The output is divided into cable-element information and cable-node information. (See [Section 1](#) in **Special Features** for more information.) The column headings for cable elements are:

- (1) cable element address
- (2) cable element segment identification number
- (3) cable node identification number – first node
- (4) cable node identification number – second node
- (5) material number for cable element
- (6) cross-sectional area of cable element
- (7) axial force in cable element (tension is negative)
- (8) length of cable element
- (9) axial strain in cable element (tension is negative)
- (10) failure indicator for element:
 - 0 elastic
 - 1 at yield in tension
 - 2 elastic, tension yield in past
 - 3 at yield in compression
 - 4 elastic, compression yield in past
 - 5 failure in extensional strain

* History types are: sine function, cosine function, history record, table and *FISH* function.

PRINT cable

The column headings for cable nodes are:

- (1) cable node address
- (2) cable node identification number
- (3) x -coordinate of cable node
- (4) y -coordinate of cable node
- (5) x -velocity of cable node
- (6) y -velocity of cable node
- (7) x -displacement of cable node
- (8) y -displacement of cable node
- (9) material number for cable node
- (10) x -force applied to node
- (11) y -force applied to node
- (12) shear force in grout
- (13) node fixed in x if = 1, else 0
- (14) node fixed in y if = 1, else 0
- (15) failure indicator for node:
 - 0 elastic
 - 1 yield
 - 2 elastic, yield in past
- (16) address of zone in which node is currently located

cave block caving settings. This displays the current settings used in simulation of a block caving mining operation.

- (1) shows if cave material property substitution is on or off
- (2) failed zone percent limit
- (3) new material type number
- (4) new constitutive model number
- (5) stress zeroing state
- (6) sum of number of blocks modified

cell prints list of cells, and the block that each cell contains. Also prints settings associated with cell space contact detection logic.

- (1) number of cells in x - and y -directions
- (2) size of each cell
- (3) origin of cell space
- (4) address of cell space storage

PRINT clos

clos prints the total area enclosed by a structural-element lined tunnel. This area can be used as an indication of tunnel closure. The area must be entirely enclosed. If multiple layers of structural elements are used, each layer will add to the total area. If two layers are used, the area printed will be double the actual area.

contact* <keyword>
 <***n***>

contact data. If the contact address ***n*** is given, all the data for that contact will be printed. The following keywords may be used to print the information separately.

aperture current contact hydraulic apertures and aperture for zero normal stress.

- (1) contact address
- (2) joint material number
- (3) joint constitutive number
- (4) *x*-coordinate of contact
- (5) *y*-coordinate of contact
- (6) hydraulic aperture
- (7) aperture at zero stress (**azero**)
- (8) contact normal displacement

corner corner links. The column headings are:

- (1) contact address
- (2) joint material number
- (3) joint constitutive model number
 (negative if **JOINT** model)
- (4) contact type:
 - 1 corner-corner
 - 2 corner-edge
 - 3 edge-corner
- (5) address of one corner next to contact in one block
- (6) address of one corner next to contact in the other block
- (7) address of corner linked to contact in one block
- (8) address of corner linked to contact in the other block
- (9) interpolation factor for contact at one block
- (10) interpolation factor for contact at the other block

PRINT **contact*** **cy**

cy	data for continuously yielding joint model. The column headings are: (1) contact address (2) joint material number (3) C-Y extension address (4) effective friction angle (5) load reversal factor (6) old plastic shear displacement
disp	contact displacements. The column headings are: (1) contact address (2) <i>x</i> -coordinate of contact (3) <i>y</i> -coordinate of contact (4) relative normal displacement of contact (5) relative shear displacement of contact
force	contact forces/stresses. The column headings are: (1) contact address (2) <i>x</i> -coordinate of contact (3) <i>y</i> -coordinate of contact (4) normal stress at contact (5) shear stress at contact (6) normal force at contact (7) shear force at contact
group	contact group name. The column headings are: (1) contact address (2) <i>x</i> -coordinate (3) <i>y</i> -coordinate (4) group name
hydraulic	hydraulic data. The column headings are: (1) contact address (2) joint material number (negative if JOINT model) (3) joint constitutive model number (4) <i>x</i> -coordinate of contact (5) <i>y</i> -coordinate of contact (6) normal displacement at contact (7) hydraulic extension address

PRINT

contact*

neighbor

neighbor

information on neighboring contacts and vertices. The column headings are:

- (1) contact address
- (2) block 1 address
- (3) block 2 address
- (4) 1st item for block 1
- (5) 2nd item for block 1
- (6) next contact on block
- (7) 1st item for block 2
- (8) 2nd item for block 2
- (9) next contact for block 2

radii

corner radii at contact. The column headings are:

- (1) contact address
- (2) joint material number
(negative if **JOINT** model)
- (3) joint constitutive model number
(negative if **JOINT** model)
- (4) contact type:
 - 1 corner-corner
 - 2 corner-edge
 - 3 edge-corner
- (5) contact overlap
- (6) radius of one corner
- (7) radius of other corner
- (8) distance from contact to one corner center of radius
- (9) distance from contact to other corner center of radius

state

contact state. The column headings are:

- (1) contact address
- (2) joint material number
(negative if **JOINT** model)
- (3) joint constitutive model number
(negative if **JOINT** model)
- (4) x-coordinate of contact
- (5) y-coordinate of contact
- (6) ratio of shear/normal force
- (7) joint length associated with contact

PRINT **contact*** **state**

- (8) joint angle associated with contact
- (9) contact type:
 - 1 corner-corner
 - 2 corner-edge
 - 3 edge-corner
- (10) address of block on one side of contact
- (11) address of block on other side of contact

stiff information on normal stiffness when applied with the **PROPERTY nstable** command. The column headings are:

- (1) contact address
- (2) material number
- (3) stiffness table number
- (4) current joint normal stiffness
- (5) maximum normal closure in past.

stress contact stresses/forces (If the joint length = 0.0, then the contact force is printed; otherwise, the joint stress is printed.) The column headings are:

- (1) contact address
- (2) x-coordinate of contact
- (3) y-coordinate of contact
- (4) normal stress at contact
- (5) shear stress at contact
- (6) normal force at contact
- (7) shear force at contact

corner **<radius>**

corner data. Corner addresses and x-, y-coordinates are listed for each block. If the optional keyword **radius** is given, then information on corner radii is listed. The column headings are:

- (1) corner address
- (2) address of host block
- (3) x-coordinate of corner
- (4) y-coordinate of corner
- (5) corner radius
- (6) address of next corner on block in clockwise direction
- (7) address of next corner on block in counterclockwise direction
- (8) address of gridpoint associated with corner

PRINT

creep**creep**

prints information on creep time settings

dlist

<*n*>

list of addresses of contacts and corners associated with each domain. If *n* is specified, only the list for domain address *n* will be printed.

domain*

<*n*>

domain data. If domain address *n* is specified, only the data for that domain will be printed.

nonwetting

Non-wetting fluid information will be printed. The column headings are:

- (1) domain address
- (2) domain pressure code:
 - 0 pressure not controlled
 - 1 pressure controlled
- (3) pressure history type
- (4) domain pressure
- (5) domain area (volume)
- (6) flow into (positive) or out of (negative) domain
- (7) fictitious domain displacement or, if fluid flow is active, domain area (double precision)
- (8) *x*-coordinate of domain center
- (9) *y*-coordinate of domain center

wetting

Wetting fluid information will be printed. (This is the default option.) The column headings are:

- (1) domain address
- (2) domain pressure code:
 - 0 pressure not controlled
 - 1 pressure controlled
 - 2 seepage condition
- (3) pressure history type
- (4) domain pressure
- (5) domain area (volume)
- (6) flow into (positive) or out of (negative) domain
- (7) fictitious domain displacement or, if fluid flow is active, domain area (double precision)
- (8) *x*-coordinate of domain center
- (9) *y*-coordinate of domain center

PRINT **energy**

energy totals for energy components stored and dissipated in the model. The components are:

- (1) current kinetic energy (U_k)
- (2) total block strain energy (U_{cb})
- (3) total fill strain energy (U_{cf})
- (4) total joint strain energy (U_{cj})
- (5) total material strain energy ($U_c = U_{cb} + U_{cf} + U_{cj}$)
- (6) total block energy excavated (U_{mb})
- (7) total joint energy excavated (U_{mj})
- (8) total strain energy excavated ($U_m = U_{mb} + U_{mj}$)
- (9) total block volume excavated (V_m)
- (10) total change in potential energy (U_b)
- (11) total mass damping work (W_k)
- (12) total viscous boundary work (W_v)
- (13) total friction work (W_j)
- (14) total plastic strain work (W_p)
- (15) total boundary loading work (W)
- (16) total energy released ($W_r = W - U_c - U_b - W_j - W_p$)
- (17) total energy released ($W_r = U_k + W_k + W_v + U_m$)
- breakdown of energy stored in joints (U_{cj}):
- (18) total energy stored in tension (U_{jt})
- (19) total energy stored in compression (U_{jc})
- (20) total energy stored in shear (U_{js})

fboundary data for the porous medium outer boundary, including a listing of fluid pressures and node coordinates. The column headings are:

- (1) grid node number
- (2) pressure at node
- (3) x -coordinate of node
- (4) y -coordinate of node
- (5) node pressure code:
 - 0 pressure not fixed
 - 1 pressure fixed
- (6) address of boundary corner linked to node

fish lists *FISH* symbols, their current values and an indication of their status (whether functions or not). Variables with names beginning with **\$** are not printed (use **PRINT \$fish**).

PRINT fishcall

fishcall	lists functions that have been assigned as fishcalls .
ffield	free-field data. The following keywords are available.
bound	free-field boundary conditions
disp	free-field displacements
ref	reference stresses
state	free-field general data
stress	free-field stresses
flow*	fluid flow data. The column headings are:
	(1) contact address
	(2) <i>x</i> -coordinate of contact
	(3) <i>y</i> -coordinate of contact
	(4) flow rate across contact
	(5) joint length associated with contact
	(6) mean aperture at contact
	(7) address of domain on one side of contact
	(8) address of domain on other side of contact
	(9) mean flow (flow rate/aperture)
fos	the current value of the factor of safety. This value is calculated by the SOLVE fos command.
gridpoint*	keyword < <i>n</i> >
	gridpoint data (for deformable blocks). If gridpoint address <i>n</i> is given, all data for that gridpoint will be printed. The following keywords are used to print the information separately. The keyword must be specified.
disp	gridpoint displacements and velocities. The column headings are:
	(1) gridpoint address
	(2) <i>x</i> -coordinate of gridpoint
	(3) <i>y</i> -coordinate of gridpoint
	(4) <i>x</i> -velocity of gridpoint
	(5) <i>y</i> -velocity of gridpoint
	(6) <i>x</i> -displacement of gridpoint
	(7) <i>y</i> -displacement of gridpoint

PRINT **gridpoint*** **force**

force gridpoint forces. The column headings are:

- (1) gridpoint address
- (2) *x*-coordinate of gridpoint
- (3) *y*-coordinate of gridpoint
- (4) *x*-direction force at gridpoint
- (5) *y*-direction force at gridpoint

state gridpoint state data. The column headings are:

- (1) gridpoint address
- (2) address of linked corner
- (3) *x*-coordinate of gridpoint
- (4) *y*-coordinate of gridpoint
- (5) gridpoint mass
- (6) density scaling factor

temp gridpoint temperature (see [Section 3](#) in **Special Features**). The column headings are:

- (1) gridpoint address
- (2) *x*-coordinate of gridpoint
- (3) *y*-coordinate of gridpoint
- (4) temperature at gridpoint

groups lists the names used for user-defined groups.

history **<n1, n2 ... >** **<keyword>**

record of variables **n1**, **n2**, ... versus time. If no history number is specified, a list of all history types and locations is printed. The printout can be limited to a specified range of steps by using the following keywords.

begin **ncb**

Histories will be output beginning with step number **ncb**.

end **nce**

Histories will be output ending with step number **nce**.

skip **nc**

Only one point for every **nc** history points recorded is actually output. For example, **skip 10** means that every 10th recorded point (starting with the first) will be displayed.

PRINT hmax

n1, n2...
maxima of histories *n1, n2, ...* set by the **HISTORY** command

info status of global variables

jhist <keyword>

joint model contact parameter for Barton-Bandis joints (see [Section 3](#) in **Constitutive Models** for details). The keywords are:

aperture aperture data for Barton-Bandis joints

normal normal direction data for Barton-Bandis joints

shear shear direction data for Barton-Bandis joints

jmtab prints joint material table which may be used to assign joints.

joint *x1 y1 x2 y2* <keyword> <*tolc*>

If no keyword is specified, contact data (e.g., state, displacement and force) will be printed 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 default value for *tolc* is twice the rounding length.

The following keywords can also be given to print specific data for contacts along a single joint. The data are given at distance (R) from joint endpoint (*x1,y1*).

ape aperture

flow flow rate

nstr normal stress

pp domain pressure

sstr shear stress

vel mean fluid flow velocity

jprop name

lists the values of named properties assigned to contacts using the **JOINT** command.

jregion lists defined joint regions.

label lists user-defined aliases for histories and tables. Also lists user-defined plotting labels.

PRINT **linked**

linked lists all zone gridpoints that are slaved due to the blocks being joined.

list block linked-list. A list is printed for all corner and contact addresses associated with each block.

maxima maxima, minima and average values of block and zone data, and summary of contact data. The memory currently used for the model (*mfree* in words) and the total memory available (*mtop* in words) are also printed.

memory memory currently in use by the model, and total memory available

pline *<n>* keyword

Selected zone or gridpoint variables of deformable blocks may be printed versus distance along a line. The line is defined by the **SET pline** command. The line is identified by its line number, *n*. If *n* is omitted, all lines will be printed. The following keywords are available to print selected variables.

disp gridpoint displacement

nstress normal stress

sig1 zone major principal stress

sig2 zone minor principal stress

ssstress shear stress

sxx zone *xx*-stress component

sxy zone *xy*-stress component

syy zone *yy*-stress component

xdisp gridpoint *x*-displacement

xvelocity gridpoint *x*-velocity

ydisp gridpoint *y*-displacement

yvelocity gridpoint *y*-velocity

NOTE: No interpolation is made – the value of the nearest data point to the search point on the **pline** is printed.

PRINT**property**

property **<n1 . . . n2>** keyword

prints property arrays. Optional parameters **n1** and **n2** can be used to define the material number range to be printed.

The following keywords are available.

anis anisotropic elastic properties

block block properties

boundary far-field material properties

cable cable properties

contact contact properties (**jcons** = 1)

fluid fluid flow properties

joint joint properties (**jcons** = 2 or 5)

name material property value assigned to property name by either the **ZONE** or **JOINT** command (see **ZONE** and **JOINT** commands for property names)

reinf reinforcing element properties

structural structural (beam) element properties

support structural support properties

thermal thermal properties

reinforce local reinforcing data. The column headings are:

- (1) reinforcing contact address
- (2) material number for reinforcing
- (3) *x*-coordinate of reinforcing contact
- (4) *y*-coordinate of reinforcing contact
- (5) axial force in reinforcing
- (6) shear force in reinforcing
- (7) axial displacement in reinforcing
- (8) orientation of reinforcing to joint
- (9) 1/2 the active length of reinforcing
- (10) address of one block at reinforcement
- (11) address of other block at reinforcement

PRINT **rigid**

rigid rigid block data. The column headings are:

- (1) block address
- (2) x -velocity at centroid
- (3) y -velocity at centroid
- (4) angular velocity at centroid
- (5) x -force at centroid
- (6) y -force at centroid
- (7) centroid moment

rockbolt keyword

information on rockbolt elements. The following keywords are available.

element <keyword>

information on rockbolt element segments. The following optional keywords are available to print information separately.

disp displacements. The column headings are:

- (1) rockbolt element address
- (2) x -coordinate of midpoint
- (3) y -coordinate of midpoint
- (4) x -displacement of element
- (5) y -displacement of element

force forces. The column headings are:

- (1) rockbolt element ID number
- (2) axial force (compression positive)
- (3) shear force
- (4) moment at one end of element
- (5) moment at other end of element
- (6) axial strain

PRINT **rockbolt** **element** **geom**

geom geometric data. The column headings are:

- (1) rockbolt element ID number
- (2) node ID number at one end of element
- (3) node ID number at other end of element
- (4) material number
- (5) length
- (6) angle
- (7) area
- (8) moment of inertia

node <keyword>

Information on rockbolt element nodes. The following optional keywords are available to print information separately.

disp displacements. The column headings are:

- (1) node ID number
- (2) *x*-displacement of node
- (3) *y*-displacement of node
- (4) *x*-velocity of node
- (5) *y*-velocity of node
- (6) rotational velocity of node

force forces. The column headings are:

- (1) node ID number
- (2) *x*-direction force at node
- (3) *y*-direction force at node
- (4) moment at node
- (5) scaled mass at node
- (6) scaled moment of inertia at node

state node state. The column headings are:

- (1) node ID number
- (2) node address
- (3) *x*-coordinate of node
- (4) *y*-coordinate of node
- (5) mass of node

PRINT security

security	prints out information that is stored on the hardware key.
status	<p>present status of global variables. The column headings are:</p> <ol style="list-style-type: none"> (1) configure settings (2) mechanical settings (3) gravity (4) block tolerances (5) model update frequencies (6) timestep parameters (7) solve setting (8) hardcopy parameters (9) creep timestep parameters
stress	<p><i>xl xu yl yu nx ny</i></p> <p>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."</p>
struct	<p>keyword</p> <p>information on structural elements. The following keywords are available.</p>
element	<p><keyword></p> <p>information on structural element segments. The following optional keywords are available to print information separately.</p>
disp	<p>displacements. The column headings are:</p> <ol style="list-style-type: none"> (1) structural element address (2) <i>x</i>-coordinate of midpoint (3) <i>y</i>-coordinate of midpoint (4) <i>x</i>-displacement of element (5) <i>y</i>-displacement of element
force	<p>forces. The column headings are:</p> <ol style="list-style-type: none"> (1) structural element ID number (2) axial force (compression positive) (3) shear force (4) moment at one end of element (5) moment at other end of element (6) failure status

PRINT**struct****element****geom****geom** geometric data. The column headings are:

- (1) structural element ID number
- (2) node ID number at one end of element
- (3) node ID number at other end of element
- (4) material number
- (5) length
- (6) angle
- (7) thickness
- (8) area
- (9) moment of inertia
- (10) cross-section shape factor

interface structural element interface data. The column headings are:

- (1) address of interface contact
- (2) material number
- (3) constitutive model number
- (4) x -/ y - coordinates of contact
- (5) normal/shear forces
- (6) normal/shear displacements
- (7) ratio of shear to normal force
- (8) failure status
- (9) length associated with contact
- (10) angle of contact surface relative to x -axis
- (11) structural lumped mass/block addresses

node <keyword>

Information on structural element nodes. The following optional keywords are available to print information separately.

disp displacements. The column headings are:

- (1) node ID number
- (2) x -displacement of node
- (3) y -displacement of node
- (4) x -velocity of node
- (5) y -velocity of node
- (6) rotational velocity of node

PRINT **struct** **node** **force**

force forces. The column headings are:

- (1) node ID number
- (2) x -direction force at node
- (3) y -direction force at node
- (4) moment at node
- (5) scaled mass at node
- (6) scaled moment of inertia at node

state node state. The column headings are:

- (1) node ID number
- (2) node address
- (3) x -coordinate of node
- (4) y -coordinate of node
- (5) mass of node
- (6) moment of inertia of node

support support element data. The column headings are:

- (1) address of support element
- (2) address of top block
- (3) address of bottom block
- (4) material number for element
- (5) x -coordinate of element
- (6) y -coordinate of element
- (7) normal force on element
- (8) normal displacement of element

table *<ntab>*

displays list of tables. With *ntab*, lists values in table *ntab*.

thermal thermal boundary conditions and sources (see [Section 3](#) in **Special Features**)

varz *<prop>*

prints out property values for specified properties. This only works for models that have been assigned using the **ZONE** or **JOINT** command.

version prints version and sub-version number of *UDEC*.

PRINT**well****well**

list of wells. The column headings are:

- (1) domain address of well
- (2) x -coordinate of well
- (3) y -coordinate of well
- (4) flow rate at well

zone

keyword

<*n*>

zone data (for deformable blocks). If the zone address *n* is given, all data for that zone are printed. The following keywords are used to print the information separately. The keyword must be specified.

group

zone group names

- (1) zone address
- (2) x -centroid of zone
- (3) y -centroid of zone
- (4) group name

mixed

mixed discretization data

- (1) zone address
- (2) mixed discretization code
- (3) matched zone
- (4) 1st zone in overlay set of zones
- (5) 2nd zone in overlay set of zones

model

constitutive models of zones

- (1) zone address
- (2) x -centroid
- (3) y -centroid
- (4) model name

PRINT **zone** **pp**

pp zone pore pressures and plasticity indicators. The column headings are:

- (1) zone address
- (2) zone pore pressure
- (3) plasticity indicator
 - 0 elastic
 - 1 at failure
 - 2 failed in the past
 - 4 tension failure
- (4) x -coordinate of zone centroid
- (5) y -coordinate of zone centroid

princ principal stresses. The column headings are:

- (1) zone address
- (2) major principal stress (σ_1)
- (3) intermediate principal stress (σ_2)
- (4) minor principal stress (σ_3)
- (5) angle of σ_1 relative to x -axis
- (6) plasticity indicator
 - 0 elastic
 - 1 at failure
 - 2 failed in the past
 - 4 tension failure
- (7) x -coordinate of zone centroid
- (8) y -coordinate of zone centroid

state zone state data. The column headings are:

- (1) zone address
- (2) addresses of gridpoints associated with the zone
- (3) zone mass
- (4) plasticity indicator
 - 0 elastic
 - 1 at failure
 - 2 failed in the past
 - 4 tensile failure

PRINT**zone****stress**

stress

zone stresses. The column headings are:

- (1) zone address
- (2) xx -stress component
- (3) xy -stress component
- (4) yy -stress component
- (5) zz -stress component
- (6) plasticity indicator
 - 0 elastic
 - 1 at failure
 - 2 failed in the past
 - 4 tensile failure
- (7) x -coordinate of zone centroid
- (8) y -coordinate of zone centroid

PROPERTY

PROPERTY **material** *n* keyword *v* <keyword *v* . . . >
jmaterial *n*

This command assigns properties to material model numbers for block, joint, reinforcement, cable, structural element and support element models. 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, structural element and support element properties are also assigned with the **PROPERTY** command to **material** number *n*. (See the **CABLE**, **REINFORCE**, **STRUCT** and **SUPPORT** commands and [Section 1](#) in **Special Features**.)

CAUTION: Do not use material numbers for structural element interface properties that are the same as those for joint material numbers. Always use the **PRINT property** command to check property assignment.

PROPERTY keywords are presented in six categories: joint material, block material, reinforcing element, cable element, structural (beam) element and support element. There is also a seventh category, which uses the **PROPERTY** command to assign strength properties used to create contour plots of strength/stress ratios for deformable blocks. These properties apply when the **PLOT mohr** or **PLOT hoek** command is invoked.

* Block and joint constitutive models and properties can be assigned locally to blocks and joints with the **ZONE**, **JMODEL** and **JOINT** commands, respectively. If the **ZONE**, **JMODEL** or **JOINT** method is used, each block or contact will have its own set of properties. This method requires more computer memory. This method is required if *FISH* is used to modify properties of individual blocks or contacts.

1. Joint Material Properties (**jmaterial** keyword – see [Table 1.3](#))

jcons = 1 (point contact – Coulomb slip)

- | | | |
|-----|-------------------|--|
| (1) | ccohesion | contact cohesion [force] |
| (2) | cdilation | contact dilation angle [degrees] |
| (3) | cfriiction | contact friction angle [degrees] |
| (4) | cperm | wetting fluid contact permeability [$\text{length}^2/(\text{stress} \cdot \text{time})$] |
| (5) | ctension | contact tensile strength [force] |
| (6) | kn | contact normal stiffness [force/displacement] |
| (7) | ks | contact shear stiffness [force/displacement] |
| (8) | nwcpm | non-wetting fluid contact permeability [$\text{length}^2/(\text{stress} \cdot \text{time})$] |

The default value for these properties is zero.

jcons = 2 (joint area contact – Coulomb slip)

- | | | |
|------|------------------|--|
| (1) | ares | residual aperture at high stress [length] |
| (2) | azero | aperture for zero normal stress [length] |
| (3) | empb | empirical multiplier for fluid flow law (default = 1) |
| (4) | expa | exponent of joint hydraulic aperture (default = 3) used for fluid flow only |
| (5) | jcohesion | joint cohesion [stress] |
| (6) | jdilation | joint dilation angle [degrees] |
| (7) | jfriction | joint friction angle [degrees] |
| (8) | jkn | joint normal stiffness [stress/displacement] |
| (9) | jks | joint shear stiffness [stress/displacement] |
| (10) | jperm | wetting fluid joint permeability constant (see Eq. (2.3) in Special Features , also called joint permeability factor) [$1/(\text{stress} \cdot \text{time})$] |
| (11) | jtension | joint tensile strength [stress] |
| (12) | nstable | normal stiffness table |
| (13) | nwjperm | non-wetting fluid joint permeability constant [$1/(\text{stress} \cdot \text{time})$] |
| (14) | zdilation | shear displacement for zero dilation (default: dilation not affected if zdilation = 0.0) [displacement] |

The default value for these properties (with the exception, as noted, of **empb** and **expa**) is zero. **jcons = 2** is the default constitutive model; at a minimum, **jkn** and **jks** *must* be specified to execute a model run.

PROPERTY

jcons = 5 (joint area contact – Coulomb slip with residual strength)

- | | | |
|------|------------------|---|
| (1) | ares | residual aperture at high stress [length] |
| (2) | azero | aperture for zero normal stress [length] |
| (3) | empb | empirical multiplier for fluid flow law (default = 1) |
| (4) | expa | exponent of joint hydraulic aperture (default = 3) used for fluid flow only |
| (5) | jcohesion | joint cohesion [stress] |
| (6) | jdilation | joint dilation angle [degrees] |
| (7) | jfriction | joint friction angle [degrees] |
| (8) | jkn | joint normal stiffness [stress/displacement] |
| (9) | jks | joint shear stiffness [stress/displacement] |
| (10) | jperm | wetting fluid joint permeability constant
(see Eq. (2.3) in Special Features , also called joint permeability factor) [1/(stress · time)] |
| (11) | jrescoh | joint residual cohesion [stress] |
| (12) | jrfric | joint residual friction angle (default = jfriction) [degrees]
The residual friction cannot be exactly zero (use a small value). |
| (13) | jrtens | joint residual tensile strength [stress] |
| (14) | jtension | joint tensile strength [stress] |
| (15) | nstable | normal stiffness table |
| (16) | nwjperm | non-wetting fluid joint permeability constant [1/(stress · time)] |
| (17) | zdilation | shear displacement for zero dilation (default: dilation not affected if zdilation = 0.0) [displacement] |

The default value for these properties (with the exception, as noted, of **empb**, **expa** and **jrfric**) is zero.

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, joint friction angle and joint cohesion are set to residual values. For **jcons = 2**, joint tensile strength, joint friction angle and joint cohesion are preserved regardless of loading path. No fluid flow is assumed to occur in unfractured **jcons = 5** joint segments. To allow flow in unfractured **jcons = 5** joint segments, use **SET j5flow on** (see the **SET** command).

PROPERTY

jcons = 3 (continuously yielding)

- | | | |
|------|------------------|---|
| (1) | ares | residual aperture at high stress [length] |
| (2) | azero | aperture for zero normal stress [length] |
| (3) | empb | empirical multiplier for fluid flow law (default = 1) |
| (4) | expa | exponent of joint hydraulic aperture (default = 3) used for fluid flow only |
| (5) | jen | exponent of joint elastic normal stiffness |
| (6) | jes | exponent of joint elastic shear stiffness |
| (7) | jifric | joint initial friction angle [degrees] |
| (8) | jfriction | joint intrinsic friction angle [degrees] |
| (9) | jkn | joint normal stiffness [stress/displacement] |
| (10) | jks | joint shear stiffness [stress/displacement] |
| (11) | jperm | wetting fluid joint permeability constant
(see Eq. (2.3) in Special Features , also called joint permeability factor) [1/(stress · time)] |
| (12) | jr | joint roughness parameter [length] |
| (13) | maxjkn | maximum value of joint normal stiffness
(default is maxjkn=jkn) |
| (14) | maxjks | maximum value of joint shear stiffness
(default is maxjks=jks) |
| (15) | minjkn | minimum value of joint normal stiffness
(default is minjkn=jkn) |
| (16) | minjks | minimum value of joint shear stiffness
(default is minjks=jks) |
| (17) | nwjperm | non-wetting fluid joint permeability constant [1/(stress · time)] |

The default values for **jen**, **jes**, **jif** and **jr** are zero. If neither **minjkn** nor **maxjkn** is defined, joint normal stiffness will be constant (i.e., exponent **jen** will have no effect). Joint shear stiffness is defined in a similar manner.

Timestep calculations for **jcons = 3** joints are based on the values of **maxjkn** and **maxjks**. In general, it will not be necessary to further reduce the timestep for numerical stability. The user should avoid very large values for **maxjkn** or **maxjks**, as these might result in very small timesteps.

NOTES: For **jcons** = 2 or 5:

- (1.) A nonlinear normal stress-normal displacement relation may be assigned, by means of a table, with the keyword

nstable **n**

Table **n** (see the **TABLE** command) contains a list of pairs (u_n, σ_n) in which u_n is normal displacement and σ_n is normal stress, defining the joint stress-displacement behavior in the compressible range. (u_n must be negative or zero, and σ_n must be positive or zero.) If the point (0,0) is not given, it will be added to the table. In the tensile range, constant normal stiffness equal to the first table segment is used. For unloading and reloading, the maximum stiffness attained in the past is used.

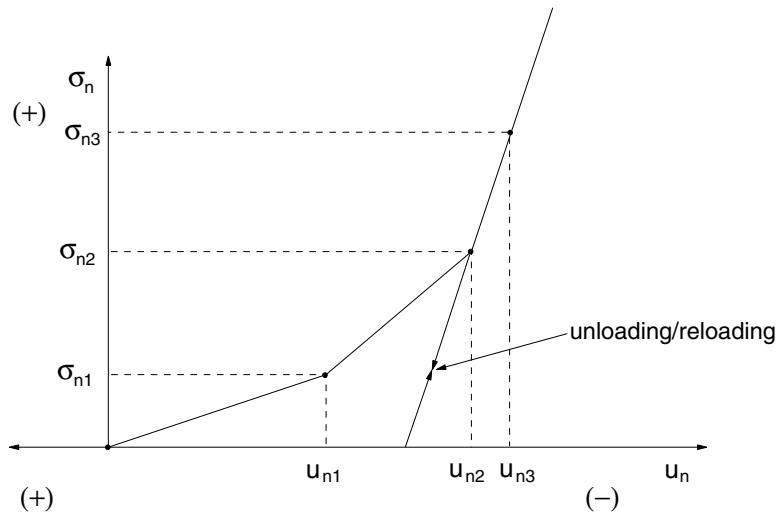


Figure 1.4 **TABLE 1** (0,0) (u_{n1}, σ_{n1}) (u_{n2}, σ_{n2}) (u_{n3}, σ_{n3})

- (2.) Joint dilation is effective when the joint is at slip. The normal displacement increment, Δu_n^d , due to dilation is calculated as

$$\begin{aligned} \Delta u_n^d &= \tan(jdil) \Delta u_s \text{ sign}(u_s) & \text{if } |u_s| < zdil \\ &= 0 & \text{if } |u_s| > zdil \end{aligned}$$

where **zdil** is shear displacement for zero dilation as prescribed by **zdilation**.

PROPERTY

Note that dilation is positive if the shear displacement increment (Δu_s) is in the direction of the total shear displacement (u_s), and negative otherwise.

If **SET add_dil off** (default), the effective friction is equal to **jfriction**. If **SET add_dil on**, the effective friction angle is obtained by adding the dilation angle (**jdilation**) to **jfriction** if the dilation is positive, or subtracting it from **jfriction** if the dilation is negative.

NOTES: For **jcons** = 2, 5 or 3:

- (1.) 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.
- (2.) Joint permeability can be specified as a function of temperature by means of the keyword

ktable **n**

Table **n** (see the **TABLE** command) contains a list of pairs (e.g., permeability and temperature) defining the temperature dependency. The table applies to *all* joint permeabilities regardless of joint material number. Note that for a “parallel-plate” joint, **jperm** = $(1/12)\mu$, in which μ is the fluid-dynamic viscosity.

2. Block Material Properties (**material** keyword – see [Table 1.2](#))

cons = 0 (null model) or **cons = 1** (isotropic, elastic)

- (1) **bulk** (or **k**) bulk modulus [stress]
- (2) **density** block density [mass/volume]
- (3) **shear** (or **g**) block shear modulus [stress]

The block properties are needed in the null model to allow internal gridpoints to move as the null block deforms. The choice of null property values will not affect model results, but values that won't reduce the critical timestep should be selected.

cons = 3 (Mohr-Coulomb plasticity) or **cons = 6** (Drucker-Prager plasticity)

- (1) **bulk** (or **k**) bulk modulus [stress]
- (2) **cohesion** block cohesion [stress]
- (3) **density** block density [mass/volume]
- (4) **dilation** block dilation angle [degrees]
- (5) **friction** block friction angle [degrees]
- (6) **shear** (or **g**) block shear modulus [stress]
- (7) **tension** block tensile strength [stress]

To convert Mohr-Coulomb properties to Drucker-Prager properties, see [Eqs. \(1.66\)](#) through [\(1.69\)](#) in **Constitutive Models**.

NOTES:

- (1.) The default value for the block material properties is zero. For a rigid block model, **density** must be specified to execute a model run. For deformable blocks, **density**, **bulk** and **shear** must all be specified.
- (2.) The equations for bulk (K) and shear (G) modulus are

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

and

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

where E is Young's modulus, and ν is Poisson's ratio. The user may enter either bulk and shear modulus, or Young's modulus and Poisson's ratio.

PROPERTY

- (3.) Thermal properties can be assigned with any of the block models. The following **PROPERTY** keywords prescribe thermal properties (see [Section 3](#) in **Special Features**).

- (1) **cond** thermal conductivity
- (2) **specheat** specific heat
- (3) **thexp** linear thermal expansion coefficient
- (4) **xcond** thermal conductivity in x -direction
- (5) **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.

- (4.) Poroelastic properties can be assigned with any of the block models. The following **PROPERTY** keyword proscribes a poroelastic property.
- (1) **biot_coef** Biot's constant for porous elastic rocks

3. Reinforcing Element Properties (**material** keyword and **REINFORCE** command)

- | | | |
|------|------------------|---|
| (1) | r_aexp | axial stiffness exponent (usually 1) |
| (2) | r_astiff | axial stiffness [force/length] |
| (3) | r_length | 1/2 “active” length [length] |
| (4) | r_rfac | reversal factor (between 0 and 1) |
| (5) | r_sexp | shear stiffness exponent (usually 1) |
| (6) | r_spacing | spacing of reinforcements in the out-of-plane direction
[length] (default = 1.0) |
| (7) | r_sstiff | shear stiffness [force/length] |
| (8) | r_str | axial failure strain (default = 10^{10}) |
| (9) | r_uaxial | ultimate axial capacity (must be > 0) [force] |
| (10) | r_ushear | ultimate shear capacity (must be > 0) [force] |

The default values for reinforcing properties (except as noted for **r_str**) are zero.

4. Cable Properties (**material** keyword and **CABLE** command)

Cable element properties:

- (1) **cb_area** cross section area of cable
- (2) **cb_density** mass density for cable reinforcing [mass/volume]
- (3) **cb_fstrain** extensional failure strain (default = 10^{10})
- (4) **cb_spacing** spacing of cables in out-of-plane direction (default = 1.0)
- (5) **cb_ycomp** compressive yield force for cable reinforcing
(use positive value) [force]
- (6) **cb_yield** tensile yield force for cable reinforcing
(use positive value) [force]
- (7) **cb_ymod** Young's modulus for cable reinforcing [stress]
- (8) **cb_thexp** thermal expansion coefficient for cable

Grout properties:

- (1) **cb_kbond** grout shear stiffness [force/unit cable length/displacement]
- (2) **cb_sbond** grout shear strength [force/unit cable length]

The default values for cable properties (except as noted for **cb_fstrain**) are zero.

5. Structural Element Properties (**material** keyword and **STRUCT** command)

Element properties:

- (1) **st_area** element area (optional; calculated from thickness, or thickness and width, if not specified) [length²]
- (2) **st_density** mass density of element [mass/volume]
- (3) **st_inertia** second moment of inertia (optional; calculated from thickness, or area and thickness, if not specified) [length⁴]
- (4) **st_phtable** table defining the moment thrust diagram
- (5) **st_prat** Poisson's ratio
- (6) **st_rcrack** crack depth ratio (default = 0)
- (7) **st_scred** residual cracked beam compressive strength [stress] (default = 0.0)
- (8) **st_shape** element shape (optional; default = 5/6; see [Figure 1.32](#) in **Special Features**)
- (9) **st_spacing** element spacing in out-of-plane direction if it isn't continuous [length] (optional; default = 1.0)
- (10) **st_thick** element thickness [length]
- (11) **st_width** element width if not continuous in out-of-plane direction [length] (optional; default = 1.0)
- (12) **st_ycomp** compressive yield strength [stress] (default = 1e10)
- (13) **st_yield** tensile yield strength [stress] (default = 1e10)
- (14) **st_ymod** elastic modulus [stress]
- (15) **st_yresid** residual tensile yield strength [stress]
- (16) **st_thexp** thermal expansion coefficient

Interface properties (**jcons = 5** material):*

- (1) **if_cohesion** interface cohesion [stress]
- (2) **if_dilation** interface dilation [degrees]
- (3) **if_friction** interface friction angle [degrees]
- (4) **if_kn** interface normal stiffness [stress/displacement]
- (5) **if_ks** interface shear displacement [stress/displacement]
- (6) **if_tensile** interface tensile strength [stress]

The default value for structural (beam) element properties is zero (except as noted above).

* **mat number** must be different than any contact material number, because the property arrays overlap.

6. Support Element Properties (**material** keyword and **SUPPORT** command)

Standard support elements:

- (1) **sup_kn** axial stiffness of the support member.

If the member contains sub-elements, **sup_kn** is the stiffness for the group of sub-elements. [force/displ.]

If the specified value for **sup_kn** is negative, then a **TABLE** provides the relation between axial force and axial displacement. The table number is the absolute value of the number specified for **sup_kn**. (Note that table should not be used if the support is subject to unloading.)

- (2) **sup_spacing** element spacing in out-of-plane direction (default = 1.0).

- (3) **sup_ycomp** compressive yield force.

If the member contains sub-elements, **sup_ycomp** is the compressive yield force for the group of sub-elements. [force]

Load-rate dependent elements:

- (1) **sup_alpha** exponent constant, α

- (2) **sup_constant** support stiffness constant, k

- (3) **sup_fmax** maximum support force addition, ΔF_{max}

- (4) **sup_tmax** tensile strength, t_{max} (stress)

7. Strength Properties for Strength/Stress Contour Plots

Contour plots of strength/stress ratios for zones can be made based on either the Mohr-Coulomb (M-C) or Hoek-Brown (H-B) failure criterion (see **PLOT mohr** or **PLOT hoek**, and [Section 3.9](#) in the **User's Guide**, Tip 10). The strength parameters for each plot are prescribed via the **PROPERTY** command and the following keywords.

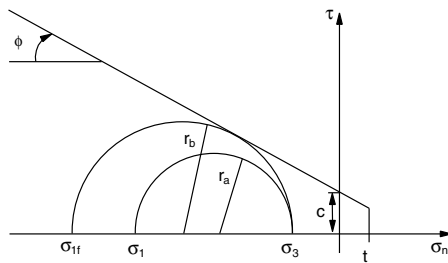
M-C Strength Criteria (**material** keywords and **PLOT mohr** command)

- (1) **cohesion** cohesion [stress]
- (2) **friction** friction angle [degrees]
- (3) **tension** tension limit [stress]

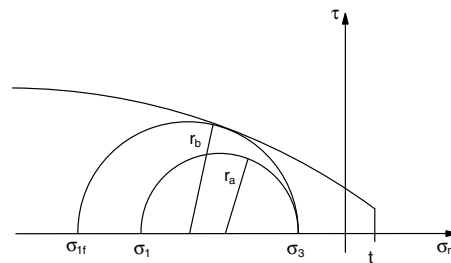
H-B Strength Criteria (**material** keywords and **PLOT hoek** command)

- (1) **hbm** Hoek-Brown “m” parameter
- (2) **hbs** Hoek-Brown “s” parameter
- (3) **ucs** unconfined compressive strength [stress]

Mohr-Coulomb



Hoek-Brown



$$\sigma_{1f} = \left(\frac{1 + \sin \phi}{1 - \sin \phi} \right) \sigma_3 - 2c \sqrt{\frac{1 + \sin \phi}{1 - \sin \phi}}$$

$$\sigma_{1f} = \sigma_3 - \sqrt{-m \sigma_c \sigma_3 + s \sigma_c^2}$$

Strength/Stress ratio:

$$r_b/r_a = \frac{\sigma_3 - \sigma_{1f}}{\sigma_3 - \sigma_1}$$

Figure 1.5 *Strength/stress ratios for the Mohr-Coulomb and Hoek-Brown failure criteria*

PROPERTY

8. Domain Material Properties (**dmaterial** keyword)

- (1) **acap** capillary pressure reference hydraulic aperture [length]
- (2) **bcap** β coefficient from the definition of the capillary pressure curve
- (3) **gcap** γ coefficient from the definition of the capillary pressure curve [length]

See [Section 2](#) in **Special Features** for a detailed explanation of the capillary pressure curve and its use in *UDEC*.

9. Rockbolts

Rockbolt properties are assigned using the **STRUCT prop** command.

QUIT

QUIT stops execution of *UDEC* (a synonym for **STOP**).

REINFORCE

REINFORCE *mat x1 y1 x2 y2*
delete <range>

Execution of this command creates reinforcing elements across joints that intersect the line with endpoints (*x1,y1*) and (*x2,y2*). Reinforcement can be used with rigid and deformable blocks. If reinforcement is to be used with deformable blocks, the blocks *must* be discretized before reinforcement locations are specified. Reinforcement properties are specified and stored using the **PROPERTY** command with material number *mat*. If the **delete** keyword is used, all reinforcement elements in the specified range will be deleted.

RESET

RESET keyword <keyword>

Certain variables are reset according to the following keywords.

- | | |
|------------------|--|
| damp | Automatic damping parameter is reset to initial (high) value for adaptive global damping. (This command is the same as the DAMPING initial command.) |
| disp | All gridpoint displacements of deformable 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 | All current histories are lost. |
| interface | <i>m1 m2</i>

Only joints between block materials <i>m1</i> and <i>m2</i> have joint stresses and displacements reset to zero (must precede stress keyword – see below). |
| jdisp | All joint normal and shear displacements are set to zero for plotting or printing incremental changes in displacement. The actual displacements are not reset, because this can affect results if dilation is present. |
| jndisp | All joint normal displacements are set to zero for plotting or printing incremental changes in displacement. The actual displacements are not reset, because this can affect results if dilation is present. |
| mat | <i>n</i>

Only blocks with material <i>n</i> have stresses reset to zero (must precede stress keyword – see below). |
| reldis | <i>dx, dy</i>

Change the displacements for gridpoints by the specified values <i>dx</i> and <i>dy</i> . |
| relstr | <i>sxx, sxy, syy</i>

Change the stresses in zones by the specified values <i>sxx</i> , <i>sxy</i> and <i>syy</i> . |
| rot | All zone and block rotations are set to zero. |

RESET**stress**

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.

Zone pore pressures are also set to zero.

thist All current thermal histories are lost.

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.

velocity All model velocities are set to zero.

RESTORE

RESTORE <filename>

A previously saved problem state is restored from the named file **filename**. (See the **SAVE** command.) A path can be part of the file name. If no file name is given, the default file “UDEEC.SAV” will be restored.

The following files are unaffected by the **RESTORE** command:

history file (see **HISTORY write**)

log file (see **SET log**)

movie file (see **MOVIE**)

plot file (see **SET output**)

These files remain open if open already, and their file names are not changed when a **RESTORE** command is given. New file names can be specified after the **RESTORE** command, if required.

The echo mode (**SET echo**) and message mode (**SET message**) are unaffected by **RESTORE**. These modes can be turned on and off as needed.

All other conditions and values are taken from the **RESTOREd** save file. For example, *FISH* functions and variables are restored from the save file. Functions, variables and histories in existence prior to the use of the **RESTORE** command are lost.

RESTORE

RETURN

RETURN This command returns control from a data file to the keyboard. This command will also cause an exit from a **COMMAND** section in a *FISH* function.

ROUND

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 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 should not be changed, especially after cycling is initiated.

RUN**RUN**<<keyword *value*> ... >

This command executes thermal timesteps (see [Section 3](#) in **Special Features** for details). 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	<i>t</i>	thermal “heating time” limit (in consistent units with input properties)
delt	<i>dt</i>	The thermal timestep, <i>dt</i> , 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 that causes instability, this option can be used to further reduce the timestep.
implicit		uses the implicit scheme instead of the default explicit scheme.
noage		turns off the previously requested test for exceeding age <i>t</i> . The default for the age parameter is that the age is not tested until an age has been explicitly requested via an “ age = <i>value</i> ” following a RUN command.
step	<i>s</i>	thermal step limit (default: <i>s</i> = 100,000)
temp	<i>dtp</i>	maximum total temperature change, <i>dtp</i> , since the previous mechanical cycles (default: <i>dtp</i> = 20)
tol	<i>tol</i>	Gridpoints within tolerance, <i>tol</i> , from each other are combined for thermal calculations (default: <i>tol</i> = 0.1).

Old limits apply upon restart. When a **RUN** command has been completed, the program will indicate which parameter has caused it to terminate. To ensure that the run stops for the correct limit, the values of the others should be set very high.

The explicit scheme is always used unless the keyword **implicit** follows the **RUN** command.

SAVE

SAVE <filename>

All information required to restart the problem is saved to the named file **filename**. In interactive mode, if the file already exists, a warning will be written so that a different file name can be chosen, if desired. The default file name is “UDECSAV” if no file name is given. Note that all currently defined *FISH* functions and variables are saved on the file.

SET

SET keyword <keyword *value*> ...

This command is used to set parameters in a *UDEC* model. The parameters can be divided into two categories: output control conditions and model conditions. The keywords used in these categories are summarized in [Table 1.8](#):

Table 1.8 Summary of SET keywords

Output Control Conditions		Model Conditions		
a_us	npens	add_dil	gpneg	satmin
a3	output	added_mass	gravity	skip_error
a4	overwrite	bminw	j5flow	small
autoname	paginate	btol	jcondf	stcbmat
back	pline	capmin	jmatdf	stld
border	plot	capratio	jmtable	sup_delete
box	png	cavi	join_jkn	szz
char	pngfile	cb_tol	join_jks	tfres
color	propcontour	cd	join_ratio	thdt
cust1	psrot	checkesc	jointid	time
cust2	psxscal	clemin	latency	twophase
echo	psxshift	crack_flow	lmul	umul
filcolor	psyscal	crdt	maxdt	upcon
giicpath	psyshift	creeptime	maxgiicmem	uplen
jrepath	quionerror	cscan	maxmech	voltol
legend	replot	cutting	mech	write
log	sgrid	cysub	memory	zpres
message	square	dami	mindt	
nowhite	text	degrees	multi	
	transparency	delc	netserver	
		deterministic	nflow	
		dscan	nfmech	
		dtflow	nmech	
		edge	nodal	
		energy	nther	
		fishcall	oorc	
		flow	ovtol	
		fluid_dt_type	processors	
		fobl	ptol	
		fobu	radi	
		freeze_con	random	
		ftime	reftime	
		gas	rot	
		gpeq	satmax	

a_us	sets paper size to 8.5 in × 11 in (default).
a3	sets paper size to A3, 420 mm × 297 mm.
a4	sets paper size to A4, 210mm × 297mm
add_dil	<p>off on</p> <p>If on, the joint dilation angle will be added to, or subtracted from, the friction angle, depending on direction and magnitude of shear displacements (default is off).</p>
added_mass	<p>y_b y_t <range></p> <p>adds the mass due to a reservoir to a model for dynamic analysis of dams. (y_b – reservoir bottom; y_t – reservoir top)</p>
autoname	<p>off</p> <p>activates autonaming of PCX and graphical output files. Each time a plot is created, a unique, sequentially numbered file name is used. The initial name is defined by the SET output command or the SET pcxfile command. The SET overwrite flag is used to control whether previously existing files may be overwritten. The default for autoname is off.</p>
back	<p>color</p> <p>sets color of screen background. (Default is gray.) See Table 1.6 for descriptions of color keywords.</p>
bminw	<p>v</p> <p>defines the minimum ratio between non-wetting fluid bulk modulus and wetting fluid bulk modulus in the case of two-phase flow, SET twophase on; when non-wetting fluid bulk modulus is pressure dependent, SET gas on.</p>
border	<p>color</p> <p>sets line color of screen border. (Default is blue.) See Table 1.6 for descriptions of color keywords.</p>
box	<p>color</p> <p>sets color of screen legend box. (Default is white.) See Table 1.6 for descriptions of color keywords.</p>

SET	b _{tol}	
	b_{tol}	<i>v</i>
		sets maximum distance a contact may open before it is deleted (default = $0.55 \times$ rounding length).
	cap_{min}	<i>a</i>
		minimum contact hydraulic aperture. (Default is 1/10 the rounding length if ares = 0; otherwise, it is the minimum ares value.) The minimum hydraulic aperture is used only to calculate the minimum domain volume for transient flow calculations. It has no effect on hydraulic conductivity (i.e., minimum hydraulic apertures are always given by ares , which may be zero).
	cap_{ratio}	<i>r</i>
		Maximum contact hydraulic aperture is limited to <i>r</i> times the maximum residual aperture. (Default is 5.) Note that the maximum hydraulic aperture is a global variable based on the maximum residual aperture in the model. It does not vary for different contacts.
	cavi	off
		sets check for cavitation pressure in fluid flow. The default is on .
	cb_{tol}	<i>v</i>
		Having a cable node placed on the edge of a block may cause problems in the calculation of the distributed forces if the node subsequently moves into a different block. To avoid this, <i>UDEC</i> automatically moves cable nodes off of the block edges. The distance they are moved defaults to half of the rounding length. The SET cb_{tol} command allows the user to modify the distance the node is moved. The command should be used prior to the CABLE command.
	cd	internal GUI command used to change folders.
	checkesc	off on
		controls checking for <Esc> key while plotting. Default is off .
	char	<i>i</i>
		used to select alternate character sets for the HP plotters. Consult the appropriate manuals for applicable values (e.g., Scandinavian <i>i</i> = 30). (The default is <i>i</i> = 0 for standard ASCII.)

SET **clemin****clemin** *cmin*

minimum contact length for **jcons** = **2**, **3** or **5** joint models. (Default is 1/10 the average contact length.) This cannot be less than 2 times the rounding length.

color color

specifies default screen line color of model. (Default is green.) See [Table 1.6](#) for descriptions of color keywords.

crack_flow **off**
on

If **on**, fluid flow is restricted to joints for which contacts have failed. Otherwise, fluid flow is calculated for all joints. The default is **off**.

crdt *v* <**auto** <**on/off**>>

sets the current creep timestep. If the keyword **auto** is specified, the timestep will automatically be adjusted. The timestep will be increased by the factor **umul** if the maximum out-of-balance force stays below **fobl** for **latency** timesteps. The timestep will be reduced by the factor **lmul** if the maximum out-of-balance force stays above **fobu** for **latency** timesteps. The automatic timestep is bounded by the values **maxdt** and **mindt**. The creep and mechanical timesteps are not coupled. However, the creep time will be incremented by one creep timestep for each mechanical step taken. The mechanical time is independent of the creep time, and is only used to maintain mechanical equilibrium. Setting **crdt** to 0 turns creep off. The default value for the creep timestep is 0.

creeptime *v*

specifies current creep time.

cscan *n*

updates coordinates of center of radius of corners every *n* cycles. (Default is *n* = 100.)

cust1 allows setting the first line of the customer title printed on plots.**cust2** allows setting the second line of the customer title printed on plots.**cutting** **off**
on

allows cutting of blocks created via the **BLOCK fill** command to occur after cycling. Default is **off**.

SET **cysub****cysub** *n*

The stress calculations for the continuously yielding joint model are done in *n* sub-steps within each timestep (default: *n* = 0). See [Section 2](#) in **Constitutive Models**.

dami *v*

sets minimum domain volume used in the plotting of interior boundaries. (Has no effect on calculation.) **PLOT boundary** will plot domains that are 50 times this value.

degrees **off**
on

All friction and dilation properties for block, joint and structural element materials are input in degrees (default = **on**). If **off**, friction and dilation properties for material models defined by the **PROPERTY** or **STRUCT** command are input as tangent of angle (the format used in previous versions). This command does not apply to properties input via the **ZONE** command.

delc **off**
on

Contacts are automatically deleted when the separation is greater than 0.55 times the rounding length (default). Use the **PRINT status** command to find the limiting separation value being used. If **off**, contacts are never deleted. (The default is **off**.)

deterministic **off**
on

If **on**, deterministic calculations are enforced for multithreading. This adds about 20% to the calculation time, but all calculations are repeatable. If **off**, small round-off differences in the calculations may accumulate. The default is **on**.

dscan *n*

sets frequency of domain check. The default is every three cycles.

dtflow *dt* *<ratio>*

Fluid timestep is prescribed (required with **SET flow incompressible**). If the optional *<ratio>* is given, the fluid timestep will be multiplied by this factor every flow step within each **CYCLE** command. At the beginning of every **CYCLE** command, the timestep is restored to its initial value, and the process is reinitiated.

SET **echo**

echo **off**
on

SET echo on causes *UDEC* to echo input lines to the screen and to the log file (if it is open) when the lines come from a **CALLed** file or from a *FISH* function. This is the default setting. **SET echo off** prevents the lines from being displayed.

edge *emin*

The minimum block edge is set to *emin* (default is twice the rounding length). *emin* must be \geq twice the rounding length. **SET edge** should be specified *before* the **BLOCK** command. (This keyword is the same as **EDGE** command.)

energy **off**
on

If **on**, energy components are calculated. (See [Section 3](#) in **Theory and Background**.)

filcolor *value*
or
keyword *value*

This keyword is used to control the colors used for filled plots in *UDEC*. It is particularly useful for changing the color-scale range on filled contour plots.

ncol color

The fourth option, **SET filcolor ncol color**, acts similarly to the third, except that the color name (as defined in [Table 1.9](#)) is used instead of the color number. For example, `set filc 3 white` is equivalent to `set filc 3 7`. The keywords for colors are the same as for plot color switches (see [Table 1.5](#)).

Table 1.9 Color scales (nscal)

Index Number (ncol)	Scale 1		Scale 2		Scale 3	
1	13	light magenta	1	blue	1	blue
2	12	light red	9	light blue	2	green
3	4	red	3	cyan	3	cyan
4	6	brown	11	light cyan	4	red
5	14	yellow	2	green	5	magenta
6	10	light green	10	light green	6	brown
7	2	green	14	yellow	7	white
8	11	light cyan	6	brown	8	gray
9	3	cyan	4	red	9	light blue
10	9	light blue	12	light red	10	light green
11	1	blue	13	light magenta	11	light cyan
12	7	white	5	magenta	12	light red
13	8	gray	8	gray	13	light magenta
14	5	magenta	7	white	14	yellow
15	15	intense white	15	intense white	15	intense white
16	0	black	0	black	0	black

Table 1.10 Available colors

Number	Color
0	black
1	blue
2	green
3	cyan
4	red
5	magenta
6	brown
7	white
8	gray
9	light blue
10	light green
11	light cyan
12	light red
13	light magenta
14	yellow
15	intense white

ncol lcol

The third option, **SET filcolor ncol lcol**, changes color index number *ncol* in the current scale to color *lcol* on the list in [Table 1.10](#). For example, `set filc 3 7` changes the third color to white.

nscal

The first format given, **SET filcolor nscal**, selects one of the three built-in color scales. (The default scale is scale 1.) The colors in these scales are tabulated in [Table 1.9](#).

table

ntab

The second format, **SET filcolor table ntab**, uses the colors previously specified by the user with the **TABLE** command. The table must be input as pairs of numbers: the first member of the pair is the location in the table (number from 1 to 13), and the second is the color number corresponding to the desired color on the list in [Table 1.10](#).

For example, the commands

```
table 3 1,0 2,3 3,5...13,7
set filc table 3
```

set up a color scale with black, cyan and magenta as the first three colors and white as the last color.

fishcall

n <remove> name

The *FISH* function *name* will be called in *UDEC* from a location determined by the value of the **fishcall** ID number *n*. The currently assigned ID numbers are listed in [Table 2.2](#) in the *FISH* volume. When placed before the *FISH* function *name*, the optional keyword **remove** causes the *FISH* function to be removed from the list.

flow

keyword

The fluid-flow modes are controlled with the **SET flow** command. There are three flow modes that can be selected by one of the following keywords. (See [Section 2](#) in **Special Features**.)

clear turns off all fluid flow settings

compressible turns on compressible flow mode.

gas turns on gas compressible flow.

SET **flow** **incompressible**

incompressible turns on the incompressible fluid, transient flow mode.

on/off turns the fluid flow on or off. If **SET flow compressible** and **bulk ν** $\neq 0$, then domain pressures will still change due to mechanical deformations.

steady turns on steady-state flow mode. This algorithm provides faster convergence than the standard transient flow calculation. The command should be used in analyses in which only the steady-state condition (and not the transient response) is required.

twophase turns on two-phase flow.

fluid_dt_type ν

controls how the fluid flow timestep is calculated:

= 1 Fluid timestep is calculated based on the minimum domain volume and the maximum aperture allowed. This will result in the most conservative (smallest) fluid timestep.

= 2 Fluid timestep is calculated based on the smallest actual domain volume and the largest actual aperture currently in the model. This is the current default and results in a timestep that is usually bigger than 1 (above), but never smaller

= 3 Fluid flow timestep will be calculated for each domain using the domain volume and the apertures associated with the domain. The smallest calculated timestep for all domains is used. This may result in a significant increase in the fluid flow timestep.

fobl ν

sets the limit for the maximum out-of-balance force that will allow the timestep in creep calculations to be increased (requires the **SET crdt auto** command). The default value is 1e4.

fobu ν

This parameter is used for controlling creep timestep and fracture fluid-flow stepping. For creep, this sets the limit for the maximum out-of-balance force at which the timestep in creep calculations may be decreased (requires the **SET crdt auto** command). For fracture fluid flow, this sets the limit in compressible flow calculations. This is used to stop the mechanical iterations within a single fluid-flow timestep prior to reaching **nmech**. The default value is 1e5.

SET freeze_con

freeze_con **on**
 off

If **freeze_con** is **on**, *UDEC* will not create or delete any contacts. This may be useful for some fluid flow or thermal models. It is recommended that the model be in static equilibrium prior to issuing this command. The default is **off**.

ftime *tf*

Initial fluid-flow time is set to *tf* for transient flow analysis.

gas **on**
 off

If **off** (which is the default), the bulk modulus of non-wetting fluid is pressure independent and is equal to **nwbulk**. If **on**, the bulk modulus of non-wetting fluid is equal to pressure in non-wetting fluid multiplied with the coefficient specified in **nwbulk**.

gliopath specifies the path to the *GLIC* folder. This is usually unnecessary.

gpeq *igp1 igp2*

locks gridpoints *igp1* and *igp2* together with equal velocities. The total number of locked gridpoints is 20.

gpneg *igp1 igp2*

locks gridpoints *igp1* and *igp2* to have equal but opposite velocities. The total number of locked gridpoints is 20.

gravity *gx gy*

Gravitational accelerations are set for the *x*- and *y*-directions.

NOTE: If fluid flow is to be included, set **gx** = 0 and **gy** = **-g**.

j5flow **on**
 off

switches fluid flow **on/off** in unfractured **jcons** = 5 joint segments (default = **off**).

jcondf *n*
 <name . . . properties>

default constitutive relation for new contacts (created after cycle 1). If **jcondf** is not specified, it is assumed equal to 2. If the name of a “jmodel” or “joint model” is specified, the property keywords and values must also be included.

SET **jmatdf**

jmatdf	<i>n</i>	default material number for new contacts (created after cycle 1). If jmatdf is not specified, it is assumed equal to the block material number on one side of the joint. Also see SET jmtable .
jmtable	<i>mat 1 mat 2 jmat jcons</i>	defines a table of property and constitutive models for contacts. <i>mat 1</i> and <i>mat 2</i> are property numbers of blocks in contact. <i>jmat</i> and <i>jcons</i> are the material and constitutive models that will be assigned to the contact. This is an alternative to jcondf and jmatdf .
join_jkn		joint normal stiffness used for joined or glued joints (default is 10 times stiffness of adjacent zones or other contacts)
join_jks		joint shear stiffness used for joined or glued contacts (default is 10 times stiffness of adjacent zones or other contacts)
join_ratio		sets multiplier used in calculating joined joints from adjacent zones. (default is 10)
jointid	<i>n</i>	sets the starting number for the joint plane ID numbers.
jrepath		specifies the path to the Java runtime library folder. This is usually unnecessary.
latency	<i>n</i>	controls the number of creep timesteps that the maximum out-of-balance force must be above or below the set limits (fobl or fobu) before the timestep is changed. This prevents spikes from changing the timestep (requires the SET crdt auto command). The default value is 100.
legend	on off	If off , plots appear on-screen without titles and other data. (Default switch for legend is on .)
lmul	<i>v</i>	sets the multiplication factor used to increase the creep timestep if the maximum out-of-balance force drops below fobl for more than latency timesteps (requires SET crdt auto). The default value is 2.0.

SET

log

log

on
off

filename

on opens an ASCII file named “UDEEC.LOG” on the default disk drive. In interactive mode, if a “UDEEC.LOG” file already exists, there is an option to overwrite or append to the existing file. Any text that is printed to the screen from this point on 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. If **SET log on** is given at some later stage in the session, subsequent screen output can be appended to the file, or existing text can be overwritten.

The log filename can also be specified directly by the user. The log file is turned **on** automatically when the file name is specified.

maxdt

v

sets the upper limit for the creep timestep if **SET crdt auto** is used. The default value is 1e4.

maxgiicmem *n*

specifies amount of memory (in MB) to be set aside for the graphical interface (*GIIC*). This value is stored in the registry, so the command is only needed to change the amount. (default = 200 MB)

maxmech *n*

maximum number of mechanical relaxation steps performed within a fluid cycle for **SET flow incompressible** (default *n* = 500)

mech

on
off

causes *UDEEC* to do mechanical calculation (**SET mech on**) or suppress mechanical calculation (**SET mech off**). The **mech on** option is the default when starting *UDEEC*. This command is normally used for mechanical calculations in conjunction with fluid flow or thermal calculations.

memory *n*

sets the amount of memory (in MB) available in *UDEEC*. This value will be stored in the registry, and will be used for all future *UDEEC* instances. (*UDEEC* must be restarted for this to take effect.)

SET **message**

message **off**
on

causes information messages during stepping to be enabled or suppressed (default is **on**). It is useful to turn this messaging **off** when issuing *UDEC* commands from a *FISH* function.

mindt *v*

sets the lower limit for the creep timestep if **SET crdt auto** is used. The default value is 1e2.

multi **off**
on

controls whether calculations are multithreaded. The default is **on**.

netserver *name*

specifies the name of the network server for network key versions.

nflow *n*

n fluid-flow timesteps are performed during each cycle (default: *n* = 1) for **SET flow compressible**. (See [Section 2](#) in **Special Features**.)

nfmech *n*

maximum number of mechanical steps executed between fluid-flow steps when **nflow** is nonzero for **SET flow compressible** (see above) (default: *n* = 1)

nmech *n*

maximum number of mechanical steps executed between thermal steps when **nther** is nonzero (see below). The default value is *n* = 500. (See [Section 3](#) in **Special Features**.)

nodal **off**
on

sets nodal mixed discretization **on**. This will improve the accuracy of plasticity in some cases.

nowhite **off**
on

controls whether a white contour is ever used in a grayscale plot. The default is **off**.

SET **npens****npens** *n*

sets maximum number of pens for pen plots. If selected color exceeds maximum number of pens, then various dashed-line patterns will be used. The default value of **npens** is 6.

nther *n*

number of thermal steps to do before switching to mechanical steps. (See [Section 3](#) in **Special Features**.)

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 (**RUN temp = value**) is violated. If the temperature change parameter is violated when **nther = 0**, thermal cycling stops, and further thermal or mechanical cycling is controlled by the user.

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 upon return to thermal steps. If large geometry changes occur, it is better to divide the run into several **RUN** commands than to have only one.

oorc *n*

sets the color to be used to plot color-scaled objects if they are outside of the specified minimum and maximum values. (default = background color)

output name

sends plotted output to the device connected to port where port can be **com1**, **com2**, **lpt1** or any other port. Alternatively, port can be the name of a disk file. In this case, the plot output will be stored in file port (see the **SET plot** command). The default is **SET output UDEC.EPS**.

overwrite **off**
 on

controls whether the files generated using the **autoname** feature may replace files that already exist. If **overwrite** is **on**, files will be overwritten. The default setting is **off**.

ovtol *ovtol*

Contact overlap tolerance is set to *ovtol* (default: *ovtol* = one-half rounding length).

SET **paginate**

paginate **on** < *n* >
off

If **off**, text pagination on the screen is turned off. The number of lines printed per page can be changed with the optional variable *n*. The defaults are **on** and *n* = 23.

pline <*xb yb xe ye n*> <**off**>

A line through the model is defined for plotting specific problem variables for a deformable block model (see **PLOT pline**). The line is defined by the endpoints (*xb,yb*) and (*xe,ye*). The line is divided into *n* points. No interpolation is made – the value of the nearest data point to the search point on the **pline** is selected.

By typing **SET pline** without the parameters, a list of the current plot lines will be printed. If the keyword **off** is specified, the **pline** list is deleted.

plot <keyword>

Controls the type of graphics hardcopy output. (See the **PLOT pen** or **COPY** command.) Hardcopy plots are formatted slightly differently than screen plots, with the assumption that they will be printed on paper. To get actual screen images, see the **SET pcx on** command. There are several output types, including Windows printer, Windows clipboard, Windows bitmap (BMP), Windows enhanced metafile (EMF), AutoCad data exchange format (DXF), PostScript, PCX and JPEG. The default hardcopy type is a PostScript file, and the default output path is a file named **UDEC.EPS**. The type, and settings for each type, are specified with the following keywords.

background *n*

allows the user to redefine the background color for hardcopy plots. The input number can have any value. Values 0 to 16 correspond to named colors. The color can also be specified by one of several names: black, blue, green, cyan, red, brown, magenta, white, gray, lblue, lgreen, lcyan, lred, yellow, lmagenta and iwhite. The default is iwhite (15). This setting does not affect the background color for screen plots.

bmp keyword

The graphics hardcopy output will be formatted as a Windows bitmap file (BMP). The settings can be modified with the following optional keywords.

SET	plot	bmp	bw
			<p>bw The hardcopy image will be in grayscale. The default for bmp is color.</p> <p>color The hardcopy image will be in color. This is the default setting.</p> <p>grayscale The hardcopy image will be in grayscale (same as bw). The default for bmp is color.</p> <p>size iw ih sets the image size for the BMP file to be iw by ih pixels. By default, iw and ih are 800 by 600.</p>
		clipboard	<p>keyword</p> <p>The graphics hardcopy output will be formatted as a Windows clipboard. The settings can be modified with the following optional keywords.</p> <p>bw The hardcopy image will be in grayscale. The default for clipboard is color.</p> <p>color The hardcopy image will be in color. This is the default setting.</p> <p>grayscale The hardcopy image will be in grayscale (same as bw). The default for clipboard is color.</p> <p>size iw ih sets the image size for the clipboard image to be iw by ih pixels. By default, iw and ih are 800 by 600.</p>
		dxf	<p>n</p> <p>AutoCAD DXF format. n = 16 if file is for 16 color AutoCAD format n = 25 if file is for 256 color AutoCAD format</p>
		emf	<p>keyword</p> <p>The graphics hardcopy output will be a Windows enhanced metafile (EMF). The settings can be modified with the following optional keywords.</p> <p>bw The hardcopy image will be in grayscale. The default for emf is color.</p> <p>color The hardcopy image will be in color. This is the default setting.</p>

SET	plot	emf	grayscale
			<p>grayscale The hardcopy image will be in grayscale (same as bw). The default for emf is color.</p> <p>size iw ih sets the image size for the EMF file to be iw by ih pixels. By default, iw and ih are 800 by 600.</p>
		filename	<p>name</p> <p>sets the name of the file used for hardcopy output. The default name is "UDEDEC.EPS." File-type extensions are not automatically provided, and must be specified for each output type. For output types Windows printer and Windows clipboard, this setting is ignored. The specified name can also be a device port such as com1, com2 or lpt1. However, if a port is specified and no device is present, or if the output format is inappropriate for that device, <i>UDEDEC</i> may hang up. Note that if an existing file is present, the user must decide whether to overwrite or append. If a data file is being processed, <i>UDEDEC</i> will automatically overwrite old files.</p>
		jpg	<p>keyword</p> <p>The graphics hardcopy output will be formatted as a JPEG file. Loss-less compression is used for JPEG files in <i>UDEDEC</i>. The settings can be modified with the following optional keywords.</p> <p>bw The hardcopy image will be in grayscale. The default for jpg is color.</p> <p>color The hardcopy image will be in color. This is the default setting.</p> <p>grayscale The hardcopy image will be in grayscale (same as bw). The default for jpg is color.</p> <p>size iw ih sets the image size for the JPEG file to be iw by ih pixels. By default, iw and ih are 800 by 600.</p>
		png	<p>keyword</p> <p>The graphics hardcopy output will be formatted as a PNG file. Loss-less compression is used for PNG files in <i>UDEDEC</i>. The settings can be modified with the following optional keywords.</p>

SET	plot	png	bw
			<p>bw The hardcopy image will be in grayscale. The default for png is color.</p> <p>color The hardcopy image will be in color. This is the default setting.</p> <p>grayscale The hardcopy image will be in grayscale (same as bw). The default for png is color.</p> <p>size iw ih sets the image size for the PNG file to be iw by ih pixels. By default, iw and ih are 800 by 600.</p>
		Postscript	<p><keyword> <<i>xshift yshift xscale yscale</i>> <<i>rot</i>></p> <p>The graphics hardcopy output will be formatted as a PostScript-compatible printer file. The following optional parameters to shift and rotate the plot may be used.</p> <p><i>xshift yshift</i> shifts the plot in <i>x</i>- and <i>y</i>-directions on the page; units are in inches; default values are zero. Note: PostScript files are ASCII format and may be edited with any text editor (internally, PostScript units are in points, 1/72").</p> <p><i>xscale yscale</i> reduce or enlarge the plot; default values are 1.0.</p> <p><i>rot</i> is the rotation in degrees; the default is landscape (90°).</p> <p>a_us sets format for US paper size, 8.5 in × 11 in. This is the default.</p> <p>a3 sets format for A3 paper size, 420 mm × 297 mm. The default is a_us.</p> <p>a4 sets format for A4 paper size, 210 mm × 297 mm. The default is a_us.</p> <p>The settings can also be modified with the following optional keywords.</p> <p>bw The hardcopy image will be in grayscale. This is the default setting for PostScript.</p> <p>color The hardcopy image will be in color. The default is grayscale.</p>

SET **plot** **Postscript** **grayscale**

grayscale The hardcopy image will be in grayscale. This is the default setting for PostScript.

printer <keyword> <*xshift yshift xscale yscale*>

The graphics hardcopy output will be sent to the current default Windows printer. The size of the plot should adjust to the size of paper specified in the printer properties. However, the following optional parameters to shift and rotate the plot also may be used.

xshift yshift shifts the plot in *x*- and *y*-directions on the page; units are in inches; default values are zero.

xscale yscale reduce or enlarge the plot; default values are 1.0.

The settings can also be modified with the following optional keywords.

bw The hardcopy image will be in grayscale. The default is **color**.

color The hardcopy image will be in color. This is the default setting.

grayscale The hardcopy image will be in grayscale (same as **bw**). The default is **color**.

png keyword

sets the PNG output mode. The following keywords are available.

filename filename

sets the file name for PNG output generated either after **SET png on** is invoked, or from the <F2> keystroke in graphics screen mode. (See **SET autaname**.)

The default file name is “UDEC.PNG.”

off
on

PNG output mode is turned **off** or **on**. When **on**, a new PNG format image is put into the file specified by the **SET png file filename** command. The default file name is “UDEC.PNG.” The new image will replace the old image. If **png** is **off**, no image is stored. The <F2> key may be pressed to generate a new image while in plot mode, even if **png** is **off**. (See **SET autaname**.)

SET**step****step** *n*

a new PNG plot file is created every *n* cycles (the default for this option is **off**; the default for *n* is 1000).

pngfile filename

sets the file name for PNG output generated either after **SET png on** is invoked, or from the <F2> keystroke in graphics screen mode. The default file name is “UDEDEC.PNG.”

processors *n*

sets the number of processors to use for multithreaded calculations. The default is the maximum number available.

propcontour **on**
off

forces plotting of individual property values to use value contouring rather than plot discrete values. Plotting will switch to this mode if there are more than 12 individual values of the property. The default is **off**.

psrot *ang*

rotates PostScript plots *ang* degree (default = 90°).

psxscal *x*

scales the PostScript plot by an amount *x* in the *x*-direction. (Default is *x* = 1.0.)

psxshift *x*

shifts PostScript plot by an amount *x* (*x* in inches) in the *x*-direction. (Default is *x* = 0.)

psyscal *y*

scales the PostScript plot by an amount *y* in the *y*-direction. (Default is *y* = 1.0.)

psyshift *y*

shifts PostScript plot by an amount *y* (*y* in inches) in the *y*-direction. (Default is *y* = 0.)

SET **ptol**

ptol p

domain pressure variation tolerance for fluid flow, steady-state (**SET flow steady**) calculation. (Default is 0.01.)

quitonerror **on**
off

controls how *UDEC* responds to an internal error condition. If this is set **on**, *UDEC* will quit when an error is found. This is useful when running sequential models from a batch file. The default is **off**.

radi x,y

changes xx -stress to a radial stress about center at x,y .

random v

reset random number generator to beginning. This will allow an identical sequence to be repeated. The value (if specified) is the random number seed. This allows a new sequence to be initiated.

reftime **flow**
mech

reference time used in load, velocity and pressure histories for the **BOUNDARY** or **PFIX** command (default = **mech**)

replot **on**
off

If **replot** is on, the current screen plot will be re-drawn when a new window is defined using the zoom feature in the graphics screen (default = **on**).

rot α

rotates zone stress. Angle of rotation is α .

satmax s

If the saturation of a domain is larger than **satmax**, the domain is considered to be fully saturated in two-phase simulations.

satmin s

is the minimum, irreducible saturation of a domain by the wetting fluid in a two-phase calculation. This limit is used because the capillary pressure curve becomes singular for wetting fluid saturation equal to zero.

SET	sgrid	
	sgrid	<p><i>nx ny</i></p> <p>partitions the window into <i>nx</i> by <i>ny</i> for sampled stress contouring. This command changes the grid for all plots. The PLOT grid nx ny command affects the current plot only. For best contour detail, the grid should approximate the zone sizes. (See '503S1052' in the User's Guide.) The default is <i>nx</i> = 50 <i>ny</i> = 50.</p>
	skip_error	<p>on off</p> <p>If set on, <i>UDEC</i> will continue operating after some errors. This is mostly used when using <i>FISH</i> to batch-process multiple parameter runs.</p>
	small	<p>on off</p> <p>sets small strain mode in zones. Gridpoint locations will not be updated due to strain calculation. (Default is off.)</p>
	square	<p>off on</p> <p>Forces <i>UDEC</i> to preserve aspect ratio when zooming (default = on).</p>
	stcbmat	<p><i>i</i></p> <p>defines material property number for structural interface node at the location of a connecting cable element.</p>
	stld	<p>off</p> <p>turns on local damping for structural element nodes. The default is off.</p>
	sup_delete	<p>off on</p> <p>If on, structural supports will be automatically deleted if they fail in tension (default is off).</p>
	szz	<p><i>szz</i></p> <p>specifies out-of-plane stress component for plane-stress analysis (CONFIG p_stress) (default <i>szz</i> = 0.0).</p>
	text	<p>color</p> <p>sets color of screen plot text. (The default is green.) See Table 1.6 for descriptions of color keywords.</p>

SET **tfres**

tfres **on**
off

controls residual volume limitation in transient fluid flow. The default is **off**.

thdt ***t***

defines the thermal timestep. By default, *UDEC* automatically calculates the thermal timestep for the explicit solution scheme. This keyword allows the user to choose a different timestep. If *UDEC* determines that the user-selected timestep is too large for numerical stability, the timestep will be reduced to a suitable value when thermal steps are taken. The calculation will not revert to the user-selected value until another **SET thdt** command is issued.

This command has the same effect as the **RUN delt** command (see [Section 3](#) in **Special Features**).

time ***t***

Initial mechanical time is set to ***t*** for mechanical analysis.

transparency **on**
off

changes colors in plots to be more suitable for transparency plotting.

twophase **on**
off

causes *UDEC* to turn two-phase pressure (if **flow** is **off**) and flow calculation on (**SET twophase on**) or off (**SET twophase off**). The **twophase off** option is the default when starting *UDEC*. Two-phase flow logic can be used with compressible flow (**SET flow compressible**) only. (See [Section 2](#) in **Special Features** for a detailed explanation of two-phase flow logic in *UDEC*.)

umul ***v***

sets the multiplication factor used to lower the creep timestep if the maximum out-of-balance force exceeds **fobu** for more than **latency** timesteps (requires **SET crdt auto**). The default value is 0.5.

upcon ***n***

updates contact locations only every ***n*** cycles. (Default is ***n* = 1.**)

SET **uplen****uplen** *n*

updates contact length only every *n* cycles (default = 100). The length is also updated every time the **CYCLE** command is issued.

voltol *v*

Mechanical stepping within a fluid cycle is stopped when the maximum ratio of unbalanced fluid volume to domain volume is below *v* for **SET flow incompressible** (default: *v* = 0.001).

write *i*

turns on verbose mode for debugging (for diagnostic use only).

zpres *pz range...*

sets pressure, *pz*, in zones of deformable blocks within a specified range (see [Section 1.1.3](#)). The pressure will obey histories defined by the **BOUNDARY** command.

SHOW

SHOW <range phrase> <keyword>

All hidden blocks with centroids within the range specified by the range phrase are made visible.

Only visible blocks may be deleted by the **DELETE** command. Only visible blocks may have their material or constitutive numbers changed by the **CHANGE** command, and group numbers assigned or changed by the **GROUP** command. Only visible blocks are made deformable with the **GENERATE** command. Only visible blocks can be cut with the **JSET**, **CRACK** or **SPLIT** command.

The range types are described in [Section 1.1.3](#). If no range is specified, all hidden blocks are made visible. The following keywords are available.

area	v
	marks as visible blocks that have areas less than the specified value.
deformable	marks as visible blocks that are fully deformable.
rigid	marks as visible blocks that are rigid.

SOLVE

SOLVE keyword *value* <keyword *value*> ...

This command enables the automatic detection of the steady-state solution for mechanical problems. A calculation is performed until the limiting conditions, as defined by the following keywords, are reached. **SOLVE force** must be used for rigid block models. The **SOLVE** command does not monitor fluid flow. It is possible that mechanical equilibrium may be reached prior to fluid-flow equilibrium for steady-state flow. **CYCLE ftime** should be used for transient fluid flow.

age *t*

t is the total problem time. The model will cycle until the total problem time equals *t*. If creep is active, creep time is used for the limit. Otherwise, mechanical time is used.

clock *t*

computer runtime limit, *in minutes*. A time limit of greater than 1440 minutes (24 hours) will *not* be accepted; for longer runtimes (e.g., over a weekend), several **SOLVE clock** commands can be given in sequence. (default is *t* = 1440 minutes)

continue

The execution of an interrupted (by pressing <Esc>) **SOLVE** command will continue from the current cycle number. Any limits previously set will still be active. If the **SOLVE** command was originally issued from within a data file, the file will continue to be read after the **SOLVE** limit is reached.

cycle *s*

cycle limit. (The default is 100,000 cycles.)

elastic <only>

sets joints and zone constitutive models to infinite strength for initial equilibrium cycling. This command does not reset previously established plasticity flags. However, plastic deformation will not occur while in effect. This command can be used for built-in, zone and joint models. It is not currently implemented in the zone or jmodel DLL models. This command does an initial solve with infinite strength, and then solves again with strengths returned to previous values. To do only the infinite strength step, use **SOLVE elastic only**.

force *f*

out-of-balance force limit. (default is *f* = 100)

fos

<keyword>

determines model minimum factor of safety. (See [Section 3.10.5](#) in the **User's Guide**.)

associated for block plasticity (default: nonassociated)

bracket *v1 v2*

sets two starting bracket values. FOS calculation stops with a warning message reported if it turns out that the FOS falls outside of the specified brackets. If *v1* = *v2*, *UDEC* tests one value for stability.

cycles sets limit to maximum number of cycles (default = 50,000)

file file name for mode of failure state (default: "FOS-MODE.FSV")

exclude keyword

include keyword

The following keywords are available.

cohesion (default: include)

friction (default: include)

jcohesion (default: include)

jfriction (default: include)

jmat *n*

joints of material *n* (default: include all)

jtension (default: exclude)

mat *n*

blocks of material *n* (default: include all)

structure (default: exclude)

tension (default: exclude)

ucohesion ubiquitous joint cohesion
(default: include)

ufriction ubiquitous joint friction (default: include)

SOLVE fos include utension

	utension	ubiquitous joint tension (default: exclude)
ncharres	nc	sets the characteristic response steps, nc . If not specified, nc is tested using the existing perturbation method.
no_restore		Use of this keyword prevents <i>UDEC</i> from loading the original strength model at the end of the bracketing phase. The current state will be the state as of the last bracket increment.
perturb	v	sets stress perturbation factor used in determining the characteristic response steps. (default is 2.0)
ratio	v	resets unbalanced force ratio used to determine equilibrium. If this is not explicitly set, <i>UDEC</i> uses the current value used by the previous SOLVE command. (default is 10^{-5})
resolution	v	sets the resolution in comparing two bracketing (stable and unstable) factors. (The default is $.005 \times$ mean value.)
noage		turns off age limit for the SOLVE command.
r_type	keyword	<p>The ratio limit for mechanical calculations can be calculated in three different ways, as defined by the following keywords.</p> <p>average The ratio is defined to be the average unbalanced mechanical force magnitude divided by the average applied mechanical force magnitude for all block centroids or gridpoints in the model (default).</p> <p>local The ratio is defined to be the maximum value of the unbalanced mechanical force magnitude divided by the applied mechanical force magnitude for all block centroids or gridpoints in the model.</p>

SOLVE**r_type****maximum****maximum**

The ratio is defined to be the maximum unbalanced mechanical force magnitude for all block centroids or gridpoints in the model divided by the average applied mechanical force magnitude for all the block centroids or gridpoints.

ratio**value**

ratio limit for the mechanical calculation process. **ratio** is the default limit, and the default value is 10^{-5} .

relax**x y <keyword>**

SOLVE relax is used to slowly reduce the forces on the inside of an excavation, to avoid a large tensile stress wave. This tensile stress wave may cause dynamic failure in zones that would not normally fail under the static stresses caused by the excavation. **SOLVE relax** can also be used to reduce the boundary forces on the internal boundary of an excavation down to a prescribed level, to simulate the 3D effect of a tunnel advance. **x** and **y** are the coordinates of a point inside the excavation. The internal boundary of the excavation must not intersect the edge of the model. The sequence used by **relax** is

1. Calculate the reaction forces on perimeter of an excavation.
2. Ramp down reaction forces in a prescribed manner.
3. Solve to equilibrium using the **SOLVE ratio** value that was in effect prior to the **SOLVE relax** command.

SOLVE relax will generate a “ground reaction” table if a table number and a history are supplied. The **y** value in the table will be the reduction factor, and the **x** value will be a specified history value. For example, a history of the crown **y**-displacement may be used. The following keywords are available.

cycles**n**

Use a specified number of cycles in each reduction step. (default = **SOLVE ratio**)

end_factor**v**

last reduction factor (default = 0.0)

ftable**n**

Use reduction factors from table **n**. The number of factors should match the number of force reduction steps.

SOLVE **relax** **grhist**

grhist	<i>n</i>	history to be used as the <i>x</i> value for the “ground reaction” table generated
grtable	<i>n</i>	table number to be used for the “ground reaction” table, where the displacements are obtained from the selected grhist history number.
nsteps	<i>n</i>	the number of force reduction steps to use (default = 10)
rtable	<i>n</i>	Use the values in the specified table as the solve ratio for each reduction step. The number of ratios in the table should match <i>n</i> steps.
sratio	<i>v</i>	solve ratio to be used in each of the reduction steps.

step ***s***

cycle limit. (The default is *s* = 100,000 cycles.)

NOTE: Once the limits have been defined (including default values), they remain in effect until specifically reset in a subsequent **SOLVE** command, or until a **NEW** command is used. The **NEW** command resets limits to their default values. If the <Esc> key is pressed during execution, *UDEC* returns control to the user after the current step is completed. If stepping is under control of a *FISH* function, then it is necessary to press <Esc> two times to stop both stepping and *FISH* execution.

The objective of the **SOLVE elastic** scheme is to get an initial state that is both in equilibrium and not actively failing. Usually we have no knowledge of how an existing system got to its present state unless we know, and can reproduce, the complete history (e.g., excavation of a slope or tectonic loading of a geologic system). Thus, any scheme that satisfies the two conditions listed above is valid, given a lack of knowledge of history. However, since the process is arbitrary, the plastic strains accumulated during the process of arriving at the initial state have no significance. The approach of switching on plasticity after an initial elastic solution will tend to minimize the region that goes plastic, which should lead to a more uniform initial stress state. The plastic strains accumulated in getting to this solution have no significance. This also applies to the displacements, which should be set to zero (in general) before proceeding.

SPLIT

SPLIT *x1, y1 x2, y2*

Cuts blocks along a line from *x1,y1* to *x2,y2*. It is similar to the **CRACK** command, except that blocks must be entirely cut by the line. Partial cuts are not retained for future possible intersections as in the case of the **CRACK** command.

STEP

STEP***n***

<keyword>

Executes ***n*** timesteps. (**STEP 0** is permitted as a check on data.) If the <Esc> key is pressed during execution, *UDEC* will return control to the user after the current cycle is completed. (Also see **CYCLE** and **SOLVE**.)

Alternatively, cycling can be controlled by specifying one of the following keywords.

continue This keyword causes the calculation to continue execution of the **STEP** command after interruption by use of the <Esc> key. The remaining cycles will be performed. If the **STEP** command was issued from a remote data file, the file will continue to be read after cycling is complete.

ftime***f***

If **SET flow incompressible** is invoked, fluid steps are executed. ***f*** is the flow duration time. **STEP *n*** can also be used for a flow calculation, in which ***n*** is the number of flow steps.

NOTES:

- (1) The fluid timestep must be set by the user (see the **SET dtflow** command).
- (2) During each flow step, a number of mechanical steps will be executed (see the **SET maxmech** command).
- (3) If the <Esc> key is pressed, the run is interrupted after the next fluid step is completed.

time***t***

t is problem-time duration in seconds for this increment of cycling.

STOP

STOP

UDEC stops executing. Note that *all* information generated while in *UDEC* will be lost unless a **SAVE** command was issued prior to the **STOP** command. The synonym **QUIT** may also be used.

STRESS

STRESS *xl xu yl yu nx ny*

In-plane principal stresses in the range $xl < x < xu$ and $yl < y < yu$ are printed on grid *nx,ny* (same as the **PRINT stress** command).

STRUCT

STRUCT keyword *value* <keyword *value*> ... <**range**...>

Structural (beam) elements are generated in internal boundary regions or along the outer boundary. Structural elements are created using keywords and values following the **generate** or **beam** keyword. Structural elements can only be placed on the perimeters of blocks; they cannot be placed inside of blocks.

When using the **generate** keyword option, the center of element generation is at (*xc,yc*), with the first node at an angle *fang* measured from the positive *x*-axis. The extent of the structural elements can be limited by specifying a total angle, *ang*, of the surface to receive the elements. *ang* is measured counterclockwise from the angle of the first node, *fang*. If an entire internal region is to receive elements, *fang* and *ang* do not need to be given. The number of structural nodes is given by *np*, the material number by *mat*, and the thickness of the structural elements by *th*.

When using the **beam** keyword option, the coordinates of the ends of each structural element are specified. The beam elements may be constructed by defining nodes and then connecting them with beam elements. Beams may also be constructed by giving the endpoints, either by coordinate pairs or by specifying a table containing the *xy*-pair coordinates.

The following keywords generate structural elements.

apply <keyword>

The logic used in solving for forces on structural element lining for tunnels is not capable of determining the fluid pressure difference between the inside and the outside of the lining. This command allows the user to specify this pressure difference so that the liner loads will include any hydrostatic forces. The following keywords are available.

pressure **p_inside p_outside**

applies pressure to structural elements in range.

p_inside pressure inside tunnel

p_outside pressure at rock liner interface

The net pressure (**p_inside-p_outside**) is applied to the structural element. It is assumed that the domain pressure, **p_outside**, is being applied to the rock.

STRUCT

apply

ref_loc

ref_loc

x y

reference location for pressure gradient

If gravity is active and the water density defined, a pressure gradient may be applied to **p_inside** and **p_outside**. The values **p_inside** and **p_outside** are taken as the pressures at the given reference location, and the hydrostatic gradient is superimposed. If the reference location is not given, no gradient is applied.

beam

begin node *n1* **end** node *n2* **mat** *mat*
begin *x1 y1* **end** *x2 y2* **mat** *mat*

creates a structural beam element. The beam element nodes may already exist, or may be created at the given location. The element will connect to preexisting elements. If the node is within the rounding length of a block edge, it places the node on the block edge.

mat*mat*

material number for element and interface contact (default: **mat** = 1)

beam

table *n* **mat** *mat*

creates a structural beam element. The beam element nodes are created at the locations given by the *xy*-pairs in table *n*. The beam elements will be connected to each other. If any node is within the rounding length of a block edge, it places the node on the block edge.

mat*mat*

material number for element and interface contact (default: **mat** = 1)

generate

xc *xc* **yc** *yc* **npoint** *np* **mat** *mat*
 <*fang fang*> <*theta ang*> <*min_length len*> <**connect**>
 <**smat** *n*> <**stol** *j*>

generate

begin *x,y* **end** *x,y* **mat** *mat*
 <**smat** *n*> <**connect**> <*min_length len*> <*max_length len*>

begin*xc yc*

location of first node on liner path

end*x y*

location of last node on liner path counterclockwise from first node

STRUCT generate fang

fang	<i>fang</i>
	angle of first node (default is 0°)
mat	<i>mat</i>
	element/interface material number
max_length	<i>len</i>
	maximum element length
min_length	<i>len</i>
	minimum element length (default is 25% of average element length)
npoint	<i>np</i>
	average number of nodes
theta	<i>ang</i>
	total angle (default is 360°)
xc	<i>xc</i>
	x-coordinate of point inside domain
yc	<i>yc</i>
	y-coordinate of point inside domain

The following keywords are available.

connect	If connect is given, the end nodes of the new structural element segment will attempt to connect to the endpoints of existing structural elements. The new endpoints must be located within the block rounding length of the existing endpoint in order to connect. The result is a single structure, and there will be a jump in the node ID numbers as the redundant node is deleted.
smat	<i>n</i>
	material number to assign the interface between two layers of structural elements. The default is mat .
stol	<i>v</i>
	controls distance that is used to attempt to connect ends between two structural elements if the connect keyword is used. The default is 2 × rounding length.

STRUCT generate

Note: The generation routine tries to support all edges around the opening, so the total number of nodes will, in general, be larger than the given **npoint**. By increasing **elemen**, small structural elements are not generated. However, small blocks may become unsupported.

node

n <keyword> **x y** <Dist **d**> <**Mat mat**>

creates a beam element node with ID **nid** at location (**x,y**). If the node is within a distance **d** of a block edge, it places the node on the block edge (default: **d** = rounding length).

fix < **x** > < **y** > < **r** >

fixes node in the **x, y** or rotation direction.

free < **x** > < **y** > < **r** >

frees node in the **x, y** or rotation direction.

initial keyword

assigns initial values to nodes.

rvel *v*

initial rotational value

xdis *v*

initial x-displacement value

xvel *v*

initial x-velocity value

ydis *v*

initial y-displacement value

yvel *v*

initial y-velocity value

load *fx fy m*

specifies nodal applied force and moment.

mat **mat**

material number for element and interface contact (default: **mat** = 1)

STRUCT node pin

pin fixes node in translational space.

slave **<x> <y> m**
 slaves specified node to node **m** in the **x** or **y** direction.

unslave **<x> <y>**
 frees the specified node from the slave condition in the **x** or **y** direction.

Certain parameters for structural elements are specified or changed using keywords and values following the **change** keyword. These parameters can assign different conditions and properties to different segments of the structural elements. The following keywords change element or interface properties (within an optional range).

change keyword

cons **i**
 changes constitutive model for structural element (**i** = 1, element is always elastic; **i** = 2, element may yield; default = 2).

jcons **i**
 changes constitutive model for structural element interfaces (default = 5).

jmat **n**
 interface material number

mat **n**
 element material number

The **PROPERTY** command assigns properties for the structural element material and interface material.

Structural element segments can be deleted with the following keyword.

delete Structural element segments are deleted. An optional range can be given; the control of the element segment must fall within the range to delete the segment.

STRUCT **prop *n*****prop *n***

keyword

specifies values for the following rockbolt properties.

area *v*

cross-sectional area (rather than radius)

cs_ncoh *v*

cohesion strength of the rock coupling spring in the normal direction

cs_nfric *v*

frictional value of the rock coupling spring in the normal direction

cs_nstiff *v*

stiffness of the rock coupling spring in the normal direction

cs_scoh *v*

cohesion strength of the rock coupling spring in the shear direction

cs_sfrc *v*

frictional value of the rock coupling spring in the shear direction

cs_sstiff *v*

stiffness of the rock coupling spring in the shear direction

cs_sctable *n*

table number for rock coupling spring shear-cohesion softening table

cs_sftable *n*

table relating confining stress factor to deviatoric stress

cs_cftable *n*

table number for rock contact spring normal-friction softening table

STRUCT	prop <i>n</i>	density
		density ν density of rockbolt element
	e	ν Young's modulus of rockbolt element
	i	ν moment of inertia of rockbolt element
	perimeter	ν length of perimeter of rockbolt in which rockbolt is inserted
	pmom	ν plastic moment
	radius	ν radius (rather than area) of rockbolt element
	spacing	ν out-of-plane spacing of rockbolt elements
	tfstrain	ν axial rupture strain limit
	thexp	ν thermal expansion coefficient
	ycomp	ν compressive yield stress of rockbolt element
	yield	ν tensile yield stress of rockbolt element

STRUCT **rockbolt****rockbolt**

begin *x1 y1* **end** *x2 y2* **prop** *n* **segment** *ns*
begin **node** *n1* **end** **node** *n2* **prop** *n* **segment** *ns*
delete *n1 n2*

creates rockbolt elements. If the **begin** *x,y* form is used, a string of *ns* elements is created. If the **begin node** form is used, a string of *ns* segments is created from **node** *n1* to **node** *n2*. The following keywords are available.

chprop*n***range***n1 n2*

changes property numbers for rockbolt located in range

hinge*n1 n2*

apply plastic hinges at nodes in range *n1* to *n2*

node *n**x y*

defines node *n* at *x,y*

node *n**

keyword

fix<**x**> <**y**> <**r**>

fixes node in the **x**, **y** or rotation direction

free<**x**> <**y**> <**r**>

frees node in the **x**, **y** or rotation direction

initial

<keyword>

assigns initial values to nodes.

xdis*v*

initial *x*-displacement value

xvel*v*

initial *x*-velocity value

ydis*v*

initial *y*-displacement value

STRUCT	rockbolt	node <i>n</i> *	initial	yvel
				yvel <i>v</i> initial y-velocity value
				rvel <i>v</i> initial rotational value
		load	<i>fx fy m</i> specifies nodal applied force and moment.	
		pin	fixes node in translational space.	
		slave	<i><x> <y> m</i> slaves specified node to node <i>m</i> in the x or y direction.	
		unslave	<i><x> <y></i> frees the specified node from the slave condition in the x or y direction.	
	prop	<i>n</i> property number for this rockbolt		

SUPPORT keyword *value* <keyword *value*> ...

When the **SUPPORT** command is given, *UDEC* searches for the two nearest block edges along the axis of a support member, and places the member between the two intersection points. An error will be detected if fewer than two block edges are found. The points of attachment to the blocks are preserved, whatever subsequent displacement occurs.

The support element location, geometry and nodal linkage are defined by the following values and keywords that directly follow the **SUPPORT** command.

x y <keyword>

x y is one point on the member. This point *must* be located in empty space. *x y* must immediately follow the **SUPPORT** command. The following keywords and values may be used after *x y* to define the support geometry or remove the support.

angle *a*

The axis of the support member is oriented at *a* degrees measured counterclockwise from the *x*-axis (the default is *a* = 90°).

delete

or

remove Either **remove** or **delete** causes the existing support member closest to *x y* to be deleted. The keyword must be the only one given in this case.

mat *np*

Material number *np* is assigned to the support element. The properties are associated with the property number by using the **PROPERTY** command. (The default is *np* = 1.)

segment *ns*

Support elements with nonzero width may be divided into *ns* segments. An odd number of segments is always placed. (The default is *ns* = 5.)

type name

The following names for support type are available.

standard standard formulation

SUPPORT $x\ y$ **type****rate_dep****rate_dep** rate-dependent formulation**width** **w**

The support member spans a width of **w** perpendicular to its axis. (The default is **$w = 0$** .) For example, if **width** = 10 m and **segments** = 5, support members are placed at 2.5 m spacing.

SYSTEM

SYSTEM The following DOS commands may be given following the **SYSTEM** command.

CD	directory string
COPY	file1 file2
DEL	file
DIR	<file-spec>
REN	file1 file2
TYPE	file

For example, the **SYSTEM** command is to issue the command with a DOS command on the same line.

```
sys dir *.dat
```

will list the files with the extension “DAT.”

CAUTION: Do not use these commands with files *UDEC* currently has open. If these commands attempt to access open files, the system may hang up, and the files may be lost.

TABLE

TABLE *n* <keyword> *x1,y1 x2,y2 <x3,y3> ...*

This command sets up a table of *x*- and *y*-values for use in *UDEC*. Tables are used to define:

- (1) load and velocity histories for the **BOUNDARY** command;
- (2) temperatures initialized in the model with the **TADD** command. **TABLE** must precede the **TADD** command;
- (3) colors for filled plots (see the **SET filcolor** command);
- (4) the variation of friction, cohesion, dilation, tensile strength and cap pressure with accumulated plastic strain for the strain-softening and double-yield models (see the **ZONE** command);
- (5) the variation in normal stress versus normal displacement for **jcons = 2** or **5** joint models (see the **PROPERTY nstable** command); and
- (6) the variation of fluid density or joint permeability with temperature (see the **FLUID dtable** and **PROPERTY ktable** commands).

Multiple tables may be defined, each uniquely identified by a table number, *n*. A maximum of 500 tables may be entered; the number of *x,y* pairs is only limited by the amount of computer memory available. The command **PRINT table** can be used to verify the contents of a table. Table numbers need not be sequential, but they must not be zero.

Optional keywords may be used to manipulate table contents:

delete	the table is completely deleted.
erase	erases all entries in table <i>n</i> without destroying the table.
insert	One or more <i>x,y</i> pairs can be added to a table at any time during an analysis. The new entries will be added to the end of the table unless the insert keyword is used. In this case, each new item is inserted between the two existing items that bracket the <i>x</i> -value of the new item. If an <i>x</i> -value for a new item is identical to that of an existing item, the existing item's <i>y</i> -value is updated (in insert mode).
read	filename read table from file (see BOUNDARY hread command for file format)

TABLE**Write**

Write***dx*** <filename>

writes table ***n*** to file filename. The parameter ***dx*** specifies the abscissa spacing for the data points. If ***dx*** is greater than zero, the data will be interpolated from the table ***n*** at a spacing of ***dx***. The file will consist of a single column of y-data at an even spacing of ***dx***. If ***dx*** = 0, the data will be the actual x,y pairs in table ***n***. If filename is not specified, the default is "UDEEC.TAB."

TADD

TADD *ntab xc,yc angl,ang2*

Temperatures can be incremented in an angular region using this command. The temperatures are taken from table *ntab* (see the **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° . The *x,y* pairs in table *ntab* 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 the *y*-values in the table to the current temperatures. The thermal stresses are also applied, based on these temperature changes.

NOTE: The **TABLE** command must precede the **TADD** command.

TFIX

TFIX *value* <**range**... >

The temperature at all corners and gridpoints in the range (see [Section 1.1.3](#)) 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

TFREE <range... >

The temperature at all corners and gridpoints in the range (see [Section 1.1.3](#)) is allowed to change during the simulation.

NOTE: By default, all temperatures are free to change initially.

THAPP

THAPP keyword = *value1*, *value2* <**range**...>

The **THAPP** command applies a thermal boundary condition to external boundaries and thermal sources to internal regions. An optional **range** can be specified to limit the range of **THAPP**. (See [Section 1.1.3.](#))

The following keywords are available.

convection *value1* is the convective heat transfer coefficient ($\text{w/m}^2 \text{ } ^\circ\text{C}$).
 value2 is the temperature of the medium to which convection occurs.
 A convective boundary condition is applied between *corners* within the range.

flux *value1* is the initial flux (watts/m^2).
 value2 is the decay constant (s^{-1}).
 A flux boundary condition is applied between *corners* within the range. If a flux is applied between two blocks, the specified flux will be applied to both blocks.

gpsource *value1* is the initial strength.
 value2 is the decay constant (s^{-1}).
 This keyword creates a thermal source at each of the gridpoints in the specified range. It differs from the **source** keyword in that it does not add a volume source to an entire block.

radiation *value1* is the radiative heat transfer coefficient (for black bodies, this is the Stefan-Boltzmann constant, $5.668 \times 10^{-8} \text{ w/m}^2 \text{ K}^4$).
 value2 is the temperature of the medium to which radiation occurs.
 A radiation boundary condition is applied between corners within the range.

source *value1* is the initial strength.
 value2 is the decay constant (s^{-1}).
 The **source** keyword results in a volume source of the stated strength in all blocks whose *centroids* are in the specified range. The user is responsible for determining the strength of the source for different size blocks. The initial strength to be given for each block is the intended power/volume ratio multiplied by the area of the block. The correct units for **source** are Watts/m, (cal/s)/cm or the British equivalents.

The decay constant in the **source** and **flux** options is defined by the equation

$$S_{curr} = S_{ini} \times \exp[c_d(t_{curr} - t_{ini})]$$

where S_{curr} = current strength;

S_{ini} = initial strength;

c_d = decay constant;

t_{curr} = current time; and

t_{ini} = initial time (when **THAPP** is invoked).

To remove a **convection** or **radiation** boundary condition, the same condition should be applied with the heat transfer coefficient of opposite sign.

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

$$S_{rep} = -S_{ini} \times \exp[c_d(t_{curr} - t_{ini})]$$

NOTE: Unless otherwise specified by the **THAPP** command, all boundaries are adiabatic (i.e., insulated).

TITLE

TITLE <'string'>

The title for a *UDEC* model is printed on subsequent output plots and recorded in the save files (same as the **TITLE** command).

If 'string' is present, then that token is parsed as a string and used as the current title. This token can be a *FISH* string variable; if it is, do not enter the token in single quotes. If 'string' is not present, then a `head>` prompt appears, and the next line input is taken as the title. If in interactive mode while this line is being read, simply pressing <ENTER> will retain the old title. To clear the title (causing the title window to disappear on output plots), give a null string (two consecutive single quotes) with the **TITLE** command:

```
title ''
```


TUNNEL

TUNNEL *xc yc r n* <**join**>

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 deformable.

Following execution of this command, future cracks or joints will not penetrate the tunnel periphery. To allow cracks to penetrate the periphery, specify a second tunnel with zero radius at some point outside the problem domain.

If the optional keyword **join** is given, any new contacts formed as a result of the command will be joined contacts.

VORONOI

VORONOI edge *l* <keyword> <range...>

Voronoi tessellation is used to create randomly sized polygonal blocks. One or more blocks in a *UDEEC* model can be subdivided into Voronoi sub-blocks of arbitrary size. A Voronoi tessellation is generated within the range specified. If no range is given, the entire model is covered by the tessellation. The range is typically given by a **jregion** keyword, referring to a previous **JREGION** command. (See [Section 1.1.3](#).) The time required to generate blocks (*n*) is proportional to n^2 .

The following keywords are available.

crack_store *v*

sets the distance from left edge of current crack generation that used cracks will be saved. Deleting saved cracks too far from current crack “face” saves time. However, if cracks are not being generated on the left side of blocks, this value may be increased (default = $2 \times \text{edge}$).

edge *l*

average edge length of the Voronoi polygons. This length must be given, and must be at least 20 times the rounding length.

iterations *n*

number of iterations in the relaxation process. The default value is 5.

round *v*

rounding length for generated blocks (the default is 0.5). This can also be set via the **ROUND** command.

rtol *v*

tolerance used for blocks. (This should be less than rounding length.) Default is $0.02 \times \text{round}$.

seed *n*

sets seed for random number generation used by the Voronoi tessellation.

trigon causes the creation of triangular blocks based on the Delaunay triangles normally used to generate the Voronoi blocks.

See [Section 3.2.2.2](#) in the **User’s Guide** for a description of the Voronoi generator.

WELL

WELL **domain** *n* **flow** *q***atdomain** *x, y* **flow** *q*

A fluid source with flow rate *q* is placed in domain *n*, or at a domain specified by *x, y*. (If *q* is negative, a sink is assumed.)

temp *v*

specifies temperature of fluid at well.

WINDOW

WINDOW *<xl xu yl yu>*

This command creates an imaginary window on the screen or plotter for the purpose 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 in which the *x*- and *y*-scales are different will be drawn; vectors will also be distorted. Window squaring can be turned off and on with the **SET square** command. The default is **on**.

If a **WINDOW** command is not defined prior to plotting, one that encompasses the entire grid is selected. The window will remain set until changed. If the window is smaller 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. The window can be restored to encompass the entire grid by typing **WINDOW** without the coordinate range.

The window can also be modified interactively, using the keyboard or mouse, while the graphics screen is displayed, by pressing <Z>. See the interactive plotting category for the **PLOT** command for additional details.

ZONE

ZONE <model name> <keyword *value* ... > ... <range... >
 <load filename>

This command associates a built-in constitutive model and/or one or more properties with all zones (of deformable blocks) that have centroids in the given range. The advantage of assigning models and properties with the **ZONE** command is that each zone has its own local properties.

This command is also used to assign user-defined models (UDM) to zones in *UDEC*. This command is also used to assign properties to these zones. The UDM option is required for user-defined models. Also, **CONFIG cppudm** must be used in order for the user-defined models to be accepted. Models exist as dynamic linked libraries and must be loaded prior to assignment. This is done with the **ZONE load filename** command. The models may then be assigned to zones via the **ZONE model name** command. Models that are placed in the “plugins\models” folder will be loaded automatically.

The built-in models are described in [Section 1](#) in **Constitutive Models**. Any block constitutive models that already exist for some or all of the zones in the given range (as specified by **CHANGE cons**) are ignored. The **model** keyword assigns constitutive models via the following names.

null	null model used for excavations
elastic	isotropic elastic model
anisotropic	transversely anisotropic elastic
drucker	Drucker-Prager plasticity model
mohr	Mohr-Coulomb plasticity model
ubiquitous	ubiquitous joint model
ss	strain softening plasticity model
subiquitous	bilinear strain softening ubiquitous joint model
dy	double yield plasticity model
cam-clay	modified cam-clay plasticity model
chsoil	modified cysoil plasticity model
cysoil	cap yield plasticity model
hoekbrown	generalized Hoek-Brown plasticity model
mhoek	modified Hoek-Brown plasticity model

The other keywords assign material properties and are model-specific. Other zone properties entered via the **PROPERTY** command are ignored if the range of zones falls within that specified by a **ZONE** command.

The property keywords and associated model type are presented in the following pages. The model must be defined *before* assigning properties. If properties that are not consistent with the chosen model are given, a warning message will be issued to inform the user that the unneeded properties were not accepted. Zone properties can

be printed with the **PRINT varz *property name*** command. Zone properties may be plotted with the **PLOT *property name*** command. In addition to the input properties, other properties calculated by the models can be printed or plotted. These are listed with the other model properties.

Creep models require the creep option. In addition, the **CONFIG creep** command must be used to access these models. The creep models follow.

viscous	classical viscous (Maxwell) model
burgers	Burgers viscoelastic model
power	two-component power creep model
wipp	WIPP reference creep model
cvisc	Burgers/Mohr visco-plastic model
cpow	power/Mohr visco-plastic model
pwipp	WIPP/Mohr visco-plastic model
cwipp	WIPP crushed salt model

WARNING: When a **model** is invoked over a specified range, all properties associated with that model are initialized to zero in that range. Properties previously assigned to that range must be specified again, even if their values have not changed.

A **model** can be assigned to regions of zones within a block for all of the above constitutive models *except* the null model. The null model must be assigned to an entire block.

The following thermal properties may be set using the **ZONE** command, but are not tied to any specific constitutive model.

cond	thermal conductivity
spec.heat	thermal specific heat (isotropic conductivity)
thexp	thermal expansion coefficient
xcond	thermal conductivity in the <i>x</i> -direction
ycond	thermal conductivity in the <i>y</i> -direction

The following fluid pore-pressure property may be set using the **ZONE** command, but is not tied to any specific constitutive model.

biot_coef	Biot's constant for porous elastic rocks
------------------	--

Null – **ZONE model null**

- (1) **bulk_mod** dummy* elastic bulk modulus, K
- (2) **density** mass density, ρ
- (3) **shear_mod** dummy* elastic shear modulus, G

* Moduli are required for the null model, to prevent the zones from collapsing. Internal stresses are calculated but not transferred to adjacent blocks. Stresses are automatically zeroed when changed back to another model for backfill simulations.

Elastic Mechanical Models

Isotropic Elastic – **ZONE model elastic**

- (1) **bulk_mod** elastic bulk modulus, K
- (2) **density** mass density, ρ
- (3) **shear_mod** elastic shear modulus, G

See [Section 1](#) in **Constitutive Models** for details.

Transversely Isotropic Elastic – ZONE model anisotropic

- | | |
|--------------------|---|
| (1) angle | orientation of plane of anisotropy taken counterclockwise from positive x -axis, θ |
| (2) density | mass density, ρ |
| (3) e1 | elastic Young's modulus in the plane of isotropy, E_1 |
| (4) e3 | elastic Young's modulus in the plane perpendicular to the plane of isotropy, E_2 |
| (5) g13 | elastic cross-shear modulus between plane of isotropy and perpendicular plane (i.e., x_1x_2 - or x_2x_3 -plane), G_{12} * |
| (6) nu12 | Poisson's ratio for the normal strain in the x_1 -direction (in the plane of isotropy) related to the normal strain in the x_2 -direction (in the perpendicular plane) due to uniaxial stress in the x_2 -direction, ν_{21} |
| (7) nu13 | Poisson's ratio for the normal strain in the x_1 -direction (in the plane of isotropy) related to the normal strain in the x_3 -direction due to uniaxial stress in the x_3 -direction, ν_{31} |

See [Section 1](#) in **Constitutive Models** for details.

* The cross-shear modulus, G_{12} , for anisotropic elasticity must be determined. S. G. Lekhnitskii (*Theory of Elasticity of an Anisotropic Body*. Moscow: Mir Publishers, 1981) suggests the following equation based on laboratory testing of rock.

$$G_{12} = \frac{E_1 E_2}{E_1 (1 + 2\nu_{12}) + E_2}$$

assuming the x_1x_3 -plane is the plane of isotropy.

Elastic-Plastic Mechanical Models

Drucker-Prager – ZONE model drucker

- (1) **bulk** elastic bulk modulus, K
- (2) **density** mass density, ρ
- (3) **kshear** material parameter, k_ϕ
- (4) **qdil** material parameter, q_ψ
- (5) **qvola** material parameter, q_ϕ
- (6) **shear** elastic shear modulus, G
- (7) **tension** tension limit, σ^t

Plasticity state indicator flags are given below. Use logical **and** to find individual and multiple state modes.

Bit Number	State
1	failure in shear now
2	failure in tension now
4	failure in shear in the past
8	failure in tension in the past

See [Section 1](#) in **Constitutive Models** for details.

Note that the default tension limit is zero for a material with $q_\phi = 0$, and is k_ϕ/q_ϕ otherwise. The value assigned for the tension limit remains constant when tensile failure occurs.

Mohr-Coulomb – **ZONE model mohr**

- (1) **bulk_mod** elastic bulk modulus, K
- (2) **cohesion** cohesion, c
- (3) **density** mass density, ρ
- (4) **dilation** dilation angle, ψ
- (5) **friction** internal angle of friction, ϕ
- (6) **shear_mod** elastic shear modulus, G
- (7) **tension** tension limit, σ^t

The following property can be printed, plotted or accessed via *FISH*.

- (1) **state** plastic state

Plasticity state indicator flags are given below. Use logical **and** to find individual and multiple state modes.

Bit Number	State
1	failure in shear now
2	failure in tension now
4	failure in shear in the past
8	failure in tension in the past

See [Section 1](#) in **Constitutive Models** for details.

Note that the default tension limit is zero for a material with no friction, and is $c/\tan\phi$ otherwise. The value assigned for the tension limit remains constant when tensile failure occurs.

Ubiquitous-Joint – ZONE model ubiquitous

- (1) **bulk_mod** elastic bulk modulus, K
- (2) **cohesion** cohesion of matrix, c
- (3) **density** mass density, ρ
- (4) **dilation** dilation angle of matrix, ψ
- (5) **friction** internal angle of friction of matrix, ϕ
- (6) **jangle** joint angle taken counterclockwise from the x -axis, θ
- (7) **shear_mod** elastic shear modulus, G
- (8) **tension** tension limit of matrix, σ^t
- (9) **ucohesion** joint cohesion, c_j
- (10) **udilation** joint dilation angle, ψ_j
- (11) **ufriiction** joint friction angle, ϕ_j
- (12) **utension** joint tension limit, σ_j^t

The following property can be printed, plotted or accessed via *FISH*.

- (1) **state** plastic state

Plasticity state indicator flags are given below. Use logical **and** to find individual and multiple state modes.

Bit Number	State
1	matrix failure in shear now
2	matrix failure in tension now
4	matrix failure in shear in the past
8	matrix failure in tension in the past
16	joint failure in shear now
32	joint failure in tension now
64	joint failure in shear in the past
128	joint failure in tension in the past

See [Section 1](#) in **Constitutive Models** for details.

Note that the default tension limit of the matrix, σ^t , is the same as that for the Mohr-Coulomb model. The default joint tension limit, σ_j^t , is zero if $\phi_j = 0$, and is $c_j / \tan \phi_j$ otherwise. The values assigned for σ^t and σ_j^t remain constant when tensile failure occurs in the matrix or on the weakness plane.

Strain-Hardening/Softening – ZONE model ss

- (1) **bulk_mod** elastic bulk modulus, K
- (2) **cohesion** cohesion, c
- (3) **ctable** number of table relating cohesion to plastic shear strain
- (4) **density** mass density, ρ
- (5) **dilation** dilation angle, ψ
- (6) **dtable** number of table relating dilation angle to plastic shear strain
- (7) **friction** angle of internal friction, ϕ
- (8) **ftable** number of table relating friction angle to plastic shear strain
- (9) **shear_mod** elastic shear modulus, G
- (10) **tension** tension limit, σ^t
- (11) **ttable** number of table relating tension limit to plastic tensile strain

The following calculated properties can be printed, plotted or accessed via *FISH*.

- (1) **e_plastic** accumulated plastic shear strain
- (2) **e_tension** accumulated plastic tensile strain
- (3) **state** plastic state

Plasticity state indicator flags are given below. Use logical **and** to find individual and multiple state modes.

Bit Number	State
1	failure in shear now
2	failure in tension now
4	failure in shear in the past
8	failure in tension in the past

The strain-hardening and -softening behavior is controlled by the variation in friction, cohesion and dilation as a function of plastic shear strain given by a specified table of values. Variation of tensile strength as a function of plastic tensile strain is also specified by a table. Note that if table numbers are given as 0 (default), the properties will take the values given (i.e., with the **cohesion**, **dilation**, **friction** or **tension** keyword).

See [Section 1](#) in **Constitutive Models** for details.

Bilinear, Strain-Hardening/Softening Ubiquitous-Joint – ZONE model subiquitous

- | | |
|----------------------|---|
| (1) bijoint | = 0 for joint linear model (default);
= 1 for joint bilinear model |
| (2) bimatrix | = 0 for matrix linear model (default);
= 1 for matrix bilinear model |
| (3) bulk | elastic bulk modulus, K |
| (4) c2table | number of table relating matrix cohesion c_2 to matrix plastic shear strain |
| (5) cj2table | number of table relating joint cohesion c_{j2} to joint plastic shear strain |
| (6) cjtable | number of table relating joint cohesion c_{j1} to joint plastic shear strain |
| (7) co2 | matrix cohesion, c_2 |
| (8) cohesion | matrix cohesion, c_1 |
| (9) ctable | number of table relating matrix cohesion c_1 to matrix plastic shear strain |
| (10) d2table | number of table relating matrix dilation ψ_2 to matrix plastic shear strain |
| (11) density | mass density, ρ |
| (12) di2 | matrix dilation angle, ψ_2 |
| (13) dilation | matrix dilation angle, ψ_1 |
| (14) dj2table | number of table relating joint dilation ψ_{j2} to joint plastic shear strain |
| (15) djtable | number of table relating joint dilation ψ_{j1} to joint plastic shear strain |
| (16) dtable | number of table relating matrix dilation angle ψ_1 to matrix plastic shear strain |
| (17) f2table | number of table relating matrix friction angle ϕ_2 to matrix plastic shear strain |
| (18) fj2table | number of table relating joint friction angle ϕ_{j2} to joint plastic shear strain |
| (19) fjtable | number of table relating joint friction angle ϕ_{j1} to joint plastic shear strain |
| (20) fr2 | matrix friction angle, ϕ_2 |
| (21) friction | matrix friction angle, ϕ_1 |
| (22) ftable | number of table relating matrix friction ϕ_1 angle to matrix plastic shear strain |
| (23) jangle | angle of ubiquitous plane measured counterclockwise from x -axis (2D models) |
| (24) jc2 | joint cohesion, c_{j2} |

- (25) **jcohesion** joint cohesion, c_{j1}
- (26) **jdilation** joint dilation angle, ψ_{j1}
- (27) **jd2** joint dilation angle, ψ_{j2}
- (28) **jf2** joint friction angle, ϕ_{j2}
- (29) **jfriction** joint friction angle, ϕ_{j1}
- (30) **jtension** joint tension limit, σ_j^t
- (31) **shear** elastic shear modulus, G
- (32) **tension** matrix tension limit, σ^t
- (33) **tjtable** number of table relating joint tension limit σ_j^t to joint plastic tensile strain
- (34) **ttable** number of table relating matrix tension limit σ^t to matrix plastic tensile strain

The following calculated properties can be printed, plotted or accessed via *FISH*.

- (1) **es_plastic** accumulated matrix plastic shear strain
- (2) **esj_plastic** accumulated joint plastic shear strain
- (3) **et_plastic** accumulated matrix plastic tensile strain
- (4) **etj_plastic** accumulated joint plastic tensile strain
- (5) **state** plastic state

Plasticity state indicator flags are given below. Use logical **and** to find individual and multiple state modes.

Bit Number	State
1	matrix failure in shear now
2	matrix failure in tension now
4	matrix failure in shear in the past
8	matrix failure in tension in the past
16	joint failure in shear now
32	joint failure in tension now
64	joint failure in shear in the past
128	joint failure in tension in the past

See [Section 1](#) in **Constitutive Models** for details.

Note that the default tension limits for the matrix and weakness planes are the same as those for the ubiquitous-joint model.

Table 1.11 Property groups by failure segment for the bilinear, strain-hardening/softening ubiquitous-joint model

Properties		Description
general		
	bijoint	1 for bilinear joint law 0 for linear joint law (default)
	bimatrix	1 for bilinear matrix law 0 for linear matrix law (default)
	bulk	bulk modulus
	density	mass density
	jangle	angle of weakness plane measured counterclockwise from <i>x</i> -axis (2D models)
	jtension <tjtable>	tension limit of joint segments 1 and 2
	shear	shear modulus
	tension <ttable>	tension limit of matrix segments 1 and 2
matrix-segment 1		
	cohesion <ctable>	cohesion
	dilation <dtable>	dilation (degree)
	friction <ftable>	friction (degree)
matrix-segment 2		
	co2 <c2table>	cohesion
	di2 <d2table>	dilation (degree)
	fr2 <f2table>	friction (degree)
joint-segment 1		
	jcohesion <cjtable>	cohesion
	jdilation <djtable>	dilation (degree)
	jfriction <fjtable>	friction (degree)
joint-segment 2		
	jc2 <cj2table>	cohesion
	jd2 <dj2table>	dilation (degree)
	jf2 <fj2table>	friction (degree)

Double-Yield – ZONE model dy

- | | |
|-------------------------|--|
| (1) bulk_mod | maximum elastic bulk modulus, K |
| (2) cap_pressure | current intersection of volumetric yield surface (cap) with pressure (mean stress) axis, p_c |
| (3) cohesion | cohesion, c |
| (4) cptable | number of table relating cap pressure to plastic volume strain |
| (5) ctable | number of table relating cohesion to plastic shear strain |
| (6) density | mass density, ρ |
| (7) dilation | dilation angle, ψ |
| (8) dtable | number of table relating dilation angle to plastic shear strain |
| (9) friction | angle of internal friction, ϕ |
| (10) ftable | number of table relating friction angle to plastic shear strain |
| (11) multiplier | multiplier on current plastic cap modulus to give elastic bulk and shear moduli, R |
| (12) shear_mod | maximum elastic shear modulus, G |
| (13) tension | tension limit, σ^t |
| (14) ttable | number of table relating tensile limit to plastic tensile strain |

The following calculated properties can be printed, plotted or accessed via *FISH*.

- | | |
|-----------------------|---------------------------------------|
| (1) dy_state | special plasticity state indicator* |
| (2) e_plastic | accumulated plastic shear strain |
| (3) e_tension | accumulated plastic tensile strain |
| (4) ev_plastic | accumulated plastic volumetric strain |
| (5) state | plastic state |

* special indicator codes for double-yield model: (1) currently at yield in shear; (2) currently at yield in volume; (4) shear yield criterion involves out-of-plane stress (does not mean at yield); and (8) currently at yield in tension

Plasticity state indicator flags are given below. Use logical **and** to find individual and multiple state modes.

Bit Number	State
1	failure in shear now
2	failure in tension now
4	failure in shear in the past
8	failure in tension in the past
256	failure in volume now
512	failure in volume in the past

The strain-hardening and -softening behavior is controlled by the variation in friction, cohesion and dilation as a function of plastic shear strain, and tension limit as a function of plastic tensile strain, given by a specified table of values. The variation in cap pressure is a function of plastic volumetric strain. Note that if table numbers are given as 0 (default), the properties will take the values given (i.e., with the **cohesion**, **dilation**, **friction**, **tension** or **cap_pressure** keyword).

See [Section 1](#) in **Constitutive Models** for details.

Modified Cam-Clay – ZONE model cam-clay

- (1) **bulk_bound** maximum elastic bulk modulus, K_{max}
- (2) **cv** initial specific volume, v_0 (by default, calculated internally)
- (3) **density** mass density, ρ
- (4) **kappa** slope of elastic swelling line, κ
- (5) **lambda** slope of normal consolidation line, λ
- (6) **mm** frictional constant, M
- (7) **mp1** reference pressure, p_1
- (8) **mpc** pre-consolidation pressure, p_{c0}
- (9) **mv_l** specific volume at reference pressure, p_1 , on normal consolidation line, v_λ
- (10) **poisson** Poisson's ratio, ν
- (11) **shear** elastic shear modulus, G

The following calculated properties can be printed, plotted or accessed via *FISH*.

- (1) **bulk** current elastic bulk modulus, K
- (2) **cam_cp** current mean effective stress, p
- (3) **cam_ev** accumulated total volumetric strain
- (4) **camev_p** accumulated plastic volumetric strain
- (5) **cq** current mean deviatoric stress, q

Plasticity state indicator flags are given below. Use logical **and** to find individual and multiple state modes.

Bit Number	State
1	failure in shear now
4	failure in shear in the past

If a nonzero Poisson's ratio, **poisson**, is given, then the shear modulus will change as the bulk modulus changes; Poisson's ratio remains constant. If the shear modulus, **shear**, is given, and Poisson's ratio is specified, then the shear modulus remains constant; Poisson's ratio will change as the bulk modulus changes.

See [Section 1](#) in **Constitutive Models** for details.

Simplified Cysoil – Chsoil – ZONE model chsoil

- | | |
|-----------------------|--|
| (1) bulk_ref | bulk modulus number, K_{ref} |
| (2) cohesion | cohesion, c |
| (3) ctable | number of table relating cohesion, c , to plastic shear strain |
| (4) density | mass density, ρ |
| (5) dilaw | = 0 for mobilized dilation angle, ψ_m , equal to input value dilf or a function of plastic shear strain if table is input with dtable
= 1 for mobilized dilation angle, ψ_m , characterized by Rowe stress-dilatancy theory
= 2 for mobilized dilation angle, $\psi_m = 0$ if $\phi_m < \phi_{cv}$, and $\psi_m =$ ultimate dilation value, ψ_f , if $\phi_m \geq \phi_{cv}$ |
| (6) dilf | ultimate dilation angle, ψ_f |
| (7) dtable | number of table relating mobilized dilation angle to plastic shear strain |
| (8) fricf | ultimate friction angle, ϕ_f |
| (9) fricv | constant used in dilation laws (dilaw = 1 or 2), ϕ_{cv} |
| (10) m_k | bulk modulus exponent, m |
| (11) n_g | shear modulus exponent, n |
| (12) p_ini | initial effective pressure, p'_m |
| (13) p_ref | reference pressure, p_{ref} |
| (14) poisson | Poisson's ratio, ν |
| (15) rf | failure ratio, R_f |
| (16) shear_ref | shear modulus number, G_{ref} |
| (17) tension | tensile strength, σ^t |
| (18) ttable | number of table relating mobilized tensile strength to plastic tensile strain |
| (19) young_ref | Young's modulus number, E_{ref} |

The following calculated properties can be printed, plotted or accessed via *FISH*.

- | | |
|-----------------------|---|
| (1) bulk_mod | mobilized elastic bulk modulus, K^e |
| (2) dilation | mobilized dilation angle, ψ_m |
| (3) es_plastic | accumulated plastic shear strain, ε^{ps} |
| (4) ev_plastic | accumulated plastic volumetric strain, ε^{pc} |
| (5) friction | mobilized friction angle, ϕ_m |
| (6) shear_mod | mobilized elastic shear modulus, G^e |

See [Section 1](#) in **Constitutive Models** for details.

Cap-Yield – **ZONE model cysoil**

- | | |
|--------------------------|--|
| (1) alpha | cap yielding surface parameter, α |
| (2) bulk | maximum elastic bulk modulus, K |
| (3) cap_pressure | current intersection of volumetric yield surface (cap) with pressure (mean stress) axis, p_c |
| (4) cohesion | cohesion, c |
| (5) cptable | number of table relating cap pressure (p_c) to plastic volume strain (ev_plastic) |
| (6) ctable | number of table relating cohesion to plastic shear strain |
| (7) density | mass density, ρ |
| (8) dilation | ultimate dilation angle, ψ |
| (9) dilation_mob | mobilized dilation angle, ψ_m |
| (10) dtable | number of table relating mobilized dilation angle to plastic shear strain |
| (11) friction | ultimate friction angle, ϕ |
| (12) friction_mob | mobilized friction angle, ϕ_m |
| (13) ftable | number of table relating mobilized friction angle to plastic shear strain |
| (14) multiplier | multiplier on current plastic cap modulus to give elastic bulk and shear moduli, R |
| (15) shear | maximum elastic shear modulus, G |
| (16) tension | tensile strength, σ^t |
| (17) ttable | number of table relating tensile strength to plastic tensile strain |

The following calculated properties can be printed, plotted or accessed via *FISH*.

- | | |
|--------------------------|---|
| (1) bulk_current | current elastic bulk modulus |
| (2) cy_p | mean effective stress, p |
| (3) cy_q | deviatoric stress, q |
| (4) es_plastic | accumulated plastic shear strain, ε^{ps} |
| (5) et_plastic | accumulated plastic tensile strain, ε^{pt} |
| (6) ev_plastic | accumulated plastic volumetric strain, ε^{pc} |
| (7) shear_current | current elastic shear modulus |
| (8) state | plastic state |

See [Section 1](#) in **Constitutive Models** for details.

Hoek-Brown – ZONE model hoekbrown

- | | |
|----------------------|---|
| (1) atable | number of table relating a to e_3^p |
| (2) bulk | elastic bulk modulus, K |
| (3) citable | number of table relating σ_{ci} to e_3^p |
| (4) density | mass density, ρ |
| (5) hb_e3plas | accumulated plastic strain, e_3^p |
| (6) hb_ind | number of iterations* |
| (7) hba | Hoek-Brown parameter, a |
| (8) hbmb | Hoek-Brown parameter, m_b |
| (9) hbs | Hoek-Brown parameter, s |
| (10) hbs3cv | Hoek-Brown parameter, σ_3^{cv} |
| (11) hbsigci | Hoek-Brown parameter, σ_{ci} |
| (12) mtable | number of table relating to m_b to e_3^p |
| (13) multable | number of table relating a multiplier to σ_3 |
| (14) shear | elastic shear modulus, G |
| (15) stable | number of table relating s to e_3^p |

The following property can be printed, plotted or accessed via *FISH*.

- | | |
|------------------|---------------|
| (1) state | plastic state |
|------------------|---------------|

*The Hoek-Brown model makes three attempts to bring a stress point to the yield surface:

- 1) A fast Newton solver is tried.
- 2) If 1) does not converge, the stress point is checked to see if it falls below the apex of the Hoek-Brown envelope (i.e., does it cross the $\sigma_1 = \sigma_3$ line?). If this is the case, then the stress point is set to the apex.
- 3) If 1) and 2) do not work, a bisection method is used to find the stress point on the yield surface.

The **hb_ind** property reflects the attempts made to bring a stress point to the yield surface. Each zone contains a number of triangular subzones that are sent in sequence to the constitutive model. When the first subzone is received by the Hoek-Brown model, the Hoek-Brown model sets **hb_ind** to 0. If case 1 works, **hb_ind** is set to max (**hb_ind**, number of iterations required by the Newton solver). If case 2 is encountered, **hb_ind** is set to max (**hb_ind**, 1000). If case 3 is encountered, **hb_ind** is set to max (**hb_ind**, 1000 + iterations required by the bisection algorithm). If none of the cases work, then **hb_ind** is set to 9999.

Plasticity state indicator flags are given below. Use logical **and** to find individual and multiple state modes.

Bit Number	State
1	failure in shear now
2	failure in tension now
4	failure in shear in the past
8	failure in tension in the past

See [Section 1](#) in **Constitutive Models** for details.

Modified Hoek-Brown – ZONE model mhoek

- | | |
|-----------------------|---|
| (1) atable | number of table relating a to the evolution parameter |
| (2) bulk | elastic bulk modulus, K |
| (3) citable | number of table relating σ_{ci} to the evolution parameter |
| (4) density | mass density, ρ |
| (5) hba | Hoek-Brown parameter, a |
| (6) hb_do | = 0 to input a constant dilation angle specified by hbpsi
= -1 to specify associated plastic flow; $\psi_c = \phi_c$
= val where val is a fraction of friction angle, ϕ_c ($\psi_c = \text{val} \times \phi_c$) |
| (7) hb_len | calibration length to calibrate model properties to account for zone size |
| (8) hbmb | Hoek-Brown parameter, m_b |
| (9) hb_po | = 0 for evolution parameter set to plastic strain in direction of least compressive principal stress
= 1 for evolution parameter set to plastic shear strain |
| (10) hbpsi | dilation angle, ψ_c (specified if hb_do = 0) |
| (11) hbsigci | Hoek-Brown parameter, σ_{ci} |
| (12) hb_so | = 0 for SOLVE fos solution controlled by shear strength
= 1 for SOLVE fos solution controlled by unconfined compressive strength |
| (13) hbs | Hoek-Brown parameter, s |
| (14) hbtension | tension cutoff, t |
| (15) mtable | number of table relating m_b to the evolution parameter |
| (16) shear | elastic shear modulus, G |
| (17) stable | number of table relating s to the evolution parameter |
| (18) ttable | number of table relating tension, t , to the evolution parameter |

The following calculated properties can be printed, plotted or accessed via *FISH*.*

- | | |
|------------------------|---|
| (1) hb_aac | current value of a |
| (2) hb_cohesion | current value of cohesion, c_c |
| (3) hb_dilation | current value of dilation angle, ψ_c * |
| (4) hb_friction | current value of friction angle, ϕ_c |
| (5) hb_mmc | current value of m_b |
| (6) hb_plas | plastic strain in direction of least compressive principal stress (if hb_po = 0)
plastic shear strain (if hb_po = 1) |
| (7) hb_scc | current value of σ_{ci} |
| (8) hb_ssc | current value of s |
| (9) state | plastic state |

* If **hb_do** = 0, **hb_dilation** = min(**hbpsi**, ψ_c). Thus, **hb_dilation** may not always be equal to the specified value of **hbpsi**.

Plasticity state indicator flags are given below. Use logical **and** to find individual and multiple state modes.

Bit Number	State
1	failure in shear now
2	failure in tension now
4	failure in shear in the past
8	failure in tension in the past

See [Section 1](#) in **Constitutive Models** for details.

Creep Models

Classical Viscoelastic (Maxwell subst.) – ZONE model viscous

- (1) **bulk** elastic bulk modulus, K
- (2) **density** mass density, ρ
- (3) **shear** elastic shear modulus, G
- (4) **viscosity** dynamic viscosity, η

See [Section 1](#) in **Constitutive Models** for details.

Burgers Model – **ZONE model burgers**

- | | |
|-----------------------|------------------------------|
| (1) bulk | elastic bulk modulus, K |
| (2) density | mass density, ρ |
| (3) kshear | Kelvin shear modulus, G^K |
| (4) kviscosity | Kelvin viscosity, n^K |
| (5) mshear | Maxwell shear modulus, G^M |
| (6) mviscosity | Maxwell viscosity, n^M |

The following calculated properties can be printed, plotted or accessed via *FISH*.

- | | |
|------------------|---------------------------------|
| (1) k_exx | Kelvin strain in x -direction |
| (2) k_eyx | Kelvin shear strain |
| (3) k_eyy | Kelvin strain in y -direction |

See [Section 1](#) in **Constitutive Models** for details.

Power Law – **ZONE model power**

- | | |
|--------------------|------------------------------------|
| (1) a_1 | power-law constant, A_1 |
| (2) a_2 | power-law constant, A_2 |
| (3) bulk | elastic bulk modulus, K |
| (4) density | mass density, ρ |
| (5) n_1 | power-law exponent, n_1 |
| (6) n_2 | power-law exponent, n_2 |
| (7) rs_1 | reference stress, σ_1^{ref} |
| (8) rs_2 | reference stress, σ_2^{ref} |
| (9) shear | elastic shear modulus, G |

See [Section 1](#) in **Constitutive Models** for details.

WIPP Model – **ZONE model wipp**

- | | |
|-----------------------|---|
| (1) a_wipp | WIPP model constant, A |
| (2) act_energy | activation energy, Q |
| (3) b_wipp | WIPP model constant, B |
| (4) bulk | elastic bulk modulus, K |
| (5) d_wipp | WIPP model constant, D |
| (6) density | mass density, ρ |
| (7) e_dot_star | critical steady-state creep rate, $\dot{\epsilon}_{ss}^*$ |
| (8) gas_c | gas constant, R |
| (9) n_wipp | WIPP model exponent, n |
| (10) shear | elastic shear modulus, G |
| (11) temp | zone temperature, T |

The following calculated properties can be printed, plotted or accessed via *FISH*.

- | | |
|--------------------|---------------------------------------|
| (1) e_prime | accumulated primary creep strain |
| (2) e_rate | accumulated primary creep strain rate |

See [Section 1](#) in **Constitutive Models** for details.

Burgers-Creep Viscoplastic Model – ZONE model cvisc

- | | |
|------------------------|-------------------------------------|
| (1) bulk | elastic bulk modulus, K |
| (2) cohesion | cohesion, c |
| (3) density | mass density, ρ |
| (4) dilation | dilation angle, ψ |
| (5) friction | angle of internal friction, ϕ |
| (6) kshhear | Kelvin shear modulus, G^K |
| (7) kviscosity | Kelvin viscosity, η^K |
| (8) mshear | elastic shear modulus, G^M |
| (9) tension | tension limit σ^t |
| (10) mviscosity | Maxwell dynamic viscosity, η^M |

The following calculated properties can be printed, plotted or accessed via *FISH*.

- | | |
|------------------|---------------------------------|
| (1) k.exx | Kelvin strain in x -direction |
| (2) k.exy | Kelvin shear strain |
| (3) k.eyy | Kelvin strain in y -direction |

See [Section 1](#) in **Constitutive Models** for details.

Power Law Viscoplastic Model – ZONE model cpow

- | | |
|---------------------|------------------------------------|
| (1) a_1 | power-law constant, A_1 |
| (2) a_2 | power-law constant, A_2 |
| (3) bulk | elastic bulk modulus, K |
| (4) cohesion | cohesion, c |
| (5) density | mass density, ρ |
| (6) dilation | dilation angle, ψ |
| (7) friction | angle of internal friction, ϕ |
| (8) n_1 | power-law exponent, n_1 |
| (9) n_2 | power-law exponent, n_2 |
| (10) rs_1 | reference stress, σ_1^{ref} |
| (11) rs_2 | reference stress, σ_2^{ref} |
| (12) shear | elastic shear modulus, G |
| (13) tension | tension limit, σ^t |

See [Section 1](#) in **Constitutive Models** for details.

WIPP-Creep Viscoplastic Model – ZONE model pwipp

- (1) **a_wipp** WIPP model constant, A
- (2) **act_energy** activation energy, Q
- (3) **b_wipp** WIPP model constant, B
- (4) **bulk** elastic bulk modulus, K
- (5) **d_wipp** WIPP model constant, D
- (6) **density** mass density, ρ
- (7) **e_dot_star** critical steady-state creep rate, $\dot{\epsilon}_{ss}^*$
- (8) **gas_c** gas constant, R
- (9) **kshear** material parameter, k_ϕ
- (10) **n_wipp** WIPP model exponent, n
- (11) **qdil** material parameter, q_k
- (12) **qvol** material parameter, q_ϕ
- (13) **shear** elastic shear modulus, G
- (14) **temp** zone temperature, T
- (15) **tension** tension limit, σ^t

Plasticity state indicator flags are given below. Use logical **and** to find individual and multiple state modes.

Bit Number	State
1	failure in shear now
2	failure in tension now
4	failure in shear in the past
8	failure in tension in the past

The following calculated properties can be printed, plotted or accessed via *FISH*.

- (1) **e_prime** primary creep strain
- (2) **e_rate** primary creep rate
- (3) **es_plastic** accumulated plastic shear strain
- (4) **et_plastic** accumulated plastic tensile strain

See [Section 1](#) in **Constitutive Models** for details.

Crushed Salt Model – **ZONE model cwipp**

- | | |
|------------------------|---|
| (1) a_wipp | WIPP model constant, A |
| (2) act_energy | activation energy, Q |
| (3) b_f | final, intact salt, bulk modulus, K_f |
| (4) b_wipp | WIPP model constant, B |
| (5) b0 | creep compaction parameter, B_0 |
| (6) b1 | creep compaction parameter, B_1 |
| (7) b2 | creep compaction parameter, B_2 |
| (8) bulk | elastic bulk modulus, K |
| (9) d_f | final, intact salt, density, ρ_f |
| (10) d_wipp | WIPP model constant, D |
| (11) density | mass density, ρ |
| (12) e_dot_star | critical steady-state creep rate, $\dot{\epsilon}_{ss}^*$ |
| (13) gas_c | gas constant, R |
| (14) n_wipp | WIPP model exponent, n |
| (15) rho | density, ρ |
| (16) s_f | final, intact salt, shear modulus, G_f |
| (17) shear | elastic shear modulus, G |
| (18) temp | zone temperature, T |

The following calculated properties can be printed, plotted or accessed via *FISH*.

- | | |
|-------------------|-----------------------------------|
| (1) frac_d | current fractional density, F_d |
| (2) s_g1 | creep compaction parameter, G |
| (3) s_k1 | creep compaction parameter, K |

See [Section 1](#) in **Constitutive Models** for details.

