1 COMMAND REFERENCE

This section contains detailed information about all the commands used by *FLAC3D*. 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 is also intended as a 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 the commands. Some common input conventions and features are described in Section 1.1. A command summary given in alphabetical order is also provided in Section 1 in the **Command and *FISH* Reference Summary** for convenience.

1.1 Common Conventions and Features

1.1.1 Syntax

FLAC3D may be operated in "interactive" mode (i.e., commands entered via the keyboard) or "file-driven" mode (i.e., commands stored in a data file and read in from the file). 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 following 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 to be typed, at a minimum, 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. The entire word for commands and keywords may be entered if the user so desires. By default, the words are not case-sensitive — either uppercase or lowercase letters may be used. Case sensitivity can, however, be activated; see the **SET case** command.

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. These are:

< > 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 on the output. A single input line, including comments, may contain up to 254 characters. An ampersand (&) at the end of a line denotes that the next line is a continuation of keywords or numeric input.

The ampersand may be placed anywhere in an input command line to break the line. The maximum length of a single command, including all continuations and macro expansions, is 1024 characters.

1.1.2 Interactive Input

FLAC3D supports an 80-character keyboard buffer for the user to type ahead while *FLAC3D* 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 input by the keyboard
<HOME>	moves cursor to beginning of input line

1.1.3 Commands Accepting the **range** *Phrase*

The commands in Table 1.2 accept an optional **range** keyword phrase. If the action of the command is to be limited to a specified range within a model, a **range** keyword phrase must be given at the *end* of the command line.

Table 1.2 Commands accepting the **range** *phrase*

APPLY
ATTACH
DELETE
FIX
FREE
GENERATE zone copy
GENERATE zone reflect
GEOM_TEST
GROUP
INITIAL
INTERFACE
MODEL
PLOT
PRINT
PROPERTY
SEL
WATER

A range object can be made up of a number of individual range elements. A range element is defined by a keyword phrase that follows the **range** keyword. (Alternatively, a range can be created with the **RANGE** command.) The following keyword phrases are available.

annulus **c**enter ***xc yc zc*** **r**adius ***r1 r2***

(***xc, yc, zc***) is the center of a spherical region; the range is between radii ***r1*** and ***r2***.

cid ***imin imax***

beginning and ending component identification numbers for structural element components (see Section 1 in **Structural Elements**). Note: If forces and moments are to be applied to structural element nodes, then the keywords **cid** and **id** both represent the ID of the node.

cylinder **end1** *x1 y1 z1* **end2** *x2 y2 z2* **r**adius *r*

cylindrical range with one end of the cylinder axis (**end1**) at location (***x1, y1, z1***) and the other end (**end2**) at location (***x2, y2, z2***), with the cylinder radius of ***r***

direction *v1, v2, v3* <**ang**le *value*>

zone surface with outward normal defined by the vector (***v1, v2, v3***). By default, the tolerance is 90 degrees. This can be changed with the optional **angle** keyword followed by the tolerance angle ***value***.

group name

group name, identified by the **GROUP** command, of zones or structural elements

id *il* <*iu*>

beginning and ending identification numbers for zones, gridpoints, structural elements, interface elements or nodes

model keyword

zones with corresponding constitutive model (see the **MODEL** command for keyword names)

name Once a range has been named (see the **RANGE** command), the name becomes a keyword that can be specified in place of the range element or as a range element in another range.

plane keyword

range above or below a specified plane. The plane is defined by the keywords:

above half-space above the plane

below half-space below the plane

dd ***dd***

dip-direction angle, measured in the global xy-plane clockwise from the positive y-axis

dip ***dip***

dip angle, measured in the negative z-direction from the global xy-plane

distance ***d***

points within distance ***d*** of the plane

normal ***xn yn zn***

unit normal vector ***xn, yn, zn*** of the plane

origin ***x y z***

one point on the plane at location (***x, y, z***)

The location of the plane is defined by **origin** and either **dd** and **dip** or **normal**.

selid ***imin imax***

Useful for application of forces and stress and boundary conditions for all nodes of SELs with particular **selid**s in the range specified.

seltype keyword

structural-element types. The keyword may be one of the following.

beam beam elements

cable cable elements

geogrid geogrid elements

liner liner elements

pile pile elements

shell shell elements

sphere **c**enter ***x, y, z*** **r**adius ***r***

spherical range with centroid at location (***x, y, z***) and radius, ***r***

volume ***n***

volume range defined by the volume ***n***, created with the **GENERATE surface** command

x ***xl*** <***xu***>

lower and upper limits for the x-coordinate. The tolerance is 10^{-6} if only ***xl*** is specified.

y ***yl*** *<**yu**>*

lower and upper limits for the y-coordinate. The tolerance is 10^{-6} if only ***yl*** is specified.

z ***zl*** *<**zu**>*

lower and upper limits for the z-coordinate. The tolerance is 10^{-6} if only ***zl*** is specified.

If multiple **range** keyword phrases are given following the **range** keyword, the range is the intersection of the separate range elements — i.e., the selected points or objects are those that are common to all the specified items.

Two additional keywords are available to modify the definition of a range element.

any If **any** is given after a range element, any object that falls within the range element is considered part of the range regardless of other range elements that define the range.

not If **not** is given after a range element, the meaning of the element is changed to signify the set *not* inside that range element.

1.1.4 Orientation of Nodes and Faces within a Zone

A *zone* is a closed geometric domain, with *nodes* at the vertices and planar faces forming the surface of the zone. The relative orientation of the nodes and faces is shown in Figure 1.1 for the five basic primitive mesh shapes: brick, wedge, pyramid, degenerate brick and tetrahedron. (See Table 1.3.) Each face has vertices; these vertices are also identified in Figure 1.1. Several *FLAC3D* and *FISH* commands (e.g., **ATTACH**) refer to this orientation.

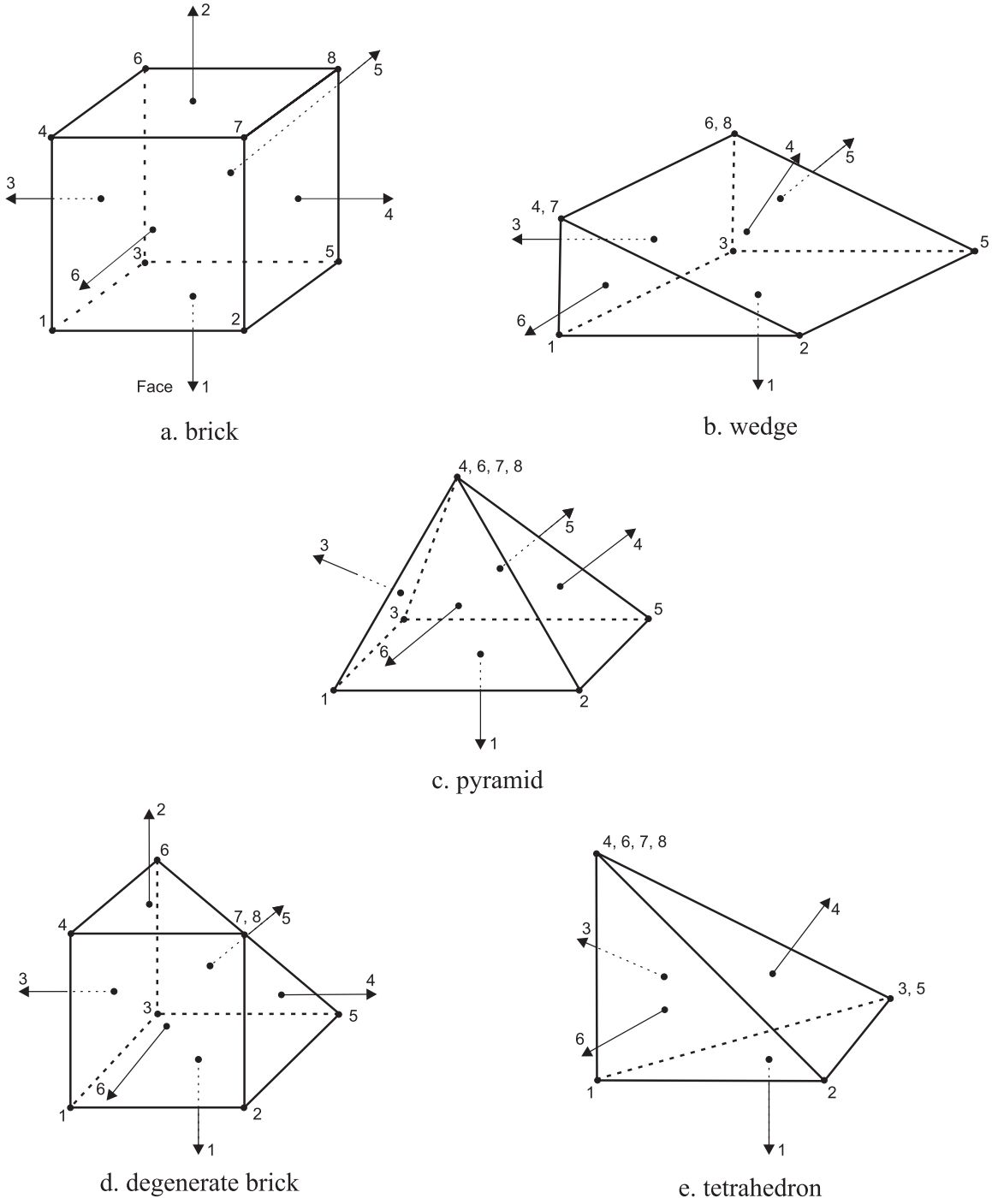


Figure 1.1 Zone geometry

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 *FLAC3D*, or to restart previous model simulations and continue from the last analysis stage. The following commands provide program control.

CALL reads a user-prepared, ASCII input data file into *FLAC3D* and executes the commands — this is called *batch mode*.

CONTINUE continues reading a batch file.

IMPGRID
EXPGRID allows the importing and exporting of *3DShop* grid files.

NEW starts a new problem without exiting *FLAC3D*.

PAUSE pauses reading a batch file.

QUIT stops execution of *FLAC3D* and returns control to the operating system.

RESTORE restores an existing (binary) saved state from a previously executed problem.

RETURN returns program control from batch mode to the local, interactive mode (or to the calling file if multiple levels of calls are nested).

SAVE saves the current state of the analysis in a binary file.

SET sets basic parameters.

STOP stops execution of *FLAC3D* and returns control to the operating system.

SYSTEM spawns a DOS command session.

It is best to give **SAVE**d files a different extension (e.g., ".SAV") from input files (e.g., ".DAT"), to avoid confusion when a saved state is **RESTORE**d or an input file is **CALL**ed.

1.2.2 Access the Graphical User Interface

The **MAINWIN** command positions the main window in the GUI.

1.2.3 Specify Special Calculation Modes

FLAC3D performs static, mechanical calculations as the standard mode. Optional calculation modes are also available and are specified with the **CONFIG** command. **CONFIG** can be issued at any stage of an analysis, but it must be specified for the calculation mode to be invoked. One or more options can be requested with the following keywords. Once a calculation mode is activated, it can only be deactivated by giving a **NEW** command.

cppudm C++ user-defined models (only available with the UDM option)

creep creep analysis (only available with the creep model option)

dynamic fully dynamic analysis (only available with the dynamic option)

fluid fluid-flow analysis

thermal thermal analysis (only available with the thermal model option)

1.2.4 Specify Additional Memory

Additional memory can be allocated with the **CONFIG** command. **CONFIG** with these keywords can be issued at any stage of an analysis. A larger value will increase the number of locations, leaving existing values intact. Specifying a smaller value will erase the higher index locations.

gpextra extra gridpoint variables for *FISH* use

zextra extra zone variables for *FISH* use

1.2.5 Input Problem Geometry

The following commands are used to specify the grid and shape it to fit the desired problem region.

ATTACH permits faces along two touching sub-grids to be attached.

GENERATE generates grid primitives, points and surfaces.

GEOM_TEST tests integrity of mesh for accuracy of solution.

INITIAL allows the manipulation of gridpoints.

1.2.6 Create Named Objects

These commands allow the user to create macro and model objects.

GROUP creates a group model object and identifies a collection of zones and gridpoints.

MACRO creates a macro object.

RANGE creates a range model object and identifies a volume of space.

1.2.7 Assign Constitutive Models and Properties

A constitutive model is associated with a specific region of the grid through the **MODEL** command. The mechanical constitutive models available in *FLAC3D* are discussed in Section 2 in **Theory and Background**. The following keywords are available to assign the appropriate model.

Mechanical Models

anisotropic	transversely isotropic elastic model
cam-clay	Cam-clay model
doubleyield	double-yield (cap) plasticity model
drucker	Drucker-Prager plasticity model
elastic	isotropic elastic model
finn	dynamic pore-pressure generation model (available only for dynamic option — see Section 3 in **Optional Features**)
hoekbrown	generalized Hoek-Brown model
mohr	Mohr-Coulomb plasticity model
null	null model (i.e., no material)
orthotropic	orthotropic elastic model
ssoftening	strain-hardening/softening plasticity model
subiquitous	bilinear strain-hardening/softening ubiquitous-joint model
ubiquitous	ubiquitous-joint model

Fluid-Flow Models (see Section 1 in **Fluid-Mechanical Interaction**)

fl_anisotropic	anisotropic fluid flow
fl_isotropic	isotropic fluid flow
fl_null	null zone for fluid flow

Creep Models (available only for creep model option — see Section 2 in **Optional Features**)

burger	Burger viscoelastic model
cpow	two component power law; viscoplastic model
cvisc	Burger viscoplastic model
cwipp	crushed-salt constitutive model

power two-component power law

pwipp viscoplastic model

viscous classical viscoelastic model

wipp WIPP reference creep formulation

Thermal Models (available only for thermal model option — see Section 1 in **Optional Features**)

th_ac isotropic advection-conduction

th_anisotropic anisotropic thermal conductivity

th_isotropic isotropic thermal conductivity

th_null null zone for heat conduction

User-Defined Model

load loads a user-defined constitutive model as a DLL

Properties are assigned for each model through the **PROPERTY** command. For the strain-hardening/softening, double-yield and bilinear models, properties can depend upon accumulated plastic strain as defined through the **TABLE** command. Fluid flow properties (fluid bulk modulus or Biot's modulus) are prescribed with the **INITIAL** command. Mass densities for the material and the fluid are also prescribed with the **INITIAL** command.

1.2.8 Assign Initial Conditions

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

INITIAL initializes certain gridpoint and zone variables such as mass density, stress state and velocity.

SET allows the user to initialize both problem and model conditions by selecting one or more keywords, such as the following:

gravity specifies gravity.

large/small selects either large- or small-strain solution.

WATER initializes water table conditions for effective stress calculation.

1.2.9 Apply Boundary Conditions

Model boundary conditions are prescribed in *FLAC3D* with the following keywords.

APPLY applies mechanical, fluid-flow and thermal conditions to any model boundary.

DELETE deletes zones from a model.

FIX/FREE allows velocity, pore pressure or temperature to be fixed (i.e., prevented from changing) or freed (i.e., allowed to change) at selected gridpoints.

1.2.10 Specify Structural Support

Six types of structural support members can be specified through the **SEL** (for Structural ELement) command. The types of support members available are denoted by the following keywords.

beam specifies a beam

cable specifies a cable

geogrid specifies a geogrid-type shell

liner specifies a liner-type shell

pile specifies a pile

shell specifies a shell

Properties for each type of support member are defined via the **property** keyword of the **SEL** command. Structural element logic in *FLAC3D* is described in Section 1 in **Structural Elements**.

1.2.11 Specify Interfaces or Joints

The special command **INTERFACE** is available to define interfaces between two or more sub-grids in *FLAC3D*. These interfaces are planes upon which slip and/or separation is allowed. Interface properties are also defined through this command. Interfaces are described in Section 3 in **Theory and Background**.

1.2.12 Specify User-defined Variables or Functions

The embedded programming language (*FISH*) in *FLAC3D* may be invoked to define special variables or functions which a user desires for a specific problem. *FISH* statements (described in Section 2 in the ***FISH* volume**) are any statements given between the *FLAC3D* commands **DEFINE** and **END**.

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

1.2.13 Monitor Model Conditions during the Solution Process

Change in model variables can be monitored as the solution process progresses. This is helpful to ascertain when an equilibrium or failure state has been reached.

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.

PDELETE deletes particles from tracking.

TRACK allows tracking of fluid particles.

1.2.14 Solve the Problem

Once the appropriate problem conditions are defined in the *FLAC3D* model, the problem is solved by taking a series of calculation steps. The following commands permit either automatic solution of the *FLAC3D* model or user-control of the solution process.

CYCLE *n* executes ***n*** timesteps.

SOLVE enables the automatic detection of a steady-state solution. The calculation is performed until a preset limiting condition is reached. Limiting conditions can be modified by keywords through the **SET** command. Keywords also allow the user to define time limits and timesteps for transient analyses such as fluid flow, heat transfer and creep, and for fully dynamic calculation.

STEP *n* executes ***n*** timesteps.

For both **SOLVE** and **STEP** (or **CYCLE**), the maximum out-of-balance force (or force ratio) for the model is continually displayed on the screen. The user may interrupt the calculation stepping at any time by pressing the <Esc> key. *FLAC3D* 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 carry on with the analysis, if desired.

1.2.15 Generate Model Output

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

PLOT requests a plot of various problem variables (either on the screen or a hardcopy device), including the **HISTORY** of a variable.

PRINT displays output for problem conditions and main grid variables.

SET provides several controls over different output conditions.

TITLE records title on save files and on plots.

1.3 *FLAC3D* Commands — Detailed Listing

APPLY keyword <keyword> ***value*** <keyword> <**ra**nge . . . >

or

APPLY **rem**ove <keyword> <**ra**nge . . . >

The **APPLY** command is used to apply mechanical, fluid-flow and thermal boundary conditions to any external or internal boundary of the model grid or to interior gridpoints. The command is also used to apply internal body forces and fluid flow or thermal sources to zones in the model. The user must specify the keyword type to be applied (i.e., gridpoint, zone or face type), the numerical value, and an optional range over which the boundary condition is to be applied. The range can be given in several forms (see Section 1.1.3). If no range is specified, then the command applies to the entire model.

Three keyword types are used to apply boundary conditions. The associated keywords are given for each type.

Gridpoint-type Keywords — Mechanical Boundary Condition

daccel ***value***

acceleration component applied in the dip direction of the local gridpoint axes (available only for dynamic option — see Section 3 in **Optional Features**)

dvelocity ***value***

velocity component applied in the dip direction of the local gridpoint axes

ff free-field boundary condition (available only for dynamic option — see Section 3 in **Optional Features**)

naccel ***value***

acceleration component applied in the normal direction of the local gridpoint axes (available only for dynamic option — see Section 3 in **Optional Features**)

nvelocity ***value***

velocity component applied in the normal direction of the local gridpoint axes

saccel ***value***

acceleration component applied in the strike direction of the local gridpoint axes (available only for dynamic option — see Section 3 in **Optional Features**)

svelocity ***value***

velocity component applied in the strike direction of the local gridpoint axes

xaccel ***value***

x-component of acceleration applied at a gridpoint (available only for dynamic option — see Section 3 in **Optional Features**)

xforce ***value***

x-component of applied force at a gridpoint

xreaction x-component of reaction force at a gridpoint

xvelocity ***value***

x-component of velocity applied at a gridpoint

yaccel ***value***

y-component of acceleration applied at a gridpoint (available only for dynamic option — see Section 3 in **Optional Features**)

yforce ***value***

y-component of applied force at a gridpoint

yreaction y-component of reaction force at a gridpoint

yvelocity ***value***

y-component of velocity applied at a gridpoint

zaccel ***value***

z-component of acceleration applied at a gridpoint (available only for dynamic option — see Section 3 in **Optional Features**)

zforce ***value***

z-component of applied force at a gridpoint

zreaction z-component of reaction force at a gridpoint

zvelocity ***value***

z-component of velocity applied at a gridpoint

NOTES

1. The local gridpoint axes are defined by the normal direction vector at each gridpoint. The default gridpoint normal is the average of the normal vectors of the faces meeting at the gridpoint. The dip, strike and normal directions form a right-handed coordinate system. Given the normal vector, the other local axes are defined by "*d*-axis," which points downward (i.e., in the negative *z*-direction) along the dip-direction, and "*s*-axis," which is horizontal (i.e., lies within the global *xy*-plane), such that *d*-*s*-*n* form a right-handed system (see Figure 1.2). The normal direction may also be specified with the **plane** keyword. This will override the default normal.

2. **xreaction**, **yreaction** and **zreaction** forces provide reactions to equilibrate current unbalanced forces. Gridpoint conditions (e.g., **xvelocity**) remain unchanged.

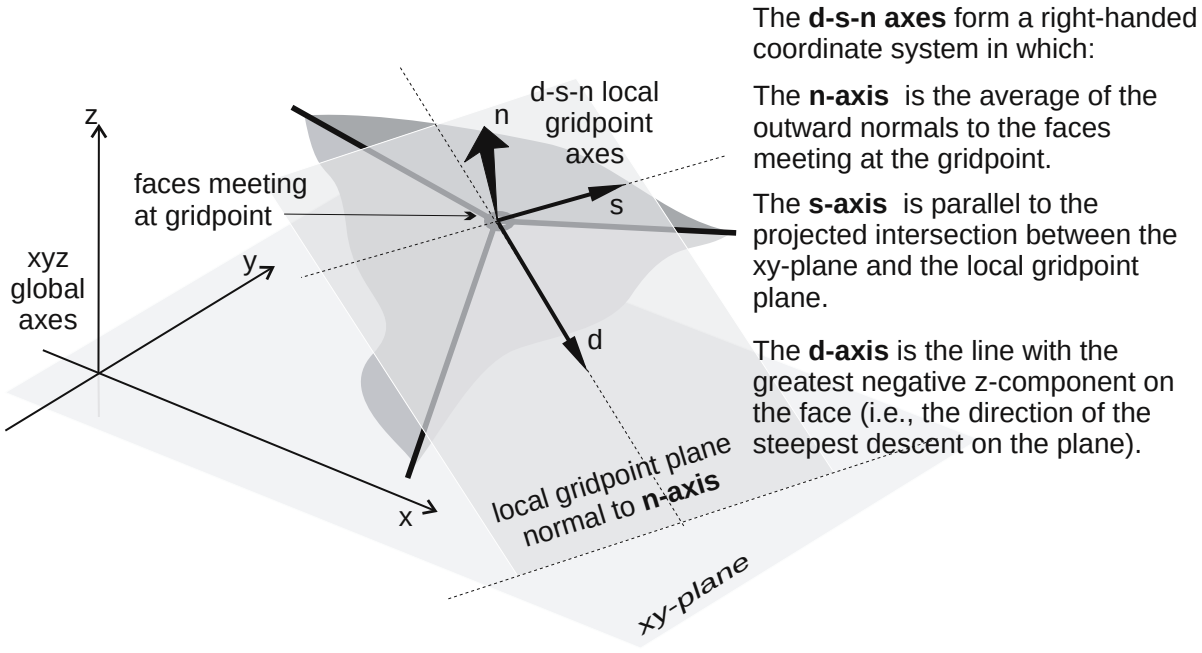


Figure 1.2 Local gridpoint axes

Gridpoint-type Keywords — Fluid-Flow Boundary Condition

pp ***v***

Pore pressure can be applied to gridpoints at each boundary gridpoint in the specified range. Use the **interior** keyword to apply the condition to an interior gridpoint. Pore pressure variation can be represented by a *FISH* function using the **history** keyword. A gradient of pore pressure can be applied using the **gradient** keyword. Please see page p. 1 - 29 for details.

Note: A fluid flow model must exist for this command to work.

pwell ***v***

A fluid-flow rate ***v*** (e.g., in m^3/sec) is applied at each boundary gridpoint in the specified range. This command is used to specify a constant inflow (***v*** > 0) or outflow (***v*** < 0) along a fluid flow boundary. Use the **interior** keyword to apply the condition to an interior gridpoint. When a new well is applied to a gridpoint with an existing well, the new well flow rate replaces the existing well flow rate.

Note: A fluid flow model must exist for this command to work.

Gridpoint-type Keyword — Thermal Boundary Condition

Note that this is available only for the thermal model option (see Section 1 in **Optional Features**).

psource *v*

A heat-generating source, ***v***, is applied as a point source of the specified strength (e.g., in *W*) at each *boundary gridpoint* in the specified range. Use the **interior** keyword to apply the condition to an interior gridpoint. When a new source is applied to a gridpoint with an existing source, the new source strength *replaces* the existing source strength.

Decay of the heat source can be represented by a *FISH* history using the **history** keyword. For example, the following *FISH* function performs an exponential decay of the applied source:

```
def decay
  decay=exp(deconst*(thtime-thini))
end
set thini=0.0 deconst=-1.0
apply psource=10 hist=decay
```

Zone-type Keywords — Mechanical Boundary Condition

xbodyforce ***value***

x-component of the body force applied to a zone

ybodyforce ***value***

y-component of the body force applied to a zone

zbodyforce ***value***

z-component of the body force applied to a zone

Zone-type Keyword — Fluid-Flow Boundary Condition

vwell ***v***

A volume rate of flow, ***v*** (i.e., fluid volume per zone volume per unit time), is specified for each *zone* in the specified range (***v*** > 0 for inflow). When a new volumetric source is applied to a zone with an existing source, the new source *replaces* the existing source. Note: A fluid flow model must exist for this command to work.

Zone-type Keyword — Thermal Boundary Condition

Note that this is available only for the thermal model option (see Section 1 in **Optional Features**).

vsource ***v***

A heat-generating source, ***v***, is applied as a volume source of the specified strength (e.g., in W/m^3) in each *zone* in the specified range. When a new source is applied to a zone with the existing source, the new source strength *replaces* the existing source strength.

Decay of the heat source can be represented by a *FISH* history using the **history** keyword. See the **psource** keyword for an example.

Face-type Keywords — Mechanical Boundary Condition

dquiet — quiet (viscous) boundary applied in the dip direction of the local gridpoint axes (available only for dynamic option — see Section 3 in **Optional Features**)

dstress ***value***

stress component applied in the dip direction of the local face axes

nquiet — quiet (viscous) boundary applied in the normal direction of the local gridpoint axes (available only for dynamic option — see Section 3 in **Optional Features**)

nstress ***value***

stress component applied in the normal direction of the local face axes

squiet — quiet (viscous) boundary applied in the strike direction of the local gridpoint axes

sstress ***value***

stress component applied in the strike direction of the local face axes

sxx ***value***

xx-component of the stress tensor applied at a face

sxy ***value***

xy-component of the stress tensor applied at a face ($\sigma_{xy} = \sigma_{yx}$)

sxz ***value***

xz-component of the stress tensor applied at a face ($\sigma_{xz} = \sigma_{zx}$)

syy ***value***

yy-component of the stress tensor applied at a face

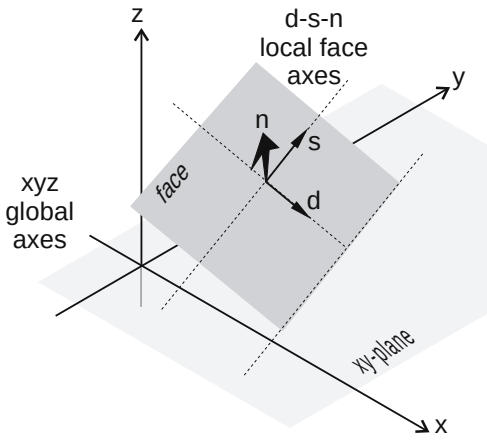
syz ***value***

yz-component of the stress tensor applied at a face ($\sigma_{yz} = \sigma_{zy}$)

szz ***value***

zz-component of the stress tensor applied at a face

The normal stress, **nstress**, and shear stresses, **dstress** and **sstress**, are applied on a face. The local face axes are defined by the normal to the face. The dip, strike and normal directions form a right-handed coordinate system. Given the normal vector, the other local axes are defined by "*d*-axis," which points downward (i.e., in the negative *z*-direction) along the dip-direction, and "*s*-axis," which is horizontal (i.e., lies within the *xy*-plane) such that *d*-*s*-*n* form a right-handed system (shown in Figure 1.3). The **plane** keyword does not apply to **nstress**, **dstress**, **sstress**, **nquiet**, **dquiet** or **squiet**.



The **d-s-n axes** form a right-handed coordinate system in which:

The **n-axis** is the outward normal to the face.

The **s-axis** is parallel to the projected intersection between the xy-plane and the face.

The **d-axis** is the line with the greatest negative z-component on the face (i.e., the direction of steepest descent on the plane).

Figure 1.3 Local face axes

Face-type Keywords — Fluid-Flow Boundary Condition

discharge ***v***

Fluid flux ***v*** is the component of the specific discharge vector (e.g., in m/s) applied normal to the boundary.

leakage ***v1 v2***

v1 is the pore pressure in the leaky layer.

v2 is the leakage coefficient, h (e.g., in m^3/N sec).

See Eq. (1.18) in **Fluid-Mechanical Interaction** for the formula for a leaky boundary condition. A leaky condition is applied over the range of *faces* specified. The **history** keyword is not active for leakage.

Face-type Keywords — Thermal Boundary Condition

convection ***v1 v2***

v1 is the temperature T_e of the medium to which convection occurs.

v2 is the convective heat transfer coefficient h (e.g., in $W/m^{2\circ}C$).

A convective boundary condition is applied over the range of *faces* specified. The **history** keyword is not active for convection.

flux ***v***

v is the initial flux (e.g., in W/m^2).

A flux is applied over the range of *faces* specified. This command is used to specify a constant flux into (***v*** > 0) or out of (***v*** < 0) a thermal boundary of the grid. Decay of the flux can be represented by a *FISH* history using the optional keyword **history**. See the **psource** keyword for an example.

Optional Keywords

Optional keywords may precede or follow the numerical value.

The following optional keywords may *immediately precede* the numerical value.

add adds the specified value to the existing value for the boundary condition at the gridpoint, zone or face.

multiply multiplies the specified value by the existing value for the boundary condition at the gridpoint, zone or face.

The following keywords may be given *immediately following* the numerical value.

gradient ***gx gy gz***

A gradient may be applied to the value in the x-, y- and z-directions. The value varies according to the relation

$$value\_modified = value + gx \times x + gy \times y + gz \times z$$

where ***(x, y, z)*** is the vector from the coordinate origin.

history keyword

A history multiplier may be applied to the numerical value with the **history** keyword. The history can be applied in one of two forms by using the following keywords.

name The history multiplier is a *FISH* function in which name is the function name.

table ***n*** <keyword>
string <keyword>

The history multiplier is applied as a table (see the **TABLE** command) in which ***n*** is a table number. A table name, string, can be given in place of a table number. By default, the x-value of the table is the step number. If one of the calculation modes (creep, dynamic, thermal, fluid flow) is active, then the x-value will correspond to the time scale for the active modes. The time scale for the x-axis can also be selected by giving one of the following keywords. (This will override the default time scale.)

creep creep time scale

dynamic dynamic time scale

fluid fluid flow time scale

thermal thermal time scale

interior allows the condition to be applied to an interior gridpoint. This only applies to gridpoint-type keywords.

plane keyword ***value*** . . .

A local plane, along which the boundary condition acts, may be specified — for the purpose of knowing the boundary normal direction, not for location purposes. The plane may be specified in one of two ways, either with the keywords

dd ***d***

dip direction, ***d***, of the plane measured in the global xy-plane clockwise from the positive y-axis

and

dip ***d***

dip angle, ***d***, of the plane measured in the negative z-direction from the global xy-plane

or with the keyword

normal ***xn yn zn***

(***xn***, ***yn***, ***zn***) is the unit normal vector to the plane.

The local plane may be specified for gridpoint keywords **nvelocity**, **dvelocity**, **svelocity**, **naccel**, **daccel** and **saccel**. The local plane does not apply to face keywords **nstress**, **dstress**, **sstress**, **nquiet**, **dquiet** and **squiet**.

Removing **APPLY** Conditions

A condition stays in effect until it is removed with the keyword phrase

remove <keyword> <**ra**nge. . . >
<**gp**> <**ra**nge. . . >
<**zo**ne> <**ra**nge. . . >
<**fa**ce> <**ra**nge. . . >

The action of individual gridpoint, zone or face keywords can be removed, or all conditions at a gridpoint, zone or face can be removed. For example, **APPLY remove gp** deletes all conditions at a gridpoint, while **APPLY remove sxx** deletes only the xx-stress component applied at faces. If **APPLY remove** is specified, all applied conditions are removed.

Rules

1. **APPLY** conditions for gridpoints, zones and faces are independent — e.g., applying face stresses does not affect the conditions at the associated face gridpoints.

2. New ***values*** replace existing ***values*** for the same keyword unless the **add** keyword precedes the ***value*** or the **history** keyword follows the ***value***. For example, at a gridpoint, a constant force (i.e., without **history**) and a step-varying force (i.e., with **history**) can be superimposed.

3. A condition can only be removed with the **remove** keyword.

4. Applied velocity conditions always refer to gridpoints, even if the velocity is prescribed in local axes with the keywords **dvelocity**, **svelocity** and **nvelocity**.

5. At each gridpoint, all prescribed velocities must be on the same axes: either global (***x, y, z***) axes or local axes.

6. Local-axes velocities cannot be applied to any gridpoints that are already **FIX**ed in any direction. A global-axes velocity cannot be applied to a gridpoint with a **FIX**ed velocity in the same direction (e.g., **APPLY xvel** is not compatible with **FIX x**).

7. Face stresses can be specified in the local axes in terms of normal stress **nstress** and shear stresses **dstress** and **sstress**.

8. At each face, all prescribed stresses must be applied on the same axes, either global (***x, y, z***) or local.

ATTACH keyword **ra**nge . . .

The **ATTACH** command allows the user to attach faces of sub-grids together rigidly to form a single grid. The faces of two sub-grids that are to be attached *must* be coplanar and touching (i.e., within a relative tolerance of 10^{-6}). However, there does not have to be the same number of gridpoints along each face.

The **ATTACH** command should be restricted to apply to faces within a specified range. The range can be given in several forms (see Section 1.1.1). If no range is given, then the command applies to the entire model. It is recommended that a separate **ATTACH** command be issued for each attached region.

The **ATTACH** command should be used carefully. For example, the **ATTACH** condition is not reflected if **GENERATE reflect** is specified for an **ATTACH**ed grid.

See Section 3.2.1.2 in Section 3 in the **User's Guide** for an example application and a discussion of limitations.

The following keywords are available to attach and detach faces.

delete Sub-grid faces within the specified range are unattached.

face Sub-grid faces within the specified range are attached.

The following keywords are available.

nosnap Gridpoints on opposing attached faces maintain their positions. (By default, gridpoints are "snapped" together.)

angtol ***value***

Faces are attached if the angle between them is less than ***value*** degrees. By default, the angle tolerance value = 0.5 degree.

tolerance ***value***

Faces are attached if the distance between the faces is less than ***value***. By default, the tolerance value = 10^{-6}.

Both the faces must be within both **tolerance** and **angtol** before they are attached.

The following keywords are available to attach individual gridpoints.

gp ***id1*** **gp** ***id2*** <**snap**>

Gridpoint with identification number ***id1*** is attached to gridpoint ***id2***. Gridpoints can be attached (i.e., slaved together) even though they are not located at the same physical location. If the optional keyword **snap** is given, gridpoint ***id1*** will be moved to the same location as gridpoint ***id2***.

gp ***id1*** **ed**ge ***id2 id3*** <**we**ight ***value***> <**snap**>

Gridpoint with identification number ***id1*** is attached to the edge defined as the line between gridpoints ***id2*** and ***id3***. The optional keyword **weight** can be given to define the location of gridpoint ***id1*** between gridpoints ***id2*** and ***id3***. If **weight** = 0.0, then ***id1*** is located at ***id2***; if **weight** = 1.0, then ***id1*** is located at ***id3***. If the optional keyword **snap** is given, then ***id1*** will be located between ***id2*** and ***id3*** automatically.

gp ***id1*** **fa**ce ***zid fid*** <**we**ight ***v1 v2 v3 v4***> <**snap**>

Gridpoint with identification number ***id1*** is attached to the face, face orientation number ***fid***, of the zone with identification number ***zid***. (Refer to Section 1.1.4 for a description of the orientation numbers.) The optional keyword **weight** can be given to define the location of gridpoint ***id1*** on face ***fid***, with respect to the face vertices. (Again, refer to Section 1.1.4 for a description of the orientation numbers.) If the optional keyword **snap** is given, then the location of ***id1*** on face ***fid*** will be done automatically.

CALL <filename>

A remote input file, filename, can be invoked with the **CALL** command. If no filename is given, the file "FLAC3D.DAT" is assumed. Any series of input instructions can be placed in this file so that *FLAC3D* will run unattended. **CALL** files can be nested. The **RETURN** command causes control to be returned to the user. The default extension ".DAT" is assumed if no extension is specified.

CONFIG keyword <keyword . . . >

This command allows the user to specify optional calculation modes that need extra memory to be assigned to each zone or gridpoint. The options are confined fluid flow, heat transfer, fully dynamic analysis and creep analysis. The **CONFIG** command can be given at any stage of an analysis, but it must be given before the calculation mode can be invoked.

The following keywords apply.

cppudm C++ user-defined models (only available with the C++ user-defined model option)

creep creep material analysis (only available with creep model option; see Section 2 in **Optional Features**)

dynamic fully dynamic analysis (only available with dynamic model option; see Section 3 in **Optional Features**)

fluid fluid-flow analysis (see Section 1 in **Fluid-Mechanical Interaction**)

gpextra ***n***

n extra gridpoint variables for *FISH* use (see Section 2 in the ***FISH* volume**)

thermal thermal analysis (only available with thermal model option; see Section 1 in **Optional Features**)

zextra ***n***

n extra zone variables for *FISH* use (see Section 2 in the ***FISH* volume**)

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. **CONTINUE** will then resume reading the data file on the next line. The **RETURN** command will have the same effect.

CYCLE ***n***

CYCLE is a synonym for **STEP**, in which ***n*** is the number of calculation steps to execute. Cycling may be interrupted by pressing the <SPACEBAR> or the <Esc> key. When in batch mode, the <SPACEBAR> will cause *FLAC3D* to stop cycling and skip to the next data line, whereas the <Esc> key will abort cycling and abort reading the data file.

DEFINE function-name

END

DEFINE and **END** are commands used to define a function written in *FISH*, the embedded language built into *FLAC3D*. Statements (described in Section 2 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 *FLAC3D* ; 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 to create new variables to display or plot (as histories), to control conditions during execution, to create special distributions of properties, or to analyze *FLAC3D* output in some special way. Section 4 in the **User's Guide** presents an introduction to *FISH*, and Section 2 in the ***FISH* volume** describes the operation and use of *FISH* in detail. Section 3 in the ***FISH* volume** contains a library of commonly used *FISH* functions.

DELETE **ra**nge . . .

All zones with centroids within the range defined by the **range** phrase are deleted. Zones can be deleted at any time in the model creation or solution. Unlike zones removed with the **MODEL null** command, deleted zones cannot be restored.

namedrange name

The named range name is deleted. Zones in the named range are *not* deleted.

EXPGRID filename

EXPGRID exports a *FLAC3D* grid to the named file, filename. A path can be part of the filename. The grid file is an ASCII file description of the *FLAC3D* geometry (zones, gridpoints, and zone groups). The grid file specification can be found with the **IMPGRID** command description. If no file extension is given, an extension of "FLAC3D" is used.

FIX keyword. . . <**ra**nge . . . >

With this command, velocity, pore pressure or temperature can be prevented from changing at selected gridpoints. If a fixed displacement is required, the appropriate velocities should be initialized to zero. (Zero velocity is the default on start-up.) Use the **APPLY** command to provide a rigid moving boundary condition. **FIX** may be specified over 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 following keywords may be used.

pp <***value***>

fixes pore pressures. If a ***value*** is given, the pore pressure is fixed at that value.

temperature <***value***>

fixes temperatures. If a ***value*** is given, the temperature is fixed at that value (for thermal model option only; see Section 1 in **Optional Features**).

xvelocity fixes x-velocity.

yvelocity fixes y-velocity.

zvelocity fixes z-velocity.

FREE keyword. . . <**ra**nge . . . >

This command releases the gridpoint constraint set by the **FIX** command. **FREE** may be specified over 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 following keywords may be used.

pp frees pore pressures.

temperature frees temperatures (for thermal model option only; see Section 1 in **Optional Features**).

xvelocity frees x-velocity.

yvelocity frees y-velocity.

zvelocity frees z-velocity.

GENERATE keyword ***value*** . . .

The **GENERATE** command creates a *FLAC3D* grid.* The grid can be "shaped" by the user into the geometry of the object being modeled. Grid generation procedure and example grids are described in Section 3 in the **User's Guide**.

Grid generation consists of five components, defined by the following keywords.

merge	ensures that separate grids created with **GEN zone** are connected, if gridpoints or adjacent faces are within a prescribed tolerance.
point	defines reference points in the 3D volume to assist with **zone** and **surface** generation.
separate	separates (unmerges) the zones belonging to the group name from the rest of the model.
surface	creates surfaces of a 3D volume.
zone	creates zones within the 3D volume.

The four components are generally invoked in the order given below. However, for model regions of regular shape, the **GENERATE zone** command can be used independently (or with the assistance of the **GENERATE point** and **GENERATE merge** commands). The keywords associated with each component are described below. See the **PLOT** command to create graphical plots of the *FLAC3D* grid, and the **PRINT generate** command to display output associated with grid generation.

* When running *FLAC3D* in interactive input mode, if the **GENERATE** command is entered alone at the `Flac3D>` prompt, then the `Gen>` prompt will appear. Any subsequent input then assumes that the **GENERATE** command was given first.

Surface Generation

surface keyword . . .

The surfaces of internal and external volumes are created with the **GENERATE surface** command. Keywords are available to define different surface segments. The segments can then be connected to create the desired shape. Surfaces can be viewed with the **PLOT volume** command. The following keywords are available to define surface segments.

brick eight-noded, six-sided brick closed volume

polygon single polygon surface

triangle single triangular surface

xarc arc surface segment extruded along a vector

xpolygon polygon surface extruded along a vector

The characteristics of the surface segments are defined by specifying the keywords below after using a surface segment keyword. Not all keywords apply to every surface segment. The application of these keywords for each surface segment is described in the rules below.

cap For **xpolygon** and **xarc** surfaces, **cap** creates surfaces at the two ends of the extruded polygon.

closed For **xpolygon** and **xarc** surfaces, **close** closes the two ends of the extruded polygon by connecting the first vertex entered to the last.

edge ***v***

For **brick** surfaces, **edge** specifies the default edge length, ***v***, of the brick.

extruded ***x y z***

For **xpolygon** and **xarc** surfaces, the vector defined by ***x, y, z*** specifies the direction and magnitude of the extruded surface.

segment ***i***

For **xarc** surfaces, the arc is divided into ***i*** segments. By default, ***i*** = 10.

vertex <**add**> ***x y z***
<**inc**rement> ***x y z***
<**po**int> ***n***

The keyword **vertex** locates a surface vertex at position (***x, y, z***). If the optional keyword **add** is given, the position is specified by adding ***x, y, z*** to the position of the first vertex. If the optional keyword **increment** is given, the position is specified by adding ***x, y, z*** to the position of the last vertex preceding this vertex. Alternately, the vertex can be specified by **point *n***, as defined previously by the **GENERATE point** command.

Rules

1. A **triangle** surface must be specified by three vertices.

2. Up to eight vertices may be specified to create the **brick** shape. At least four vertices must be given unless the **edge** keyword is used. When specifying the first four vertices, the vectors ***v1 - v2***, ***v1 - v3***, ***v1 - v4*** (see Figure 1.4, below), must form a right-handed coordinate system. The order of the eight vertices to form a brick are shown in Figure 1.4.

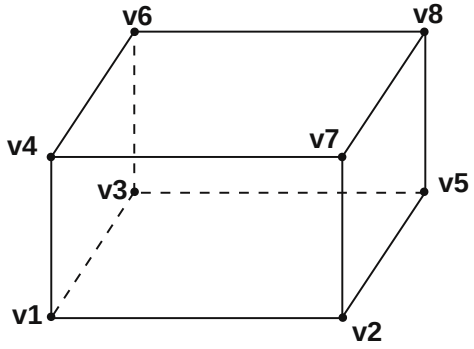


Figure 1.4 Brick volume defined by eight vectors

3. At least three vertices are required to define a **polygon** surface. The polygon is automatically subdivided into triangular surfaces by first creating an additional vertex at the centroid of the polygon, and then creating triangular surfaces defined by the centroid and each edge of the polygon. In this way, the vertices do not have to be coplanar. For best results, the polygon should be defined by vertices that create a closed convex surface (see Figure 1.5).

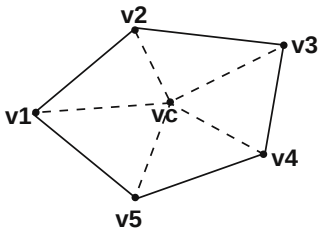


Figure 1.5 Polygon surface

4. An **xpolygon** surface is generated in two steps. First, a continuous edge is created by connecting vertices. At least three vertices are required. Then the surface is created by extruding lines from each vertex in the direction (with a magnitude) defined by the **extrude** vector (see Figure 1.6). For example, the vertices ***v1′***, ***v2′***, ***v3′*** and ***v4′*** are created automatically, in Figure 1.6, by adding the extrude vector to vertices ***v1***, ***v2***, ***v3*** and ***v4***. If the **close** keyword is given, vertices ***v1*** and ***v4*** and ***v1′*** and ***v4′*** will be connected. This will create a closed polygon surface (see Figure 1.7). If the **cap** keyword is specified, a vertex will be created at the centroid of each end, and surfaces will be created. If **close** and **cap** are both specified, a closed volume will be created (see Figure 1.8).

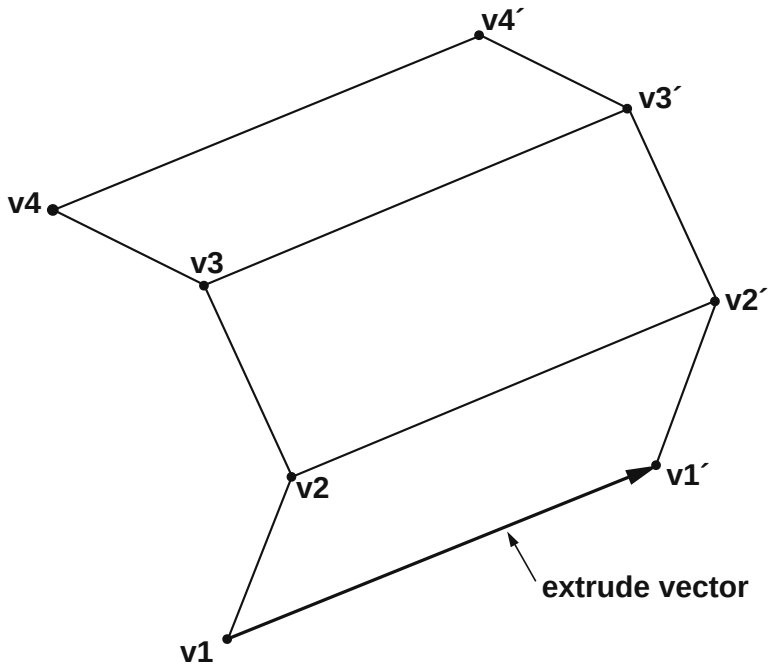


Figure 1.6 Extruded polygon surface

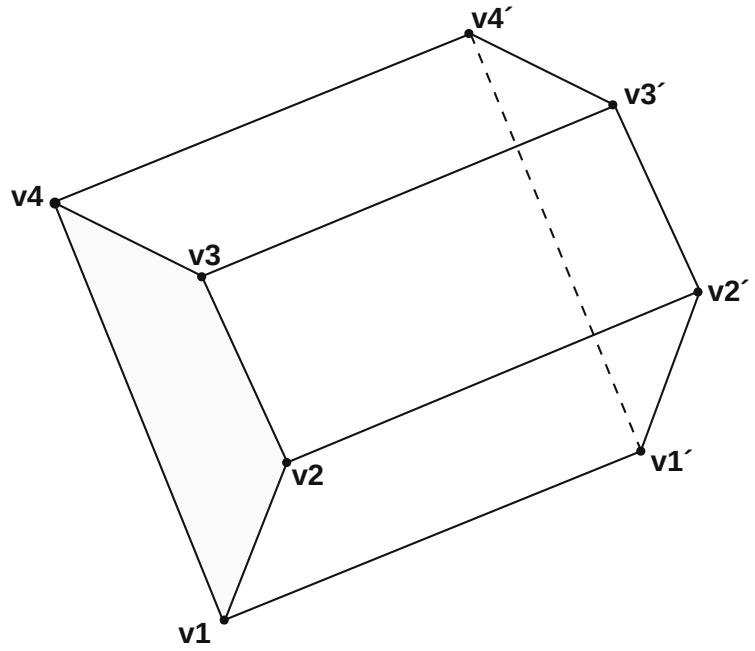


Figure 1.7 Extruded polygon surface with two ends closed (v1, v4, v1′ and v4′ form a surface)

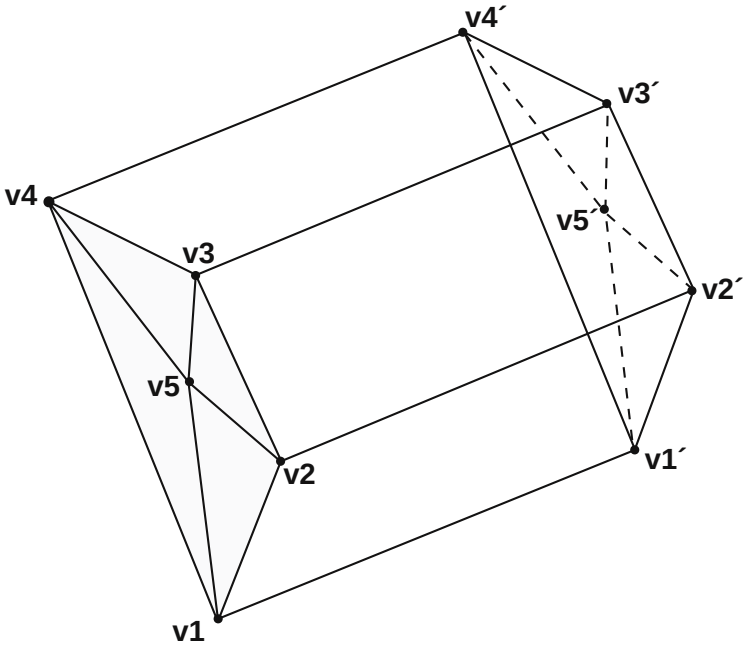


Figure 1.8 Extruded polygon surface with two ends closed and capped (v5 and v5′ are vertices at the centroids of the end surfaces)

5. An **xarc** surface is generated in two steps. First, an arc is created from three vertices. The first vertex locates the center of the arc, and the next two vertices locate the endpoints. Then, the surface is created by extruding lines from each vertex along the arc, with direction and magnitude defined by the **extrude** vector (see Figure 1.9). For example, the vertices ***v1′*** and ***v2′***, and the vertices defining the segments of the arc from ***v1′*** to ***v2′***, are created automatically by adding the extrude vector to vertices ***v1***, ***v2*** and the vertices defining the segments of the arc from ***v1*** to ***v2***. The **close** and **cap** keywords also work with the **arc** surface in the same way as with the extruded polygon surface (see Rule 4). If **close** is specified, a vertex is created at ***v0*** to generate the closed surface.

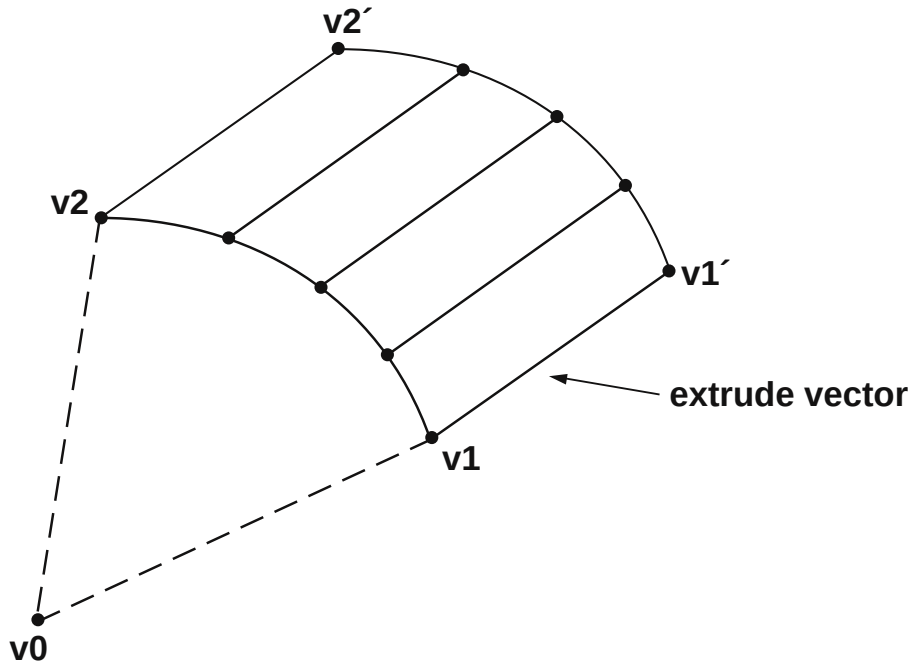


Figure 1.9 Arc polygon surface (for* segment *= 4)

Reference Point Definition

point <**id** ***n***> ***x y z***

Reference points are defined to assist with the zone-generation phase (see **GENERATE zone**, below). A point is located by its global coordinates (x, y, z). Each point is given a sequential number, based upon the order in which the **GENERATE point** commands are entered. A point can also be assigned an optional **id** number ***n***.

Zone Generation

zone keyword . . .

Zones are created within a 3D volume with the **GENERATE zone** command. This command can be used independently to create a zoned model of a specific primitive shape (or grid-connectivity type). The primitive shapes presently available are summarized in Table 1.3. They range from a simple brick shape to a complex tunnel intersection shape. Several **GENERATE zone** commands can be given to connect two or more primitive shapes together to build a *FLAC3D* grid. The commands **GENERATE zone copy** and **GENERATE zone reflect** are available to assist with creating a model composed of multiple shapes.

A primitive shape is selected by entering a keyword, as listed in Table 1.3, following **GENERATE zone**. The shapes are illustrated separately in Figures 1.10 through 1.22. The following keywords are available to define primitive mesh shapes.

brick	brick-shaped mesh
cshell	cylindrical shell mesh
cylinder	cylindrical-shaped mesh
cylint	cylinder intersection
dbrick	degenerate brick mesh
pyramid	pyramid-shaped mesh
radbrick	radially graded mesh around brick
radcylinder	radially graded mesh around cylindrical-shaped tunnel
radtunnel	radially graded mesh around parallelepiped-shaped tunnel
tetrahedron	tetrahedral-shaped mesh
tunint	tunnel intersection
uwedge	uniform wedge-shaped mesh
wedge	wedge-shaped mesh

Table 1.3 Summary of primitive mesh shapes

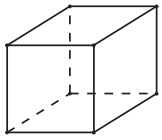
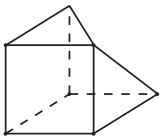
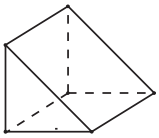
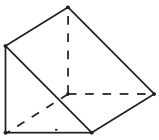
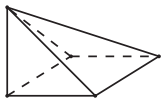
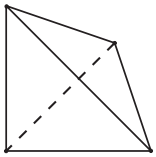
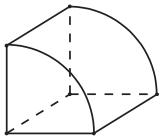
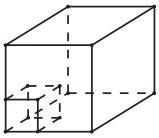
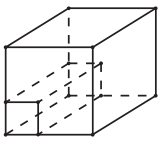
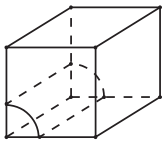
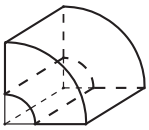
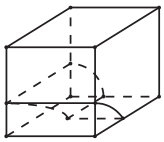
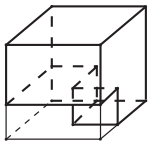
Shape	Name	Keyword	Reference Points	Size Entries	Dimension Entries	Fill
	**Brick**	**b**rick	**8**	**3**	**0**	**No**
	**Degenerate Brick**	**db**rick	**7**	**3**	**0**	**No**
	**Wedge**	**w**edge	**6**	**3**	**0**	**No**
	**Uniform Wedge**	**u**wedge	**6**	**3**	**0**	**No**
	**Pyramid**	**p**yramid	**5**	**3**	**0**	**No**
	**Tetrahedron**	**tet**rahedron	**4**	**3**	**0**	**No**
	**Cylinder**	**c**ylinder	**6**	**3**	**0**	**No**
	**Radial Brick**	**radb**rick	**15**	**4**	**3**	**Yes**
	**Radial Tunnel**	**rad**tunnel	**14**	**4**	**4**	**Yes**

Table 1.3 Summary of primitive mesh shapes (continued)

Shape	Name	Keyword	Reference Points	Size Entries	Dimension Entries	Fill
	**Radial Cylinder**	**rad**cylinder	**12**	**4**	**4**	**Yes**
	**Cylindrical Shell**	**c**shell	**10**	**4**	**4**	**Yes**
	**Cylinder Intersection**	**cylint**	**14**	**5**	**7**	**Yes**
	**Tunnel Intersection**	**t**unint	**17**	**5**	**7**	**Yes**

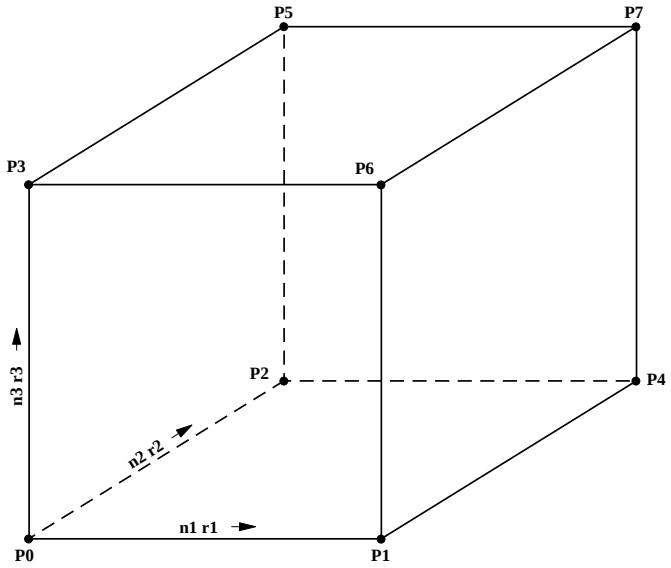


Figure 1.10 Brick mesh — brick

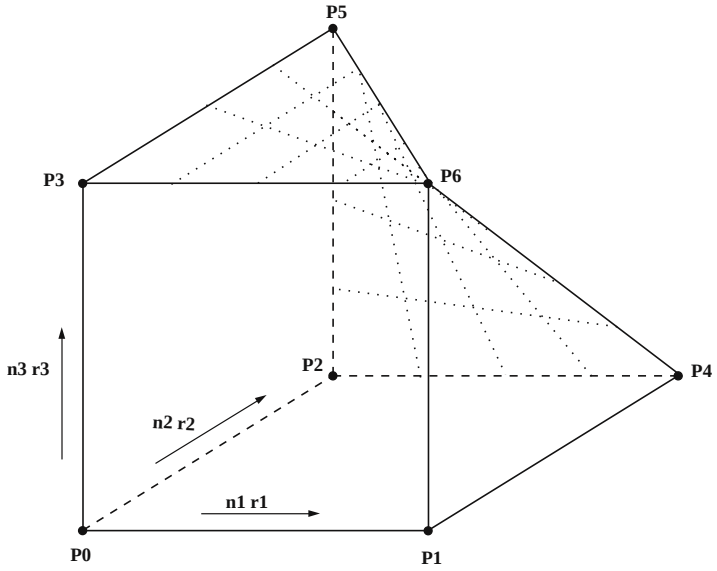


Figure 1.11 Degenerate brick mesh — dbrick

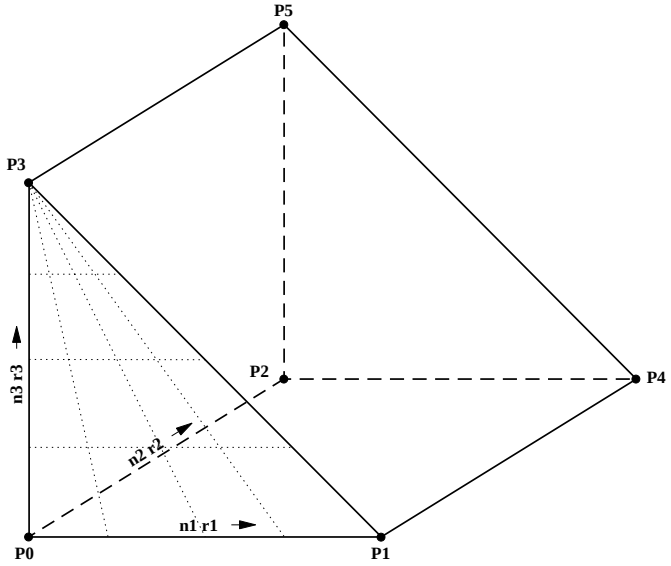


Figure 1.12 Wedge mesh — wedge

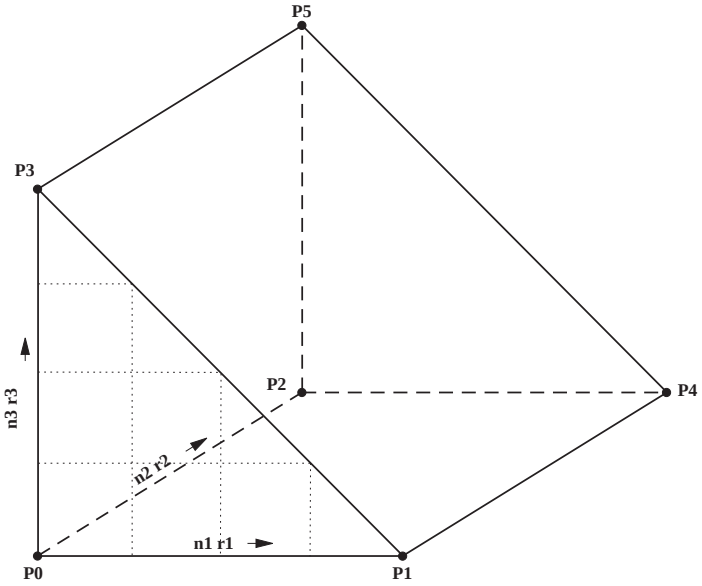


Figure 1.13 Uniform wedge mesh — uwedge

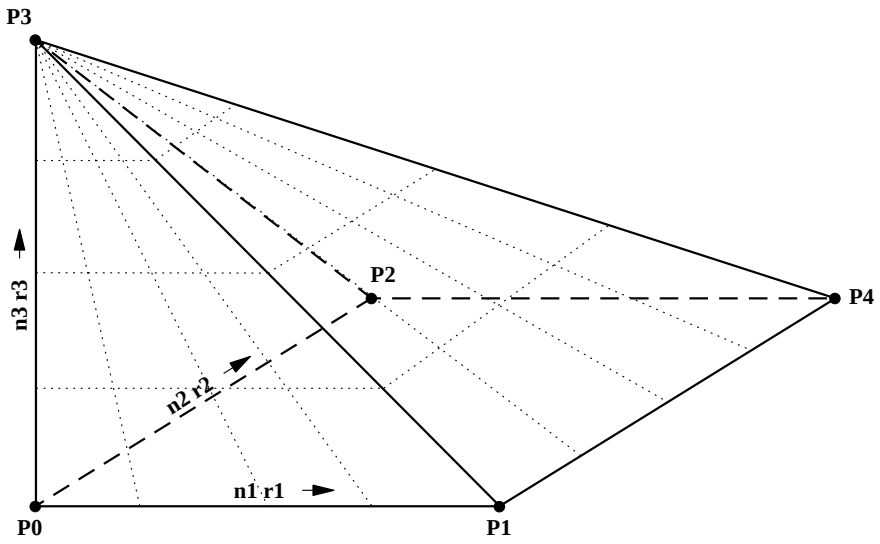


***Figure 1.14 Pyramid mesh —* pyramid**

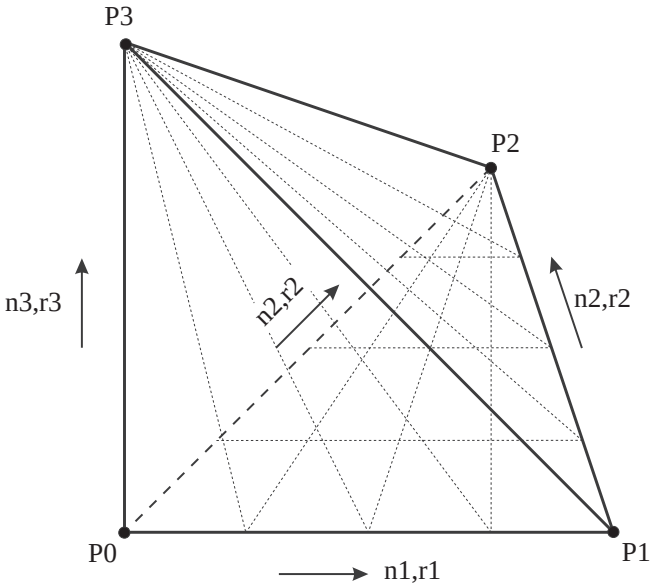


***Figure 1.15 Tetrahedron —* tet**

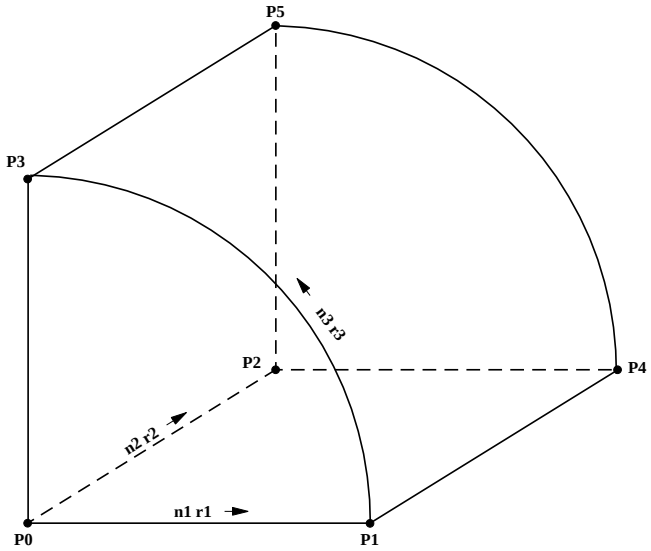


Figure 1.16 Cylindrical mesh — **cylinder**

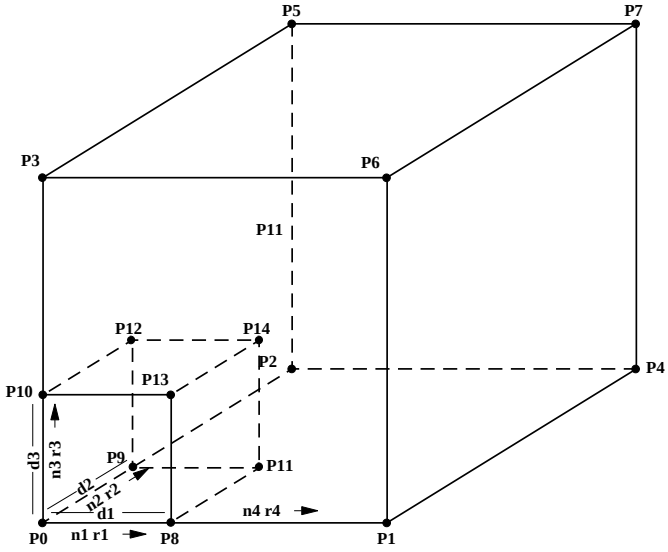


Figure 1.17 Radially graded mesh around brick — **radbrick**

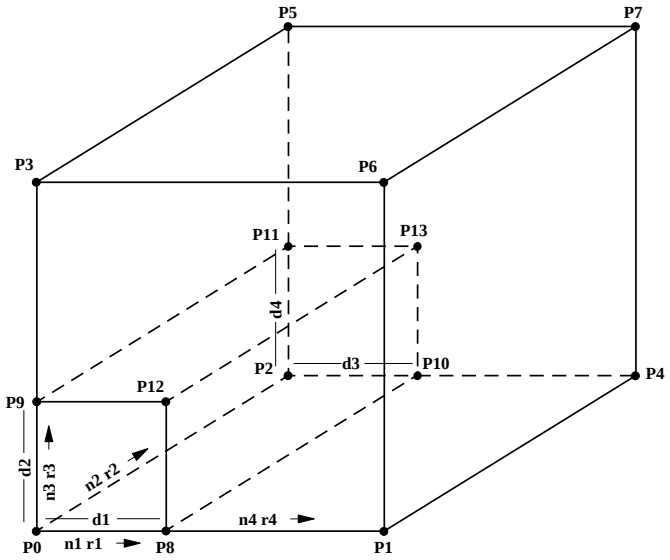


***Figure 1.18 Radially graded mesh around parallelepiped-shaped tunnel —* radtunnel**

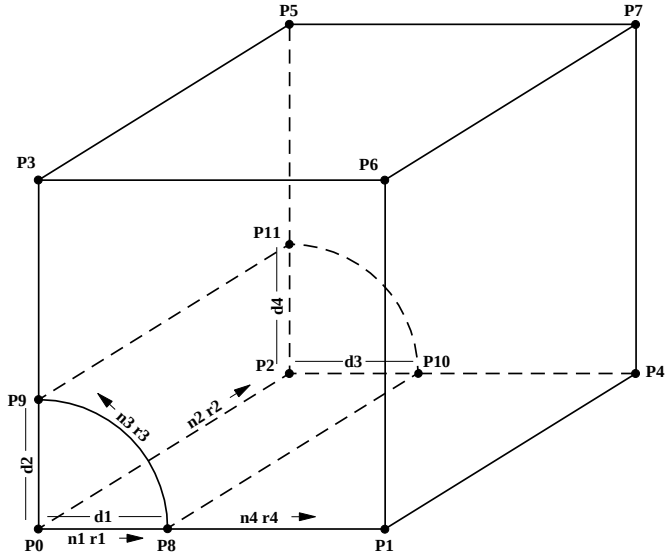


***Figure 1.19 Radially graded mesh around cylindrical-shaped tunnel —* radcylinder**

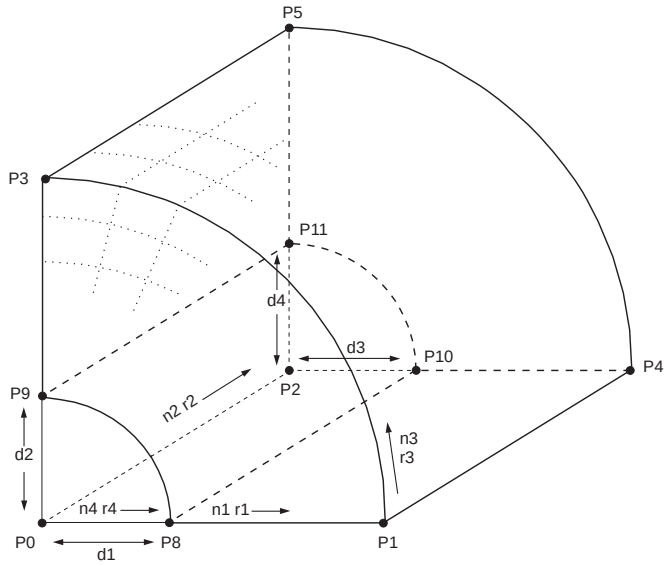


Figure 1.20 Cylinder shell mesh — cshell

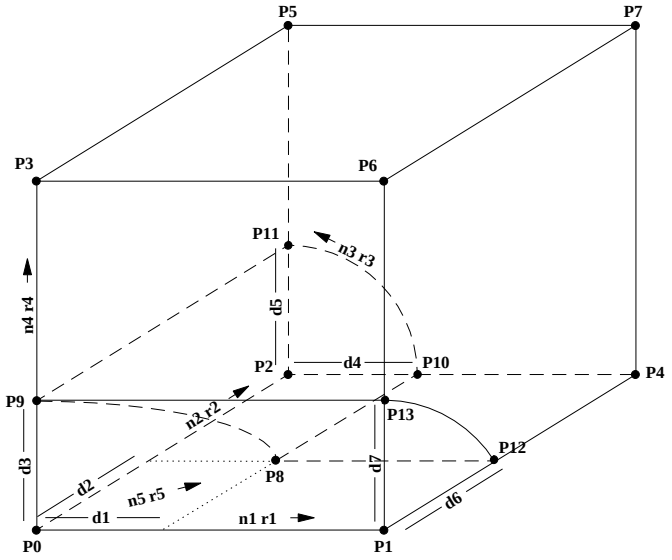


Figure 1.21 Cylinder intersection — cylint

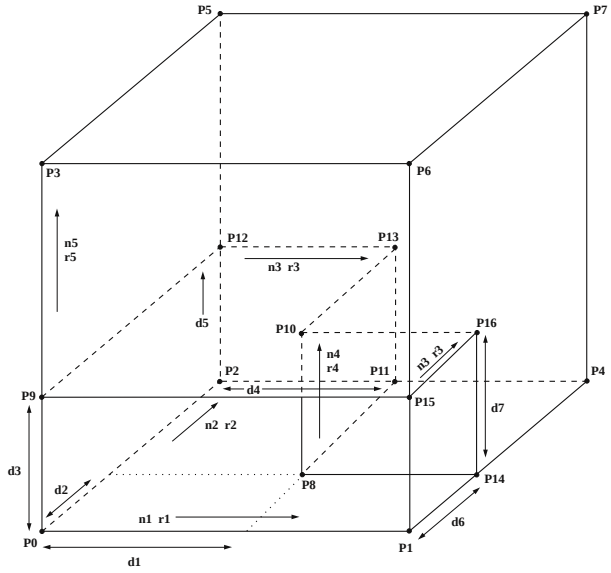


***Figure 1.22 Tunnel intersection —* tunint**

keyword The characteristics of the shape (e.g., global coordinate positions, number of zones) are defined by specifying the keywords given below, *after* the shape keyword. Not all keywords apply to every shape. The numerical entries associated with each shape are listed in Table 1.3, and illustrated in Figures 1.10 through 1.22.

dimension ***d1*** <***d2***> <***d3***> <***d4***> <***d5***> <***d6***> <***d7***>

specifies the dimensions of interior regions for the shapes: radial tunnel, radial cylinder, radial brick, cylinder shell, cylinder intersection and tunnel intersection. Not all dimension entries (***d1***, ***d2***, ***d3***, ***d4***, ***d5***, ***d6***, ***d7***) are required for every shape. The entries and definitions for each shape are shown in Figures 1.17 through 1.22, respectively. If **dimension** is not given, default values for the entries are calculated as 20% of the lengths between reference points.

edge ***value***

specifies an edge length for the sides of the mesh. If **p1**, **p2** and **p3** are not specified, then the magnitude of the distance from **p0** will be defined by **edge**.

fill <**gro**up groupname>

If **fill** is specified, the interior region for the shape's radial tunnel, radial cylinder, radial brick, cylinder intersection and tunnel intersection will be filled with zones. If not specified, the interior region will not contain zones. If the optional **group** keyword is given, with a valid groupname, then groupname is assigned to the filled zones. A group will be created if none exists.

group name

assigns a group name to this primitive at creation.

nomerge Gridpoints on the outer faces of the mesh are not merged with existing mesh faces.*

p0		<*x y z*>	<**po**int ***n***>
p1	<**add**>	<*x y z*>	<**po**int ***n***>
p2	<**add**>	<*x y z*>	<**po**int ***n***>
p3	<**add**>	<*x y z*>	<**po**int ***n***>
p4	<**add**>	<*x y z*>	<**po**int ***n***>
p5	<**add**>	<*x y z*>	<**po**int ***n***>
p6	<**add**>	<*x y z*>	<**po**int ***n***>
p7	<**add**>	<*x y z*>	<**po**int ***n***>
p8	<**add**>	<*x y z*>	<**po**int ***n***>
p9	<**add**>	<*x y z*>	<**po**int ***n***>
p10	<**add**>	<*x y z*>	<**po**int ***n***>
p11	<**add**>	<*x y z*>	<**po**int ***n***>
p12	<**add**>	<*x y z*>	<**po**int ***n***>
p13	<**add**>	<*x y z*>	<**po**int ***n***>
p14	<**add**>	<*x y z*>	<**po**int ***n***>
p15	<**add**>	<*x y z*>	<**po**int ***n***>
p16	<**add**>	<*x y z*>	<**po**int ***n***>

The keywords **p0**, **p1**, . . . **p16** specify the reference (corner) points of the shapes. If the **add** keyword is used, the current position for **p0** is added to the location specified. By default, **p0** is located at (***x*** = 0, ***y*** = 0, ***z*** = 0). By default, **p1**, **p2** and **p3** are set to the orthogonal distances in the x-, y- and z-directions, with a magnitude equal to the number of zones in that direction. When specifying **p1**, **p2** and **p3**, the vectors **p0 - p1**, **p0 - p2** and **p0 - p3** *must* form a right-handed coordinate system. Alternatively, **point *n*** may be specified, as previously defined by the **GENERATE point** command.

Not all points are required for every shape. The number required is listed in Table 1.3. The point locations corresponding to these

* During execution of a **GENERATE zone** command, a check is made for each boundary gridpoint against the boundary gridpoints of zones that already exist. Internal gridpoints are not checked. If two boundary gridpoints fall within a tolerance of 1×10^{-7} (relative to the magnitude of the gridpoints' position vector) of each other, they are assumed to be the same point, and the first gridpoint is used rather than creating a new one for all subsequent calculations. If **nomerge** is specified, two gridpoints are not merged into one.

keywords are given in Figures 1.10 through 1.22.

ratio *r1* <*r2*> <*r3*> <*r4*> <*r5*>

This specifies a ratio that is used to space zones with an increasing or decreasing geometric ratio. Not all ratio entries (***r1***, ***r2***, ***r3***, ***r4***, ***r5***) are required for every shape. For each shape, the entries and their associated zone direction are shown in Figures 1.10 through 1.22. If **ratio** is not given, all entries default to 1.0.

size *n1* <*n2*> <*n3*> <*n4*> <*n5*>

This specifies the number of zones for each shape. Not all entries (***n1***, ***n2***, ***n3***, ***n4***, ***n5***) are required for every shape. The number required is listed in Table 1.3. The entries and their corresponding direction for each shape are shown in Figures 1.10 through 1.22. If **size** is not specified, all entries default to 10.

There are two **GENERATE zone** keywords available to assist the zone generation phase: **copy** and **reflect**. A range can be specified (see Section 1.1.3) to limit the range of these keywords.

copy *x y z* <**ra**nge. . . >

With the **copy** keyword, all zones within the given range are copied to a new position by adding the offset ***x, y, z***.

reflect keyword. . . <**ra**nge. . . >

With the **reflect** keyword, all zones within the given range are reflected across the plane specified by the following keywords.

dd *value*

This specifies the dip direction, ***value***, of the plane measured in the global xy-plane clockwise from the positive y-axis. (The default is ***value*** = 0.)

dip ***value***

This specifies the dip angle, ***value***, of the plane, measured in the negative z-direction from the global xy-plane. (The default is ***value*** = 0.)

normal ***xv yv zv***

This specifies a normal vector to the plane, with the components ***xv***, ***yv*** and ***zv***. If **normal** is specified, **dip** and **dd** are not required.

origin ***xv yv zv***

This specifies the location of one point through which the plane crosses. The coordinates of the point are ***xv***, ***yv*** and ***zv***.

A warning message will be given if gridpoints are reflected across both sides of the reflection plane.

All primitive mesh shapes except the **dbrick** shape can be reflected.

Merge Separate Grids

merge ***vtol*** <**ra**nge>

Gridpoints on separate grids that fall inside a specified range, ***vtol***, are merged into a single gridpoint. Surface faces are merged into interior zone edges if all gridpoints on each face have been merged.

The number of gridpoints and surface faces that are removed from the model are reported after merging is complete.

Separate Zones

separate ***gname***

separates (unmerges) the zones belonging to group ***gname*** from the rest of the model. If a zone in group ***gname*** shares a face with a zone belonging to another group (including the null group), then the gridpoints on this face are duplicated (so the face is no longer shared). This command can be used in conjunction with the **INTERFACE wrap** command.

If you have a history or other item associated with a gridpoint in group ***gname***, make certain it still has the same association after this command is issued (this command will make new gridpoints for shared faces in ***gname***).

GEOM_TEST <**ra**nge. . . >

FLAC3D has three basic tests built-in to check the integrity of meshed models to make sure that the model is adequate for simulation purposes. For example, these tests can be used to check whether there is improper mapping of node points during execution of a grid-generation data file in *FLAC3D*, resulting in some zones being inside out and, hence, not suitable for simulation. It can also be used to check whether a zone is degenerate. Degeneracy can occur if a primitive is created without satisfying the requirements of geometry conditions, such as the number of vertices, edges and faces for that particular primitive. The geometry tests in *FLAC3D* are designed for hexahedral elements only.

The geometric aspects of a hexahedral element are evaluated using three quantities: *orthogonality*, *aspect ratio* and *face planarity*. The quantities compare the hexahedrons to a perfect cube, which is the ideal shape for hexahedral meshes. The **GEOM_TEST** command invokes the test for all three of these geometric quantities.

Orthogonality — For each gridpoint in each zone, the determinant of the matrix defined by the three edge vectors is computed and divided by the product of their lengths. This gives 1.0 for a cube, and approaches zero as pairs of edges approach being coplanar or all three approach being coplanar. Each zone is measured by the worst orthogonality value of all gridpoints.

Aspect Ratio — For each gridpoint, the ratio of the shortest edge length by the longest edge length is computed. This will be 1.0 for a cube. It will approach zero as the zone becomes stretched or flattened. Each zone is measured by the worst aspect ratio of all gridpoints.

Face Planarity — Hexahedrons are composed of 6 quadrilateral faces, just like a cube's square faces. However, it is possible that the four vertices of a quadrilateral polygon will not be coplanar in 3D. *FLAC3D* allows faces to be non-planar but, the greater the deviation, the less accurate the solution process will be. There is no clear singular method of measuring planarity. A method that compares the volume of a tetrahedron filling the 4 vertices, and the area of the quadrilateral face, computing the area by adding a central point, m = (A+B+C+D)/4, and computing the 4 triangle areas, ABC, ABD, ACD, BCD, is chosen. The ratio of the cube root of the volume to the square root of the area (to get a dimensionless value) is computed. This value is zero if planar, and positive if non-planar. This test can be "scaled" by a constant because there is no fundamental limit on how non-planar a face can be. (Values should be $<<$ 1.0 for a face to be good.) Each zone has its six faces tested and the worst value is reported.

Reporting average values for these tests are meaningless because some meshes might be 90% good and 10% bad and give the same average as a mesh with an 80% test quality. Instead, distribution of values for all of the three parameters across all zones in the range specified is reported. The distribution ranges from -1.0 to 1.0 with a 0.1 interval. Thus, all zones fall within these 20 intervals. This distribution helps identify the number of bad zones, and how bad these zones are. Note that these tests

are useful for relative comparisons between different grids for the same geometry. For a model to perform well with *FLAC3D*, orthogonality and aspect ratio zone test values should be near 1.0 and planarity test values near 0.0. *FISH* functions can be written to group all zones that failed to meet minimum standards, and thus visualize the bad zones.

GP {**id** = *gpid*} *x y z*

This command creates a single new gridpoint at position *x, y, z*. If **id** is specified, then the gridpoint is assigned this ID number, provided this ID number is not already in use by another gridpoint. If **id** is not specified, then the gridpoint is assigned the next available gridpoint ID number. Also see the **ZONE** command.

GROUP name <**remainder**> <**none**> name <color> <**ra**nge. . . >

Groups are sets of zones identified by a group name. Groups may not overlap; each zone may only belong to one group. A new group name will replace an old name. The **GROUP** command assigns a name to all zones and gridpoints 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. If a group name of none (case insensitive) is specified, then the zone and its associated gridpoints are unassigned from their current group and the zone no longer belongs to a group.

The command **PRINT group** lists the existing group names. The command **PRINT zone** gives the group name for each zone.

The **remainder** keyword assigns group name to all null group zones (zones that don't have a group assigned to them) that also fall within the range.

The keyword **none** unassigns groups — e.g.,

```
group none range group granite
```

A color can be assigned to a group with the optional color switch keyword (see Table 1.7). The color, if specified, is used in the **PLOT block group** command instead of automatic color selection.

HISTORY <**id** ***nh***> <**n**step = ***n***> keyword . . . ***x y z***

or

HISTORY <**id** ***nh***> <**n**step = ***n***> keyword . . . **id** = ***n***

The values of a variable are sampled and stored during a model run by using the **HISTORY** command. Histories can be plotted versus step number or versus other histories (see the **PLOT history** command). Selected histories can also be dumped to a file (with the **HISTORY write** command). Only one variable may be given per **HISTORY** command. History variables may be added at any time, but not deleted — *all* current histories can be erased (with the **HISTORY reset** command). Each history is given a sequential number, based upon the order in which the **HISTORY** commands are received. A history number **id** can also be assigned with the optional value ***nh***. A summary of requested histories can be displayed at any time, by using the **PRINT history** or **HISTORY print** command. By default, values for histories are sampled every ten timesteps unless the repeat cycle is changed with the **nstep** keyword. The location of a gridpoint, zone or structural element history is defined by its (***x, y, z***) — coordinate positions or **id** number ***n***. Either of these *must* be given last on the input line for gridpoint, zone or structural element histories.

Valid keywords for a history are as follows.

gp keyword

Certain gridpoint variables can be sampled. The variables are identified by the following keywords.

displacement	displacement magnitude
force	force magnitude
ppressure	pore pressure at gridpoint
temperature	temperature (for thermal option only — see Section 1 in **Optional Features**)
velocity	velocity magnitude
xdisplacement	x-displacement
xforce	x-unbalanced force
xvelocity	x-velocity
ydisplacement	y-displacement
yforce	y-unbalanced force
yvelocity	y-velocity

zdisplacement z-displacement

zforce z-unbalanced force

zvelocity z-velocity

interface ***n*** keyword

Certain interface variables can be sampled for interface ***n***. The variables are defined by the following keywords.

ndisplacement normal displacement

nstress normal stress

sdisplacement shear displacement

sstress shear stress

ratio ratio of maximum unbalanced force (or heat flux or fluid flow) versus applied force (or heat flux or fluid flow) as specified by the **SET ratio** command.

sel keywords

The following SEL histories are described: **beamsel**, **cablesel**, **geogridsel**, **linersel**, **node**, **pilesel**, **recover** and **shellsel**.

beamsel keyword . . .

samples beam responses that include force and moment carried by the beam. The force and moment vectors are expressed in the beamSEL local coordinate system — see Figure 1.25 for sign convention. If a beam is created using the **BEAM** command, then the nodes of each beamSEL so created will be ordered such that the overall beam direction goes from the **begin** point to the **end** point — i.e., the nodal connectivity of each beamSEL will be ordered such that the direction from end-1 to end-2 corresponds with the direction from the **begin** point to the **end** point.

The following keywords are available.

force keyword <end> ***x y z***

keyword <end> **cid** = ***cid***

The particular beamSEL is identified either by (***x,y,z***) coordinates (the nearest beamSEL is taken) or by the SEL CID number ***cid***. The end at which the quantity will be sampled is designated by the keyword end ∈ {**end1, end2**}, which corresponds with the order in the nodal-connectivity list. If end is not specified, then end is set equal to **end1**. The following force components are sampled based upon the keywords:

Fx force (x-component, beamSEL system)

Fy force (y-component, beamSEL system)

Fz force (z-component, beamSEL system)

moment keyword <end> ***x y z***

keyword <end> **cid** = ***cid***

The particular beamSEL is identified either by (***x,y,z***) coordinates (the nearest beamSEL is taken) or by the SEL CID number ***cid***. The end at which the quantity will be sampled is designated by the keyword end ∈ {**end1, end2**}, which corresponds with the order in the nodal-connectivity list. If end is not specified, then end is set equal to **end1**. The following moment components are sampled based upon the keywords:

Mx moment (x-component, beamSEL system)

My moment (y-component, beamSEL system)

Mz moment (z-component, beamSEL system)

cablesel keyword . . .

sample cable responses, including: cable element force, stress and yield state; and shear coupling spring displacement, stress and slip state

If a cable is created using the **CABLE** command, then the nodes of each cableSEL so created will be ordered such that the overall cable direction goes from the **begin** point to the **end** point — i.e., the nodal connectivity of each cableSEL will be ordered such that the direction from end-1 to end-2 corresponds with the direction from the **begin** point to the **end** point.

1. Cable response quantities

keyword ***x y z***

keyword **cid** = ***cid***

The particular cableSEL is identified either by (***x,y,z***) coordinates (the nearest cableSEL is taken) or by the SEL CID number ***cid***. The following cable responses can be sampled.

force average axial force in cable (positive: tension; negative: compression)

stress average axial stress in cable (positive: tension; negative: compression)

yield <**c**ompression>

Yield state of cableSEL. Value {0, 1, 2} denotes: never yielded, now yielding or yielded in the past, respectively. If **compression** is specified, then the compressive yield state is sampled; otherwise, the tensile yield state is sampled.

2. Grout response quantities

grout keyword <end> ***x y z***

keyword <end> **cid** = ***cid***

The particular cableSEL is identified either by (***x,y,z***) coordinates (the nearest cableSEL is taken) or by SEL CID number ***cid***. The end at which the quantity will be sampled is designated by keyword end ∈ {**end1, end2**} which corresponds with the order in the nodal-connectivity list. If end is not specified, then end is set equal to **end1**.

disp Displacement in grout (shear coupling spring) at end of cable-SEL. Sign convention is +/- with respect to average axial direction of cable.

slip Slip state in grout (shear coupling spring) at end of cableSEL. Value {0, 1, 2} denotes: never slipped, now slipping or slipped in past, respectively.

stress Stress in grout (shear coupling spring) at end of cableSEL. Sign convention is +/- with respect to average axial direction of cable.

geogridsel keyword . . .

samples geogrid responses that include displacement, stress and yield state in the coupling springs. The stresses in the geogrid can be sampled by the **HISTORY sel recover** history item. If a geogrid is created using the **GEOGRID** command, then the nodes of each geogridSEL will be ordered counterclockwise with respect to the outside of the zone faces upon which the geogrid was created. The available keywords follow.

coupling keyword <node> ***x y z***

keyword <node> **cid** = ***cid***

The particular geogridSEL is identified either by (***x,y,z***) coordinates (the nearest geogridSEL is taken) or by the SEL CID number ***cid***. The node at which the quantity will be sampled is designated by the keyword node ∈ {**nd1, nd2, nd3**}, which corresponds with the order in the nodal-connectivity list — see **SEL print geogrid conn**. If node is not specified, then node is set equal to **nd1**.

disp Displacement magnitude (always positive) in coupling spring at node of geogridSEL.

stress Stress magnitude (always positive) in coupling spring at node of geogridSEL.

yield Yield state in coupling spring at node of geogridSEL. Value {0, 1, 2} denotes: never yielded, now yielding or yielded in past, respectively.

linersel keyword . . .

samples liner responses that include displacement, stress and yield state in both the normal and shear coupling springs. The stresses in the liner itself can be sampled by the **HISTORY sel recover** history item. If a liner is created using the **LINER** command, then the nodes of each linerSEL so created will be ordered counterclockwise with respect to the outside of the zone faces upon which the liner was created. The available keywords follow.

coupling keyword <node> ***x y z***

keyword <node> **cid** = ***cid***

The particular linerSEL is identified either by (***x,y,z***) coordinates (the nearest linerSEL is taken) or by the SEL CID number ***cid***. The node at which the quantity will be sampled is designated by the keyword node ∈ {**nd1, nd2,**

nd3}, which corresponds with the order in the nodal-connectivity list — see **SEL print liner conn**. If node is not specified, then node is set equal to **nd1**.

disp ***dir***

Displacement in ***dir*** coupling spring at node of linerSEL where ***dir*** ∈ {**normal, shear**}. Sign convention for normal spring is +/-, indicating separation/overlap. Shear spring stores displacement magnitude and is always positive.

stress ***dir***

Stress in ***dir*** coupling spring at node of linerSEL where ***dir*** ∈ {**normal, shear**}. Sign convention for normal spring is +/-, indicating separation/overlap. Shear spring stores displacement magnitude and is always positive.

yield **shear**

Yield state in shear coupling spring at node of linerSEL. Value {0, 1, 2} denotes: never yielded, now yielding or yielded in past, respectively. The yield state of the normal coupling spring cannot be sampled.

node keyword . . .

samples nodal responses that include displacement, velocity, position and out-of-balance force.

keyword <**l**ocal> ***x y z***

<**l**ocal> **id** = ***id***

The particular node is identified either by (***x,y,z***) coordinates (the nearest node is taken) or by node ID number ***id***. The sampled quantity is expressed in the global system, unless the keyword **local** is specified, in which case it is expressed in the node-local system. Note that the

local keyword cannot be specified for the position histories, which are always expressed in the global system. The available keywords follow.

xdisp translational displacement (x-component, global or node-local system)

xfob translational out-of-balance force (x-component, global or node-local system)

xpos current position (x-component, global system)

xrdisp rotational displacement (x-component, global or node-local system)

xrfob rotational out-of-balance force (x-component, global or node-local system)

xrvel rotational velocity (x-component, global or node-local system)

xvel translational velocity (x-component, global or node-local system)

ydisp translational displacement (y-component, global or node-local system)

yfob translational out-of-balance force (y-component, global or node-local system)

ypos current position (y-component, global system)

yrdisp rotational displacement (y-component, global or node-local system)

yrfob rotational out-of-balance force (y-component, global or node-local system)

yrvel rotational velocity (y-component, global or node-local system)

yvel translational velocity (y-component, global or node-local system)

zdisp translational displacement (z-component, global or node-local system)

zfob translational out-of-balance force (z-component, global or node-local system)

zpos current position (z-component, global system)

zrdisp rotational displacement (z-component, global or node-local system)

zrfob rotational out-of-balance force (z-component, global or node-local system)

zrvel rotational velocity (z-component, global or node-local system)

zvel translational velocity (z-component, global or node-local system)

pilesel keyword . . .

samples pile responses that include force and moment acting on the pile itself, and displacement, stress and yield state in both the normal and shear coupling springs.

If a pile is created using the **PILE** command, then the nodes of each pileSEL so created will be ordered such that the overall pile direction goes from the **begin** point to the **end** point — i.e., the nodal connectivity of each pileSEL will be ordered such that the direction from end-1 to end-2 corresponds with the direction from the **begin** point to the **end** point.

1. Coupling-spring quantities

Coupling-spring quantities include the displacement, stress and yield state of both the normal and shear coupling springs.

coupling keyword dir <end> ***x y z***

keyword dir <end> **cid** = ***cid***

The particular pileSEL is identified either by (***x,y,z***) coordinates (the nearest pileSEL is taken) or by the SEL CID number ***cid***. The coupling spring is designated by the keyword dir ∈ {**normal, shear**}. The end at which the quantity will be sampled is designated by the keyword end ∈ {**end1, end2**}, which corresponds with the order

in the nodal-connectivity list. If end is not specified, then end is set equal to **end1**.

disp Displacement in dir coupling spring at end of pileSEL. Sign convention for shear spring is +/- with respect to average axial direction of pile. Sign convention for normal spring is +/-, indicating separation/overlap.

stress Stress in dir coupling spring at end of pileSEL. Sign convention for shear spring is +/- with respect to average axial direction of pile. Sign convention for normal spring is +/-, indicating separation/overlap.

yield Yield state in dir coupling spring at end of pileSEL. Value {0, 1, 2} denotes: never yielded, now yielding or yielded in past, respectively.

2. Force and moment quantities

force keyword <end> ***x y z***

keyword <end> **cid** = ***cid***

The particular pileSEL is identified either by (***x,y,z***) coordinates (the nearest pileSEL is taken) or by the SEL CID number ***cid***. The end at which the quantity will be sampled is designated by end ∈ {**end1**, **end2**}, which corresponds with the order in the nodal-connectivity list. If end is not specified, then end is set equal to **end1**. The forces are expressed in the pileSEL coordinate system — see Figure 1.25 for sign convention. The following force components are sampled based upon the keywords:

Fx force (x-component, pileSEL system)

Fy force (y-component, pileSEL system)

Fz force
(z-component, pileSEL system)

moment keyword <end> ***x y z***

keyword <end> **cid** = ***cid***

The particular pileSEL is identified either by (***x,y,z***) coordinates (the nearest pileSEL is taken) or by the SEL CID number ***cid***. The end at which the quantity will be sampled is designated by the keyword end ∈ {**end1, end2**}, which corresponds with the order in the nodal-connectivity list. If end is not specified, then end is set equal to **end1**. The moments are expressed in the pileSEL coordinate system — see Figure 1.25 for sign convention. The following moment components are sampled based upon the keywords:

Mx moment (x-component, pileSEL system)

My moment (y-component, pileSEL system)

Mz moment (z-component, pileSEL system)

recover keyword . . .

provides a mechanism to sample stresses and stress resultants at the centroids of all shell-type SELs. Nodal averaging does not occur, because each time the history is sampled, the logic of the **SEL recover sres** or **SEL recover stress** command is invoked for this SEL only. If more complex recover is necessary, it must be performed directly (for example, by establishing a consistent surface system and recovering quantities over a specified patch of shell-type SELs), and the results must be accessed via *FISH*.

1. Displacement histories

Displacement histories are obtained through the **HISTORY sel node** command.

2. Stress resultant histories

sres keyword **s**urfX ***Xx Xy Xz x y z***

keyword **s**urfX ***Xx Xy Xz*** **cid** = ***cid***

The particular shell-type SEL is identified either by (***x,y,z***) coordinates (the nearest shell-type SEL is taken) or by SEL CID number ***cid***. Stress resultants are expressed in the surface system associated with the specified shell-type SEL. The surface system corresponds with the given value of **surfx** — i.e., each time a stress-resultant history is sampled, the logic of the **SEL recover surface** command is invoked using **surfx** for the specified shell-type SEL. The following stress resultants are sampled based upon the keywords:

Mx stress resultant M_x (surface system of SEL)

My stress resultant M_y (surface system of SEL)

Mxy stress resultant M_{xy} (surface system of SEL)

Nx stress resultant N_x (surface system of SEL)

Ny stress resultant N_y (surface system of SEL)

Nxy stress resultant N_{xy} (surface system of SEL)

Qx stress resultant Q_x (surface system of SEL)

Qy stress resultant Q_y (surface system of SEL)

3. Stress histories

Stresses are computed at a shell depth given by the **depth_fac** keyword as follows. The shell depth equals $\frac{Ft}{2}$, where F is the depth factor and t is the shell thickness. F equal to +1/-1 corresponds with the outer/inner

shell surface (outer surface defined by positive shell z-direction), and $F = 0$ corresponds with the shell mid-surface.

stress keyword **d**epth_fac ***v*** ***x*** ***y*** ***z***

keyword **d**epth_fac ***v*** **cid** = ***cid***

The particular shell-type SEL is identified either by (***x,y,z***) coordinates (the nearest shell-type SEL is taken) or by SEL CID number ***cid***. Stresses are expressed in the global system and correspond with a shell depth given by the depth factor ***v***. The following stresses are sampled based upon the keywords:

xx stress (xx-component, global system)

yy stress (yy-component, global system)

zz stress (zz-component, global system)

xy stress (xy-component, global system)

zy stress (zy-component, global system)

zx stress (zx-component, global system)

pstress keyword **d**epth_fac ***v*** ***x*** ***y*** ***z***

keyword **d**epth_fac ***v*** **cid** = ***cid***

The particular shell-type SEL is identified either by (***x,y,z***) coordinates (the nearest shell-type SEL is taken), or by SEL CID number ***cid***. Principal stresses (σ_1, σ_2 and σ_3) are assigned such that compressive stresses are negative and $\sigma_1 \leq \sigma_2 \leq \sigma_3$. Principal stresses correspond with a shell depth given by the depth factor ***v***. The principal stresses are sampled based upon the keywords:

1 principal stress σ_1

2 principal stress σ_2

3 principal stress σ_3

shellsel keyword . . .

samples generalized nodal forces acting on a shell. These are the forces that the nodes exert on the shellSEL. The stresses in the shell can be sampled by the **HISTORY sel recover** history item.

node ***nd_id*** keyword ***x y z***

nd_id keyword **cid** = ***cid***

The particular shellSEL is identified either by (***x,y,z***) coordinates (the nearest shellSEL is taken), or by SEL CID number ***cid***. The node at which the quantity will be sampled is designated by the node ID number ***nd_id***. The generalized nodal forces (acting on the shellSEL) are expressed in the global coordinate system. The following force components are sampled based upon the keywords:

Fx nodal force (x-component, global system)

Fy nodal force (y-component, global system)

Fz nodal force (z-component, global system)

Mx nodal moment (x-component, global system)

My nodal moment (y-component, global system)

Mz nodal moment (z-component, global system)

unbalance maximum unbalanced force

zone keyword

Certain zone variables can be sampled. The variables are identified by the following keywords.

pp	pore pressure in zone
smax	maximum principal stress (Compressive stresses are negative, so this is the least negative (minor) principal stress.)
smid	intermediate principal stress
smin	minimum principal stress (Compressive stresses are negative, so this is the most negative (major) principal stress.)
ssi	shear-strain increment
ssr	shear-strain rate
sxx	xx-stress
sxy	xy-stress
sxz	xz-stress
syy	yy-stress
syz	yz-stress
szz	zz-stress
vsi	volumetric-strain increment
vsr	volumetric-strain rate

Histories of Real Time

In addition, special keywords are provided to allow the user to plot histories for transient calculations against real time. These keywords are as follows.

crtime — creates a history of real time for creep problems (only available for creep model option — see Section 2 in **Optional Features**).

dytime — creates a history of real time for dynamic problems (only available for dynamic model option — see Section 3 in **Optional Features**).

fltime — creates a history of real time for fluid-flow problems (see Section 1 in **Fluid-Mechanical Interaction**).

thtime — creates a history of real time for heat-transfer problems (only available for thermal model option — see Section 1 in **Optional Features**).

If histories corresponding to these keywords are not available, the dependent variables will be plotted versus step number, *not* real time.

A history of the timestep for the transient calculations can also be monitored with the following keyword.

dt — timestep. If the calculation involves coupled processes, **dt** will correspond to the process that is active for a given step.

User-Defined *FISH* Variables

Using *FISH*, histories of user-defined variables may be specified (see Section 2 in the ***FISH* volume**).

Manipulating History Data

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

delete — All current histories are erased, and history numbering is reset. (The synonym is **HISTORY reset**.)

dump ***nhis1*** <***nhis2*** . . . ***nhisn***> <keyword . . . >

The history (***timestep number, history value***) of history number ***nhis1*** to ***nhisn*** is written to the screen (maximum of 10 histories per output). The dump 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 number ***nce***.

new clears all histories

skip ***nc***

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

vs ***nhis***

One or more histories will be written in columns adjacent to a column containing another history, ***nhis***, rather than adjacent to a column containing the step number. For example,

```
his dump 1 3 7 vs 2 begin 150 end 375
```

outputs histories 1, 3 and 7, beginning at the recorded history data number 150 and ending at 375. The histories are displayed next to history number 2.

xmaximum sets the maximum for abscissa of the plot

xminimum sets the minimum for abscissa of the plot

ymaximum sets the maximum for ordinate of the plot

yminimum sets the minimum for ordinate of the plot

limits displays the minimum and maximum values of each history

print displays a list of histories currently being recorded (synonymous with the **PRINT hist** command).

purge Contents of all history traces are erased, but the traces themselves remain.

range The maximum and minimum values of output histories are written to the screen.

reset All current histories are erased, and history numbering is reset. (The synonym is **HISTORY delete**.)

write ***nhis1*** <***nhis2*** . . . ***nhisn***> <keyword . . . >

The history (timestep number, history value) of history number ***nhis1*** to ***nhisn*** is written in ASCII form to a file or table (maximum of 10 histories output). The file may be printed or manipulated after stopping *FLAC3D*. This command writes to a file, histories existing at the time the command is issued. Histories are not written to the file during cycling.

The output 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 number ***nce***.

file filename

Histories will be written to filename. If not specified, the history will be written to "FLAC3D.HIS."

skip ***nc***

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

table ***n***

Alternatively, history number ***nhis*** can be written to table number ***n*** with the command **HIS write *nhis* table *n***. Only one history can be written to a table. If the table number already exists, the table contents will be overwritten.

vs ***nhis***

One or more histories will be written in columns adjacent to a column containing another history, ***nhis***, rather than adjacent to a column containing the step number. For example,

```
his write 1 3 vs 2 &
  begin 150 end 375 file test.his
```

outputs histories 1 and 3, beginning at the recorded history data number 150 and ending at 375. The histories are displayed next to history number 2.

IMPGRID filename

IMPGRID imports a *FLAC3D* grid from the named file filename. A path can be part of the filename. If no extension is specified, ".FLAC3D" is assumed.

This command allows importing grids from third-party grid generators such as *3DShop*.

IMPGRID creates a new gridpoint for every gridpoint specified in the input file. Checking for duplicate gridpoints is not done. It is up to you to merge imported grids with existing geometry.

A *FLAC3D* grid file is an ASCII file with the following specifications.

Comment lines start with an asterisk. Blank lines are ignored. Comma separators between numeric values are optional.

A line starting with "G" indicates a gridpoint definition. A gridpoint contains an integer identifier (ID number) followed by the x, y, z coordinates of the gridpoint (floating point numbers).

A line starting with "Z" indicates a zone definition. A zone contains a zone type identifier, an integer zone identifier and the identifiers of the gridpoints comprising this zone.

The ID numbers for the gridpoints and zones must be positive integers. The ID numbers do not have to be sequential or consecutive (gaps are allowed in the numbering); however, every gridpoint ID referred to by a zone must exist in the gridpoint list somewhere. Gridpoint and zone IDs are not used by *FLAC3D* after the file is read. It is purely coincidental if the IDs used in the grid file coincide with the gridpoint and zone IDs assigned by *FLAC3D*.

The following zone type identifiers are recognized:

B8 – brick type zone

W6 – wedge type zone

P5 – pyramid type zone

B7 – degenerate brick type zone

T4 – tetrahedral type zone

For the zone definition, the gridpoint ID numbers must be specified in the standard *FLAC3D* ordering described in Section 1.1.4. The redundant points for wedge, pyramid, degenerate brick and tetrahedron are not specified in the gridpoint definition in the GRD file. For a B8 zone, 8 gridpoints must be specified, for a W6 zone, 6 gridpoints must be specified, etc.

A line starting with “ZGROUP” indicates a group definition for zones. The text immediately following the work ZGROUP is the name of the group. Following the group name, all zones belonging to this group are specified with zone ID numbers.

Example 1.1 shows a portion of a *FLAC3D* grid file.

Example 1.1 A portion of a FLAC3D grid file

```
*
*FLAC3D  input deck produced by KUBRIX version 7.1.0
*mesh built: Wed Nov 19 20:54:43 2003
*GRIDPOINTS
G 1, -6.133336e+000, -6.144340e+000, -3.999275e+001
G 2, -4.732805e+000, -7.033142e+000, -3.999276e+001
G 3, -2.777756e+000, -4.116792e+000, -3.999988e+001
G 4, -3.535428e+000, -3.535424e+000, -3.999989e+001
G 457, -2.745877e+000, -4.129570e+000, -3.201193e+001
                              .
                              .
                              .
G 6, -3.512750e+000, -3.501916e+000, -3.200895e+001
G 17, -8.901190e+000, -8.922812e+000, -3.999268e+001
G 8, -6.845476e+000, -1.015358e+001, -3.999268e+001
*ZONES
Z B8 1, 1, 2, 4, 1443, 3, 6, 1444, 5
Z B8 2, 7, 8, 1, 1445, 2, 1443, 1446, 1444
Z B8 3, 9, 10, 7, 1447, 8, 1445, 1448, 1446
Z B8 4, 11, 12, 9, 1449, 10, 1447, 1450, 1448
                              .
                              .
                              .
Z B8 2785, 13, 14, 11, 1451, 12, 1449, 1452, 1450
Z B8 62, 15, 16, 13, 1453, 14, 1451, 1454, 1452
Z B8 78, 17, 18, 15, 1455, 16, 1453, 1456, 1454
*GROUPS
ZGROUP rock
1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19
20 21 22 23 24 25 26 27 28 29 30 31 32 33 34 35
36 37 38 39 40 41 42 43 44 45 46 47 48 49 50 51
ZGROUP gravel
1601 1602 1603 1604 1605 1606 1607 1608
```

INITIAL keyword <keyword> ***value*** <**grad** ***gx gy gz***> <**ra**nge . . . >

Certain gridpoint or zone variables are assigned initial values over a given 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. One or more keywords can be chosen from the following list.

biot_mod Biot modulus for fluid-flow model
(see Section 1 in **Fluid-Mechanical Interaction**)

damping keyword. . .

Damping parameters are defined for a spatial variation. The following keywords are available.

combined <***value***>

combined local damping (see Section 1 in **Theory and Background**). The damping ***value*** is 0.8, by default.

hysteretic keyword ***values***

Hysteretic damping is invoked with the **hyst** keyword. The following keywords and coefficients are the functions available to represent the variation of G/G_{max} (see Section 1 in **Fluid-Mechanical Interaction**).

default ***value value***

default model represented by cubic equation with two parameters (***c0 c1***)

hardin ***value***

Hardin/Drnevich equation with one parameter (***c0***)

off no hysteretic damping

sig3 ***value value value***

sigmoidal equation with three parameters (***c0 c1 c2***)

sig4 ***value value value value***

sigmoidal equation with four parameters (***c0 c1 c2 c3***)

local <*value*>

local damping (see Section 1 in **Theory and Background**). The damping ***value*** is 0.8, by default.

rayleigh ***frac freq*** <**m**ass **st**iff>

For dynamic calculations, a certain fraction of critical damping is usually required over a given frequency range. This type of damping is known as Rayleigh damping, where ***frac*** = the fraction of critical damping operating at center frequency of ***freq***. (NOTE: Input frequencies for the program are in cycle/sec or Hertz — *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 left out, 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 mesh deformation occurs. In such a case, lower the timestep with the **SET dynamic dt** command.

density mass density of zone

fdensity fluid mass density in zone for fluid-flow model (see Section 1 in **Fluid-Mechanical Interaction**)

fmodulus fluid bulk modulus for fluid-flow model (can only be used for **biot_c** = 1 — see Section 1 in **Fluid-Mechanical Interaction**)

gpextra ***i***

extra grid variable for extra array index ***i***

pp pore pressure at gridpoint

saturation saturation at gridpoint (only for the fluid model option — see Section 1 in **Fluid-Mechanical Interaction**).

state ***0***

The plasticity indicators for tetrahedrons are set to 0 (i.e., no plastic failure)

sxx xx-component of stress

sxy xy-component of stress

sxz	xz-component of stress
syy	yy-component of stress
syz	yz-component of stress
szz	zz-component of stress
temperature	temperature at gridpoint (only for thermal model option — see Section 1 in **Optional Features**)
x	x-coordinate of gridpoint
xdisplacement	x-displacement of gridpoint
xvelocity	x-velocity of gridpoint
y	y-coordinate of gridpoint
ydisplacement	y-displacement of gridpoint
yvelocity	y-velocity of gridpoint
z	z-coordinate of gridpoint
zdisplacement	z-displacement of gridpoint
zextra	***i***
	extra zone variable for extra array index ***i***
zvelocity	z-velocity of gridpoint

NOTES

1. Remember that compressive stresses are negative. Also, these are *total* stresses if either **WATER table** or **INITIAL pp** is issued.

2. Velocity units are length per timestep (real time only if a dynamic analysis is performed).

The following optional keywords may also be given immediately *following* the variable keyword. These keywords apply to gridpoints, but not to zones.

add The keyword **add** adds the specified value (including any variations given) to existing values (of the variable specified) in the grid. This command is useful, for example, in translating coordinates for gridpoints without resorting to using the **GEN** command.

multiply The keyword **mul** multiplies the existing value of the specified variable by the quantity given (including any variations). This command is useful in distorting the grid. Do not use negative multipliers unless you really know what you are doing.

An optional keyword **grad** can be placed after any ***value***. In this case, three values are expected to follow: the gradient ***gx***, in x; the gradient ***gy***, in y; and the gradient ***gz***, in z. The final value installed is

$$value\_modified = value + gx \times x + gy \times y + gz \times z$$

in which x, y, z is the position vector of the gridpoint. If the **mul** keyword is also present, the given gradient is applied to **mul**.

INTERFACE ***i*** keyword <**ra**nge . . . >

Interfaces are planes within a *FLAC3D* model along which sub-grids can interact. Interfaces can be used to model the effect of joints, faults and frictional interfaces between bodies (e.g., cement in a bin, a billet being forced through a die, a concrete foundation on a soil mass). Interfaces can be attached to a sub-grid or they can be located at any position in space; when sub-grids touch the interface, shear and normal forces will develop. Both slip and separation are allowed along the interface. See Section 3 in **Theory and Background** for a full explanation and examples.

Interfaces are composed of triangular elements that are connected at their vertices by nodes. Each interface element has an active side and an inactive side, that are determined by the direction of the normal vector to the interface. Interface nodes can be slaved directly to a grid face or they can be positioned at any location in space. The following keywords are available to create an interface.

ctol ***value***

sets the maximum depth of penetration (of faces that contact an interface) for which contact is detected. If this depth (or overlap) is greater than ***value***, then no contact is detected. By default, ***value*** is determined from zone dimensions, and is updated if deformation occurs during a simulation.

The **ctol** parameter may be used to force contact detection if a large initial overlap exists between two bodies. Use with caution, because large geometrical overlaps may be physically unrealistic. The current values of **ctol** can be displayed with **PRINT int *i* ctol**.

effective **off**
on

The effective stress will be used in the interface calculation if **on**. By default, effective stress is used, **effective** is **on**.

element **p1** <***x y z***> <**no**de ***i***>
p2 <***x y z***> <**no**de ***i***>
p3 <***x y z***> <**no**de ***i***>

This creates a triangular interface element. The three vertices **p1**, **p2** and **p3** must be specified following the **element** keyword. The element can either be created at a location in space corresponding to the ***x***-, ***y***-, ***z***-coordinates of the three vertices, or it can be connected to a previously existing interface node ***i***. The interface element that is created is not attached to a grid face even if the location corresponds to that of the face. The element is fixed in space. The active side of the element is defined by walking around the edge of the element, from **p1** to **p2** to **p3**; the active side is up when walking in a *clockwise* direction.

face Interface elements are created on all grid faces that are within the specified range. An error will occur if interface elements from that interface already exist on the selected faces.

maxedge ***value***

value is the maximum edge length for all interface elements in the interface. The interface is subdivided until all of its elements have an edge length smaller than the assigned value. By default, two triangular elements are created for each quadrilateral zone face.

node <**id** ***i***> ***x y z***

This creates an interface node at position (***x, y, z***). An ID number can be assigned with the optional keyword **id**, otherwise the ID is determined automatically. If a node already exists at the selected location, an error is reported. The created node is fixed in space.

nstress ***x***

sets normal stress increment to ***x*** for all interface number ***i*** nodes in optional range (compression positive). Normal stress increment is added to the normal stress calculated from interface penetration.

permeability **off**
on

Interfaces are permeable by default. An interface can be made impermeable by setting permeability **off**. If **maxedge** is used, the interface becomes impermeable and cannot be made permeable.

smalldisp ***value***

Interfaces operating in small-strain mode derive their forces from a comparison of "virtual positions" of the two interacting faces, where a "virtual position" of a point is the original coordinate of the point plus the accumulated displacement (**smalldisp**) to date. If one contacting object is removed (e.g., backfill within a tunnel), and another substituted (e.g., a liner is installed), using the same interface, then large initial stresses may appear because the two sides of the interface appear to interpenetrate (because the virtual position of the old object has changed with respect to the specified position of the new object).

The stored displacements associated with a particular interface ***n*** may be multiplied by a factor ***value***, using the keyword **smalldisp**. Normally, this factor will be zero, thus restoring the virtual positions of the faces comprising the interface "host" to their original locations. In this case, a new interacting object will not cause initial stresses to be generated when its coordinates match the original coordinates

of the interface host faces. However, it is possible to give ***value*** a nonzero value, in order to simulate an initial lack-of-fit between the two contacting objects.

The command changes the virtual positions in the interface, and in the two faces that are in contact. However, for a newly created interface, the "target" face is not found until cycling has begun. Therefore, for all positions to be reset, at least one cycle should be executed before giving the command, if the target (i.e., the face without the interface elements) has some existing deformation.

sstress ***x y z***

sets the incremental shear stress (vector components) ***x, y, z*** for all interface number, ***i***, nodes in optional range.

update **off**
on

This command prevents the search for new contacts after movement occurs on an interface. The same contacts are preserved, whatever the magnitude of displacement. Use with caution, because physically unrealistic behavior can result if displacements are large. The default behavior is **update on**, which allows normal searching for new and broken contacts.

wrap ***gname1 gname2***

Interface elements are created on all zone faces belonging to group ***gname1*** where a twin face exists on a zone in group ***gname2***. A face is considered to be a twin if the faces occupy the same position in space, and each face has its own unique set of gridpoints (i.e., each face in the pair has a different set of gridpoint ID numbers). To separate one group from another, see the **GENERATE separate** command.

For example, the command below would find the twinned faces between group "rock" and group "soil," and put interface elements on these "rock" faces. Only faces with centroid within the range *x* 50.0 75.0 would be considered.

```
interface 1 wrap rock soil range x 50.0 75.0
```

Interfaces can be deleted with the following keyword.

delete — All interface elements associated with this interface within the selected range are deleted.

The interface is characterized by Coulomb sliding and/or tensile separation. The interface properties are assigned with the **property** keyword, followed by the interface properties given below. Normal and shear stiffnesses, **kn** and **ks**, must be specified for all cases. Any set of consistent units may be used for these properties. If not specified, the properties are zero.

property keyword ***value*** . . .

The property keywords available are:

cohesion — cohesion [stress]

dilation — dilation angle [degrees]

friction — friction angle [degrees]

kn — normal stiffness [stress/displacement]

ks — shear stiffness [stress/displacement]

tension — tensile strength [stress]

If a bonded interface is specified (**tension** is set), then the following optional keywords can be specified.

bslip **off**
on

Slip is allowed, or not allowed, for a bonded interface segment. (Default is **off** — i.e., slip is not allowed if bond is intact.)

sbratio ***sbr***

The shear bond strength is set to ***sbr*** times the normal bond strength (**tension**). Note that giving **sbratio** alone does not cause a bond to be established; the tensile bond strength must also be set. The default value for **sbratio** is 100 (i.e., shear bond is 100 times tensile bond).

Further descriptions of the **INTERFACE** command and example applications can be found in Section 3 in **Theory and Background**.

MACRO string1 string2

The command **MACRO** enables macro substitution of token string1 into string2 whenever it is entered in a command line. For an extended substitution containing delimiters (such as spaces), string2 must be surrounded by single quotes. To override an existing macro, surround string1 with single quotes in the second declaration. Nested macro calls are allowed to a current limit of 10 levels. Recursive calls will not crash *FLAC3D*, but they will not execute correctly. To remove a macro, name it in string1 (surrounded by single quotes) and do not specify string2.

In **safe** mode (see **SET safe on**), any macro *must* be preceded by the # symbol. This unambiguously identifies the name as a macro. The # symbol is ignored if **safe** is off.

Macros are cleared when the **NEW** command is issued.

MAINWIN keyword

The **MAINWIN** command positions the main window in the Graphical User Interface (GUI). The following keywords are available.

position *x y*

The upper-left corner of the main window is located at (***x,y***).

size *x y*

The length of the main window is ***x***, and the height is ***y***.

The values ***x*** and ***y*** are fractions of the desktop size. The upper-left corner of the screen is at ***x*** = 0, ***y*** = 0 and the lower-right corner is at ***x*** = 1, ***y*** = 1. For example, ***x*** = 0.5 means 50% of the horizontal length of the desktop. See "TRENCH1.DAT" in Section 2.12 in the **User's Guide** for an example application of this command.

MODEL keyword <**ov**erlay ***n***> <**ra**nge . . . >
or
load filename

This command associates a constitutive model with specified zones in the grid. If a **range** is given (see Section 1.1.3), then the model is associated with only those zones within the range. If no range is specified, then the model is associated with the entire grid.

During the calculation, zones will behave according to a constitutive model corresponding to one of the keywords given below. (The mechanical constitutive models are described in Section 2 in **Theory and Background**, the creep models in Section 2 in **Optional Features**, the thermal models in Section 1 in **Optional Features**, and the fluid flow models in Section 1 in **Fluid-Mechanical Interaction**.)

The optional keyword **overlay** assigns either two overlaid sets of five constant-strain tetrahedral elements to each zone (***n*** = 2), or one set of five tetrahedral elements to each zone (***n*** = 1). The default is set to double-overlaid elements (***n*** = 2).

The keyword **load** loads a DLL version of a constitutive model, identified by filename. Once loaded, the DLL model can be assigned to zones in the grid using the **MODEL** command and **range** keywords. See Section 4 in **Optional Features** for a guide to creating user-defined models as DLLs. **MODEL load** can only be applied if **CONFIG cppudm** is invoked.

Mechanical Models

anisotropic	transversely isotropic elastic model
cam-clay	modified Cam-clay plasticity model
doubleyield	double-yield (cap) plasticity model
drucker	Drucker-Prager plasticity model
elastic	isotropic elastic model
finn	dynamic pore-pressure generation model (available only for the dynamic option — see Section 3 in **Optional Features**)
hoekbrown	generalized Hoek-Brown model
mohr	Mohr-Coulomb plasticity model
null	null model
orthotropic	orthotropic elastic model
ssoftening	strain-hardening/softening plasticity model
subiquitous	bilinear strain-hardening/softening ubiquitous-joint plasticity model

ubiquitous ubiquitous-joint model

Creep Models (available only for creep model option — see Section 2 in **Optional Features**)

burger	Burger's substance viscoelastic model
cpower	power-law viscoplastic model
cvisc	Burger-creep viscoplastic model
cwipp	crushed-salt constitutive model
power	two-component power law
pwipp	WIPP-creep viscoplastic model
viscous	classical viscoelastic model
wipp	WIPP reference creep formulation

Fluid-Flow Models

fl_anisotropic anisotropic fluid flow

fl_isotropic isotropic fluid flow

fl_null null flow model. Null zones model impermeable material. Note that, for the coupled fluid-flow mechanical process, if zones are nulled mechanically, they must also be nulled for fluid flow (default).

Thermal Models (available only for thermal model option — see Section 1 in **Optional Features**)

th_ac	isotropic advection-conduction
th_anisotropic	anisotropic thermal conductivity
th_isotropic	isotropic heat conduction
th_null	null thermal model. Null zones model excavated material and insulators. Note that zones made null mechanically are *not* automatically made null thermally.

MOVIE keyword

This command controls the capture of screen plots and permits rapid replay as a "movie." (Also see the **SET movie** command. Two types of movie files can be created: DCX and AVI. Examples of creating movies can be found in Examples 1.2 and 1.3. Refer to the two examples below.

A DCX file contains a set of PCX images that are strung together. The DCX format has a limit of 1024 frames. DCX movies may be viewed using the movie utility supplied with the Itasca software products ("movie.exe" is located in "c:\Program Files\ITASCA\SHARED\Utility"). This movie utility may be freely distributed with Itasca movie files.

AVI (Audio/Video Interleave) is a Windows movie format. These files can be played with the standard Windows media player.

In the movie files, images are generated as a series of screen captures (bitmaps). The current plot is the only plot that can be sent to the movie file. The current plot can be set through the **PLOT current viewid** command.

Available keywords are:

start — opens the movie file.

finish — closes the movie file.

snap — adds a frame (screen image) to the currently open movie file. The image is the same as that appearing in the current plot (see the **PLOT current viewid** command).

Example 1.2 Example DCX movie creation

```
; dcx.dat
; Create a movie file called test1.dcx.
def rot_plot
loop ii(1,90)
xr = ii*10
yr = ii*2
command
movie snap ; dump a frame to the dcx file
plot set rotation xr yr 0
end_command
endloop
end

gen zone brick size 5 10 15
plot sur yellow
```

```
set movie dcx size 400 300 file test1
movie start
rot_plot
movie finish
```

Example 1.3 Example AVI movie creation

```
; avi.dat
; Create an AVI movie file (steptest.avi) using the step option.
; Snap a picture every 30 steps while solving.

gen zone radcylinder size 25 1 25 25
gen zone reflect normal -1 0 0
gen zone reflect normal  0 0 -1
gen merge 1e-5
model mohr
prop dens=1000 bu=1e8 sh=7e7 fric 25 coh 3.5e4 tens 1e10
fix y
fix x range x -24.9 -25.1
fix x range x 24.9 25.1
fix z range z -24.9 -25.1
set grav 10
ini szz -1e6   grad 0 0 1e6
ini sxx -0.5e6 grad 0 0.5e6 0
ini syy -0.5e6 grad 0 0.5e6 0

plot create testview
plot current testview
plot add contour szz average outline on

set movie avi step 30 file steptest.avi
movie start
solve ratio 1e-2
movie finish
```

NEW This command allows the user to begin a new problem without leaving *FLAC3D*. The grid becomes undefined, and problem variables are reset to zero or to their default values. The file "FLAC3D.INI" is consulted again for any start-up commands.

The echo mode (**SET echo**) and log file (see **SET log**) are unaffected by **NEW**. These can be turned on or off as needed. The plot filename is reset to "FLAC3D.PS" after the **NEW** command is given.

The random number generator seed, used with **PROPERTY** . . . **gauss_dev**, for example, is not reset when **NEW** is given.

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

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

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

This command allows the user to pause reading a **CALL**ed file. When **PAUSE** is encountered, *FLAC3D* will stop processing the data at that point and pass control back to the keyboard. Any commands can then be typed (e.g., **PLOT grid**). When the **CONTINUE** command is typed, *FLAC3D* will resume reading the data file.

Two options are available:

key — *FLAC3D* 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.

t — *FLAC3D* will pause ***t*** seconds and then resume processing.

The <SPACEBAR> will cause *FLAC3D* to immediately continue processing the next data line after the **PAUSE key** command, whereas the <Esc> key will abort all processing and abort reading the data file.

PDELETE keyword. . . **ra**nge . . .

All particles in zones with centroids within the **range** defined by the range phrase are deleted. See related command **TRACK**.

The following keyword may be used:

inactive

specifies that only particles that have exited the grid will be deleted.

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

This command requests that a plot be made on the screen* or directed to a hardcopy plotting device or file.

The plotting logic is built around the concept of a "view." A particular view includes the view-setting parameters (background, size, etc.), as well as the actual items being plotted (model surfaces, vectors, etc.). The plotting logic initializes the view list with a single default view, with view identification (viewid) number 0 and name "Base."

More than one view may be created and stored, and the user can switch between stored views to define the "active" view. The viewid can be either an integer, indicating the identification number of the view, or a string, indicating the name of the view.† Views are created with the **create** keyword and are made active with the **current** keyword. A list of views can be obtained with the **print** keyword.

Every view stores a number of "plot items": these are the particular graphical items the view displays (for instance, a plot of the model and velocity vectors). Plot items are added to a view with the **add** keyword, removed with the **subtract** keyword, modified with the **modify** keyword, and reordered in a view with the **move** keyword. A plot combining all plot items is displayed on the screen when a **show** keyword is given. A list of plot items assigned to a specific view is obtained with the **print item** keyword phrase.

Plotting manipulation is grouped into four categories:

(1) view manipulation — keywords define the view and output conditions;

(2) view-setting manipulation — keywords describe the background and foreground color settings, view position settings, and plot caption and title settings;

(3) plot-item manipulation — keywords are used to build (e.g., add, subtract and modify) plot items within a view; and

(4) interactive manipulation — in graphics screen mode, certain keystrokes allow user-interaction with *FLAC3D*.

The four categories are described below. Table 1.4 summarizes the keywords associated with the first three categories. Table 1.5 lists the keystrokes that are active in the fourth category.

---

* When running *FLAC3D* in interactive mode, if the **PLOT** command is entered alone at the `Flac3D>` prompt, then the `Plot>` prompt will appear. Any subsequent input then assumes that the **PLOT** command was given first.

† If a name is chosen for a viewid that can be evaluated as an integer (e.g., the name of a *FISH* function or variable), the code will attempt to operate on a view with that value. To use a name that is already used as a *FISH* function, enclose the name in single quotes.

Plot items define the graphical representations of the components of the model (e.g., model surfaces, vectors, histories). The plot items available in *FLAC3D* are summarized in Table 1.6 and are described separately following the plotting manipulation categories.

Switches are also provided to modify or enhance the plot item. The switches include color switches which can be used to change the colors in a plot. The color switches are listed in Table 1.7. The switches are optional, but they must follow immediately after the plot-item keyword on the same command line. Applicable switch keywords are listed with each plot item.

It is possible to combine and view plot items directly without first creating a view. The default view, Base, is used as a scratch plot view. Some view manipulation keywords and plot-item manipulation keywords (see Table 1.4) may be ignored, but the plot that is created will be overwritten when a new plot is created. For example, in order to plot the grid, velocity vectors in red and the axes in black, simply type

```
plot grid velocity red axes black
```

and the view Base, containing these three plot items, will appear. The contents of this view are overwritten whenever a new plot item is sent to it. It is possible to copy this view to a permanent view by giving the **PLOT copy** command immediately after exiting the graphics mode.

Table 1.4 Summary of* PLOT *manipulation keywords

View manipulation	View-setting manipulation	Plot-item manipulation
clipboard	**reset**	**add**
close	**set** keyword . . .	**clear**
copy	**angle**	**modify**
create	**background**	**move**
current	**caption**	**print item**
destroy	**center**	**subtract**
export	color	
extract	**dd**	
hardcopy	**dip**	
print	**direction**	
quit	**distance**	
rename	**eyedistance**	
show	**foreground**	
	light	
	magnification	
	mode	
	moveincrement	
	normal	
	origin	
	perspective	
	plane	
	position	
	rotation	
	rotincrement	
	size	
	title	
	vertical	
	wait	
	window	
	zangle	

1. View Manipulation

The keywords and switches listed in Table 1.4 are defined below.

clipboard <viewid> or <name>

The plot item identified by viewid or name is sent to the clipboard as an enhanced metafile. If no viewid or name is identified, the current plot is sent to the clipboard.

close <viewid>

The current (or the specified) plotting view is closed.

copy viewid1 viewid2 <**se**ttings > <**it**ems > <**both**>

The view identified by viewid1 is copied to viewid2. If viewid2 does not exist, one is created. The optional keywords, **settings**, **items** and **both**, determine whether the view settings, the plot item list, or both, are copied. The default is **both**. If a new view is created, it is made the active view.

create viewid

A new view named viewid is created, added to the view list, and made the active view.

current viewid

The view named viewid is made the current (i.e., active) view.

destroy viewid

The view named viewid is erased. Neither the active view nor the "Base" view may be destroyed.

export <viewid> <**fi**le filename>

This command displays the *FLAC3D* commands needed to create the current or specifically identified viewid view settings to the screen. If filename is specified, then the commands are listed to that file and can be read in again via the **CALL** command.

extract viewid xarr yarr zarr datarr

This command extracts data from the view named viewid into a *FISH* data array, datarr. The extraction points are defined by x, y, z coordinates that are specified in *FISH* arrays xarr, yarr, zarr. The four arrays must be defined and initialized with x, y, z data before the **extract** keyword is issued. The arrays can be any valid *FISH* array. All arrays must have the same dimensions. When the **extract** keyword executes, it searches the view for a contour-type plot item (e.g., **plot cont pp**, **plot cont szz**, etc.), interpolates the contour data to the extraction point coordinates, and places the value into the *FISH* data array. The data is interpolated from the zone gridpoint values using a volume weighting function using the tetrahedral overlays. The extraction occurs with every contour-type plot item in the view and the extracted data will reflect the last item plotted (all other extracted data will be overwritten). If data can't be obtained at an extraction point (e.g., the point falls outside the model) then the data array will contain a value of zero at this point.

hardcopy <viewid> <**fi**le filename>

The view identified by <viewid>, or the current view if none is specified, is routed to the current hardcopy device (see **SET plot**). If the device is one that produces a file, then the name will be <EXT>, by default, where <EXT> is determined by the current hardcopy device. Alternatively, a filename can be specified directly by using the **fi**le keyword. The plot window should be full-screen when creating a hardcopy plot, to most closely resemble what will appear on the hardcopy output, as the hardcopy aspect ratio may not match the plot window's dimensions.

print <**vi**ew <viewid>>
<**list**>
<**in**formation>
<**it**em <i>>

The keyword **print** defaults on its own to the behavior of **print list**. This prints a list of all plot views and their identifier names. If the keyword **view** is used, a detailed list of all settings for that view are printed. A specific view can be selected by adding a viewid. The keyword **information** prints the current display and hardcopy device settings. The **item** keyword will print a list of plot items currently in the view. If a number is specified, a detailed printout of the plot-item settings will be made.

quit quits the plotting mode and returns to command mode.

rename viewid1 viewid2

The view identified as viewid1 is renamed to viewid2. The base view cannot be renamed. Integers and strings must be unique.

show <viewid>

All plot items in the current plot view are displayed on the screen. If viewid is specified, that view is displayed.

2. View-setting Manipulation

The following keywords assign view settings. All keywords apply to the *current* view.

reset All view settings are reset to their default values.

set keyword

The available setting keywords are:

angle ***a***

The view **angle** controls perspective distortion — a higher value for ***a*** produces more distortion. Adjusting the magnification effectively changes the view angle.

back color

The screen background can be set to color. Note that the default screen foreground color may be set with the **foreground** keyword. See Table 1.7 for color keywords.

caption keyword

This sets and positions the caption legend. The following keywords apply.

left puts the caption legend to the left of the plot (default).

off turns off the caption legend. Setting the legend **off** allows the entire screen to be used for plotting.

on turns on the caption legend (default).

right puts the caption legend to the right of the plot.

size ***n***

integer value ***n*** can be set from 10 to 50, indicating the percent size of the caption legend relative to the plot window (default ***n*** = 35).

color **off**
on

This sets the color mode **off** or **on**. If **off**, the viewport will not use color, even if available. This provides a closer representation of a hardcopy grayscale plot (such as PostScript). The <CTRL-G> key can be used to toggle between **on** and **off** when in graphics mode. (The default is **on**.)

eyedistance ***d***
auto

The distance is specified from the plane of the screen to the viewer's "eye" for perspective calculations. If **auto** is specified, the distance is calculated automatically.

foreground color

The screen foreground can be set to color. See Table 1.7 for color keywords.

light ***xv yv zv***

The direction of light for surface-shading calculations is specified. The default is the unit vector of -1, 3, -2.

magnification ***m***

The view is magnified by the factor ***m***.

mode keyword

The **mode** toggles give the user flexible ways to view a model. For instance, to "fly through" a tunnel, the **first-person** mode is most useful, whereas the **model** mode is more convenient for simply observing the model from a stationary point in space.

Modes can be changed interactively in a plot window — refer to Table 1.5 for the interactive keystrokes. Note that the **mode** toggles take no parameters themselves — appropriate values have to be specified beforehand or defaults are used.

Essentially, a viewing system is dependent on two entities, the *viewer* and the viewed object (or *object*). The view describes the relation between these two, and controls which one of them "moves" and which is "stationary." Once all the parameters are established, *FLAC3D*

can translate between the various modes and display the results on the viewport.

The different modes are described below. For convenience, the settings keywords associated with each mode are listed with that mode. Note that the settings are keywords to the **PLOT set** command, *not* to the mode toggles themselves.

Mode 1 — **firstperson**

firstperson In this mode, the *object* is stationary and the *viewer* moves. For example, a rotation about the *z*-axis means the *viewer* is turning away from the *object*, while the *object* remains undisturbed — the *viewer* loses sight of the *object*. The advantage of this mode is that the *viewer* can move through the stationary *object*.

This mode uses data entered via the **position**, **direction** and **vertical** keywords to compute the view.

direction ***x y z***

A vector from the *origin* (not the position) to this coordinate describes the viewing direction.

position ***x y z***

The eye position is placed at this coordinate.

vertical ***x y z***

A vector from the *origin* (not the position) to this coordinate describes the "up" direction.

Mode 2 — **model**

model In this (the default) mode, the *viewer* is stationary and the *object* moves. For example, a rotation about the z-axis means the *object* is spun about its z-axis, while the *viewer* remains undisturbed — the *object* appears to spin in front of the *viewer*.

This mode uses data entered via the **center**, **distance** and **rotation** keywords to compute the view.

center ***x y z***
auto

The center of the model view is specified by the coordinates (***x, y, z***). If **auto** is specified, the center is calculated as the geometric center of the model.

distance ***d***
auto

The distance is specified from the plane of the screen to the viewer's "eye" for perspective calculations. If **auto** is specified, the distance is calculated automatically.

rotation ***xr yr zr***

This sets the rotation of the model for viewing; ***xr***, ***yr***, ***zr*** are the rotation angles about the x-, y- and z-axes, respectively. All angles are input in degrees.

Mode 3 — **plane**

plane This mode behaves in exactly the same way as the **model** mode, allowing the user to describe the view in terms of a plane rather than a point. The origin of the plane can be specified as a point outside the actual model and, as in **model** mode, rotations can cause the model to "disappear" from the view. By default, the plane origin is at the model center.

This mode uses data entered via the **origin**, **dip**, **dd**, **normal**, **distance** and **zangle** keywords to compute the view.

dd ***value***

This specifies the dip angle (degrees) of the plane measured in the global xy-plane clockwise from the y-axis (default is 0).

dip ***value***

This specifies the dip angle (degrees) of the plane measured downward from the global xy-plane (default is 0).

distance ***d***
auto

The distance is specified from the plane of the screen to the viewer's "eye" for perspective calculations. If **auto** is specified, the distance is calculated automatically.

normal ***xv yv zv***

This specifies a normal vector to the plane, with the components ***xv***, ***yv*** and ***zv***.

origin ***xv yv zv***
auto

This specifies the location of one point on the plane. The coordinates of the point are (***xv, yv, zv***).

zangle ***degrees***

angle of the local vertical in degrees clockwise from the zenith angle of the current plane description

moveincr ***m***

The amount, in model dimensions, **center**, **origin** or **position** is moved when an interactive key-press causes a translation.

perspective **off**
on

This sets the perspective mode **on** or **off**, and determines whether perspective distortion will be introduced to the output (**on**) or the output will be a simple projection onto the viewing plane (**off**). The default value is **on**.

plane keyword ***value*** <keyword ***value***> . . .

A cutting plane through the model is defined. The following keywords specify the location of the plane.

dd ***value***

This specifies the dip direction, ***value***, of the plane measured in the global xy-plane clockwise from the y-axis (default ***value*** = 0).

dip ***value***

This specifies the dip angle, ***value***, of the plane measured downward from the global xy-plane (default ***value*** = 0).

moveincr A cutting plane can be moved through a contour plot by pressing either the <PAGE UP> or <PAGE DOWN> key. The amount the plane translates along its normal is determined from the value set for the **moveincr** parameter, which is also accessible in the cutting-plane dialog box (EDIT ---> CUTTING PLANE menu item).

normal ***xv yv zv***

This specifies a normal vector to the plane, with the components ***xv***, ***yv*** and ***zv***. If **normal** is specified, **dip** and **dd** are not required.

origin ***xv yv zv***

This specifies the location of one point on the plane. The coordinates of the point are (***xv, yv, zv***).

rotincrement ***r***

the number of degrees the view is rotated when an interactive key-press causes a rotation

size value
auto

The size entered represents the extents of the viewing plane. The viewing plane extents are set to (xlow, xhigh, ylow, yhigh) = (-value, value, -value, value) when value is specified. The extents of the actual viewing area will also depend on the aspect ratio of the output device. If **auto** is specified, the size will be calculated based on the model extents. This command only applies when perspective is turned off.

title keyword

The title window for the plot title is activated or deactivated. The following keywords are available.

bottom positions the title window at the bottom of the plot view.

off deactivates the title window.

on activates the title window (default is **on** if **title** is specified).

text <string>

A view title is specified with **text**. If a parameter follows **text**, it is parsed as a string and assigned to the view title. Single quotes may be used to embed blank characters in the string. If no parameter follows **text**, a new prompt will be displayed when <ENTER> is pressed, and a new view title can then be given. If no title is entered, the existing view title will remain. The existing title will appear if the <F3> key is pressed.

top positions the title window at the top of the plot view (default setting).

wait ***t***

This command is for use on computers whose keyboard interrupts take a long time to process (most notably, some portables). **wait** sets the minimum time, ***t*** (in seconds), that the plotting logic will take before checking the keyboard. If plotting operations seem unreasonably slow, try setting this value to 0.25 or 0.5. The default value is 0. This value is the same for all viewports and is not stored in a save file.

window keyword

This command positions and sizes the current plot viewport in the GUI. The following keywords apply.

position ***x y***

The upper-left corner of the viewport is located at (***x, y***).

size ***x y***

The length of the viewport is ***x*** and the height is ***y***.

The values ***x*** and ***y*** are fractions of the desktop size. The upper-left corner of the screen is at ***x*** = 0, ***y*** = 0 and the lower-right corner is at ***x*** = 1, ***y*** = 1. For example, ***x*** = 0.5 assigns the plot 50% of the horizontal length of the desktop. See Example 2.25 in the **User's Guide** for an example application of this command.

3. Plot-item Manipulation

add item

A plot item, identified by the plot-item keyword item (see Table 1.6) is added to the current view.

clear All plot items are removed from the current view.

modify ***i*** switch <***value***> . . .

The plot item, identified by the plot-item number ***i***, is modified or changed by assigning new keyword switches.

move ***i1 i2***

Plot-item number ***i1*** is moved before ***i2*** in the sequence of plotting for the current view. To move an item to the end of the list, specify a number greater than any in the list for ***i2***.

print **it**em <***i***>

The list of plot items in the current view is printed. If ***i*** is specified, then detailed information about plot item ***i*** is printed.

subtract ***i***

The plot-item number ***i*** is removed from the active view, and the view items are renumbered.

4. Interactive-Plotting Manipulation

When the graphics plot of a *FLAC3D* model is displayed, certain keystrokes may be used to manipulate the view. (It is *not* necessary to follow the keystroke with the <ENTER> key.) Available keystrokes are summarized in Table 1.5.

Table 1.5 Interactive keystrokes in plot mode

Keystroke	Function
<\>	toggle view mode
<+>	increases increment (× 1.25)
<−>	decreases increment (× 0.8)
<←>	moves left
<→>	moves right
<↑>	moves up
<↓>	moves down
<CTRL-C>	brings up the camera dialog
<CTRL-G>	toggles between color and grayscale plotting
<CTRL-L>	cutting plane
<CTRL-R>	reset view to default
<CTRL-Z>	zoom rectangle
<DELETE>	moves the eye distance toward the model
<END>	moves toward (out-of-plane)
<ENTER>	returns to command mode
<HOME>	moves away (out-of-plane)
<INSERT>	moves the eye distance away from the model
<M>	decreases viewing width (magnify)
<SHIFT-M>	increases viewing width (unmagnify)
<PG DN>	moves viewing plane in the opposite direction of the outward normal to the viewing plane
<PG UP>	moves viewing plane in the direction of the outward normal to the viewing plane

Table 1.5 Interactive keystrokes in plot mode (continued)

Keystroke	Function
<X>	rotates about *x*-axis (front-up)
<SHIFT-X>	rotates about *x*-axis (front-down)
<Y>	rotates about *y*-axis (clockwise)
<SHIFT-Y>	rotates about *y*-axis (counterclockwise)
<Z>	rotates about *z*-axis (front-right)
<SHIFT-Z>	rotates about *z*-axis (front-left)
<F3>	sends output to current hardcopy device (see **SET plot**) and current output file (see **SET out**)
<F4>	executes **SOLVE** command*
<F5>	executes calculation cycles*
<F9>	redraw
<CTRL-F1 TO CTRL-F10>	modify item

* Plot is redrawn every ***n*** cycles; ***n*** is set with the **SET pinterval** command for the mechanical stepping interval (default is ***n*** = 1). Press any key to stop calculation.

Plot-item Keywords

All plot items are discussed in the following pages. Table 1.6 lists all the available plot items.

Table 1.6 List of available plot items

attach	**axes**	**bcontour**	**block**
boundary	**contour**	**displacement**	**fap**
flow	**fob**	**fos**	**gpfix**
grid	**history**	**interface**	**location**
sel	**sketch**	**stensor**	**surface**
table	**track**	**velocity**	**volume**
water			

If any switches apply, descriptions of these will be located with the descriptions of their associated plot items.

Switches are used to change the color of plot items on the screen and the fill shade of b/w hardcopy plots. Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots. Color index numbers are assigned to each color; the numbers are integers starting from zero. Color switches must be the first switches given for a plot item.

Table 1.7 Color switch keywords

Keyword	Screen Color	Greyscale	Line Style
black	black		
blue	blue		
green	green		
cyan	cyan		
red	red		
magenta	magenta		
orange	orange		
brown	brown		
dgray	gray		
lgray	light gray		
lblue	light blue		
lgreen	light green		
lcyan	light cyan		
lred	light red		
lmagenta	light magenta		
lorange	light orange		
yellow	yellow		
white	white		

NOTES

1. Colors can also be specified by number to obtain a specific color. It is recommended that a **MACRO** be used to facilitate the use of a favorite color. Three keywords are available.

   gray ***v***

   v is a real value between 0.0 and 1.0; ***v*** = 0.0 represents black.

   hsb ***v1 v2 v3***

   v1, ***v2*** and ***v3*** are real numbers in the range 0.0 to 1.0. **hsb** allows a general color to be specific, much like RGB, but in the Hue, Saturation, Brightness color space. ***v1*** represents Hue, ***v2*** represents Saturation and ***v3*** represents Brightness.

   rgb ***v1 v2 v3***

   v1, ***v2*** and ***v3*** are real numbers in the range 0.0 to 1.0. ***v1*** represents Red, ***v2*** represents Green and ***v3*** represents Blue.

2. The keyword **none** can be used to terminate the list of defined colors. This can be useful to reduce the number of colors defined. The number of colors that a plot item considers to be assigned can be specified directly with the keyword **numcolor *i***. If ***i*** is specified as higher than the current assigned colors, the new color indices will be set to black. Once they have been specified, the only other way to reduce the number of assigned colors is to recreate the plot item.

attach <switches>

Attached grid faces are plotted. A mark is plotted on the attached gridpoint in color 1, and an outline of the object to which it is attached is plotted in color 2.

If an attached point is not in the same physical location as the item to which it is attached, a line is drawn between them indicating the connection.

The following switches apply.

color — Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots.

activate — **off**
on (default)

If **off**, the item is not plotted, but is still assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias — name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

behind — plots the plot item behind the current viewing plane (see **PLOT set plane**).

front — plots the plot item in front of the current viewing plane (see **PLOT set plane**).

plane — plots the plot item on the current plotting plane (see **PLOT set plane**).

axes <switches>

This keyword adds an axes display to the current view. The following optional switches apply.

color Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots. Only 1 color may be specified.

activate **off**
on (default)

If **off**, the item is not plotted, but is still assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

fix ***x, y***

If the optional keyword **fix** is given, the axes will be fixed in position on the screen at coordinates ***(x,y)***, where ***x*** and ***y*** are fractions of the screen width. ***(0,0)*** is the lower-left position and ***(1,1)*** is the upper-right position.

position ***x y z***

This keyword moves the location of the axes display to global position (***x, y, z***). The default is for the axes to be drawn at the origin. Note: Only the plotting position of the axes is changed — the global position of the model origin is unaffected.

scale ***v***

scales the object as a fraction ***v*** of the screen size. (default ***v*** = 0.1)

bcontour keyword <switches>

A block contour plot of the specified zone variable is added to the current view. Available switches are listed after the keywords. The following keywords apply.

density zone mass density

ppressure average zone pore pressure

property name

The property identified as name is plotted. See the **PROPERTY** command for all of the properties.

smax maximum principal stress (σ_3). Note that because compressive stresses are negative, this is the least negative (minor) principal stress.

smid intermediate principal stress (σ_2)

smin minimum principal stress (σ_1). Note that because compressive stresses are negative, this is the most negative (major) principal stress.

ssi shear-strain increment

ssr shear-strain rate

sxx xx-stress

sxy xy-stress

sxz xz-stress

syy yy-stress

syz yz-stress

szz zz-stress

vsi volumetric shear-strain increment

vsr volumetric shear-strain rate

zextra ***i***

the ***i***th extra zone variable

The following switches apply.

color — Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots. 24 colors are available; new colors are selected and assigned automatically if needed.

activate
off
on (default)

If **off**, the item is not plotted, but is still assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

behind — plots the plot item behind the current viewing plane (see **PLOT set plane**).

effective
off (default)
on

modifies zone stress contour plot to plot effective stress.

fast
off
on (default)

toggles the fast plotting mode. In fast plotting, the face-sorting is speeded up at the risk that some faces may be plotted out of sequence.

front — plots the plot item in front of the current viewing plane (see **PLOT set plane**).

gpnum
off (default)
on

gpnum on plots gridpoint numbers at surface face vertices or interface node IDs.

gppos
off (default)
on

plots the position of gridpoints.

id **off** (default)
on

identification numbers of objects. Numbers are only displayed if their corresponding entities are also displayed.

interval ***v***
auto (default)

sets the contour interval for contour plots. If **auto** is specified instead of ***v***, the interval is calculated automatically for a total of between eight and sixteen contours.

maximum ***v***
auto (default)

sets the maximum value for contour plots to ***v***. If the keyword **auto** is specified, the maximum value for the model is selected for ***v***.

minimum ***v***
auto (default)

sets the minimum value for contour plots to ***v***. If the keyword **auto** is specified, the minimum value for the model is selected for ***v***.

null **off** (default)
on

If **null** is **on**, only null zones for the plot item are plotted.

outline **off**
on (default)
color (default is black)

If **outline** is **on**, an outline of the ball surface will be drawn in the outline color, which can be changed by specifying a color after the **outline** keyword. **outline** is **off**, by default.

plane plots the plot item on the current plotting plane (see **PLOT set plane**).

range keyword . . .

limits the plotting to a specified range. The available keywords are described in Section 1.1.3.

reverse	**off** (default) **on** reverses the color scale for contour plots.
scale	***v*** scales the object as a fraction ***v*** of the screen size. In this case, the text is scaled. (default ***v*** = 0.04)
shade	**off** **on** Color shading, with respect to the light vector, is turned off or on (default is shading **on**). See **PLOT set light**.
total	modifies zone stress contour plot to plot total stress (default).

block keyword <switches>

The surfaces of zones are filled with a different color which corresponds to a different value of specific model variables. Available switches are listed after the keywords. The following keywords apply.

density zone mass density

group groups in the model (see the **GROUP** command)

model mechanical material models (see the **MODEL** command)

property name

material property of the zones; refer to the **PROPERTY** command for valid property names.

state <keyword>

plasticity state of the zones. The following states will be identified by different colors.

shear-n at shear failure now

shear-p elastic, but previously at shear failure

tension-n at tensile failure now

tension-p elastic, but previously at tensile failure

u:shear-n at ubiquitous-joint shear failure now

u:shear-p elastic, but previously at ubiquitous-joint shear failure

u:tension-n at ubiquitous-joint tensile failure now

u:tension-p elastic, but previously at ubiquitous-joint tensile failure

Each different combination of these states will have its own color.

The plasticity states can be plotted separately by specifying an optional state keyword. For example, to plot only those zones that have failed in tension in the past, type

```
plot block state tension-p
```

To plot all zones that have failed in shear in the past, or are failing in shear now, type

```
plot block state shear
```

To plot all zones that have *not* failed in shear, type

```
plot block state not shear
```

To plot all zones that have failed in tension *and* shear, type

```
plot block state tension shear
```

To plot all zones that have failed in tension in the past *and* shear now, type

```
plot block state tension-p shear-n
```

To plot the converse of the above, type

```
plot block state not tension-p shear-n
```

To plot all zones that have failed in the **past**, type

```
plot block state past
```

To plot all zones that are failing **now**, type

```
plot block state now
```

If the **now** and **past** keywords are used, *FLAC3D* searches for "-n" and "-p" at the end of the state descriptor string returned by the constitutive model. Keep this in mind if you are writing your own constitutive model.

The optional keywords **any** or **average** can also be specified at the end of the **PLOT block** state command line.

any plots plasticity state for state set in any tetrahedral sub-zone (default)

average plots plasticity state if more than 50% of tetrahedral sub-zones are at this state — for example,

```
plot block state shear average
```

zextra ***i***

the ***i***th extra zone variable

The following switches apply.

color — Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots. 24 colors are available; new colors are selected and assigned automatically if needed.

activate
off
on (default)

If **off**, the item is not plotted, but is still assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

behind — plots the plot item behind the current viewing plane (see **PLOT set plane**).

fast
off
on (default)

toggles the fast plotting mode. In fast plotting, the face-sorting is speeded up at the risk that some faces may be plotted out of sequence.

front — plots the plot item in front of the current viewing plane (see **PLOT set plane**).

gpnum
off (default)
on

gpnum on plots gridpoint numbers at surface face vertices or interface node IDs.

gppos
off (default)
on

plots the position of gridpoints.

id
off (default)
on

identification numbers of objects. Numbers are only displayed if their corresponding entities are also displayed.

outline **off**
on (default)
color (default is black)

If **outline** is **on**, an outline of the ball surface will be drawn in the outline color, which can be changed by specifying a color after the **outline** keyword. **outline** is **on**, by default.

plane plots the plot item on the current plotting plane (see **PLOT set plane**).

range keyword . . .

limits the plotting to a specified range. The available keywords are described in Section 1.1.3.

boundary <switches>

A boundary wireframe of all surface faces is added to the current view.

The following switches apply.

color — Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots.

activate — **off**
on (default)

If **off**, the item is not plotted, but is still assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias — name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

behind — plots the plot item behind the current viewing plane (see **PLOT set plane**).

front — plots the plot item in front of the current viewing plane (see **PLOT set plane**).

magfac — ***value***

controls gridpoint positions used for plotting, to create a deformed view of the boundary. ***value*** defines the magnification factor which will be applied to all translational gridpoint displacements. The plotted nodal position is the base position plus ***value*** times the translational grid displacement. Note: In large strain mode, the default magnification factor is 1.0. The user can see the original gridpoint position by resetting the magnification factor to 0.0.

null — **off** (default)
on

If **null** is **on**, only null zones for the plot item are plotted.

plane — plots the plot item on the current plotting plane (see **PLOT set plane**).

range keyword . . .

limits the plotting to a specified range. The available keywords are described in Section 1.1.3.

contour keyword <switches>

A contour surface rendering is added to the current view. Available switches are listed after the keywords.

The following keywords apply to gridpoint contours.

disp displacement magnitude

fob out-of-balance force

gpextra ***i***

the ***i***th extra gridpoint variable

ppressure pore pressure

temperature temperature

velocity velocity magnitude

xdisp x-displacement magnitude

xvelocity x-velocity magnitude

ydisp y-displacement magnitude

yvelocity y-velocity magnitude

zdisp z-displacement magnitude

zvelocity z-velocity magnitude

The following switches apply to gridpoint contour items.

color Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots. 24 colors are available; new colors are selected and assigned automatically if needed.

By default, contour colors are graded from blue to red. (Blue is the minimum value, and red is the maximum value.) The number of colors and color types can be changed with the color switches.

activate **off**
on (default)

If **off**, the item is not plotted, but is still assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

behind plots the plot item behind the current viewing plane (see **PLOT set plane**).

fast **off**
on (default)

toggles the fast plotting mode. In fast plotting, the face-sorting is speeded up at the risk that some faces may be plotted out of sequence.

front plots the plot item in front of the current viewing plane (see **PLOT set plane**).

gpnum **off** (default)
on

gpnum on plots gridpoint numbers at surface face vertices or interface node IDs.

gppos **off** (default)
on

plots the position of gridpoints.

id **off** (default)
on

identification numbers of objects. Numbers are only displayed if their corresponding entities are also displayed.

interval ***v***
auto (default)

sets the contour interval for contour plots. If **auto** is specified instead of ***v***, the interval is calculated automatically for a total of between eight and sixteen contours.

magfac ***value***

controls gridpoint positions used for plotting, to create a deformed view of the system. ***value*** defines the magnification factor which will be applied to all translational gridpoint displacements. The plotted nodal position is the base position plus ***value*** times the translational grid displacement. Note: In large strain mode, the default magnification factor is 1.0. The user can see the original gridpoint position by resetting the magnification factor to 0.0.

maximum ***v***
auto (default)

sets the maximum value for contour plots to ***v***. If the keyword **auto** is specified, the maximum value for the model is selected for ***v***.

minimum ***v***
auto (default)

sets the minimum value for contour plots to ***v***. If the keyword **auto** is specified, the minimum value for the model is selected for ***v***.

null **off** (default)
on

If **null** is **on**, only null zones for the plot item are plotted.

outline **off**
on (default)
color (default is black)

If **outline** is **on**, an outline of the ball surface will be drawn in the outline color, which can be changed by specifying a color after the **outline** keyword. **outline** is **off**, by default.

plane plots the plot item on the current plotting plane (see **PLOT set plane**).

range keyword . . .

limits the plotting to a specified range. The available keywords are described in Section 1.1.3.

reverse **off** (default)
on

reverses the color scale for contour plots.

shade **off**
on

Color shading, with respect to the light vector, is turned off or on (default is shading **on**). See **PLOT set light**.

The following keywords apply to zone contours.

density — zone mass density

smax — maximum principal stress (σ_3). Note that because compressive stresses are negative, this is the least negative (minor) principal stress.

smid — intermediate principal stress (σ_2)

smin — minimum principal stress (σ_1). Note that because compressive stresses are negative, this is the most negative (major) principal stress.

ssi — shear-strain increment

ssr — shear-strain rate

sxx — xx-stress

sxy — xy-stress

sxz — xz-stress

syy — yy-stress

syz — yz-stress

szz — zz-stress

vsi — volumetric strain increment

vsr — volumetric strain rate

zextra ***i***

the ***i***th extra zone variable

The following switches apply to zone contour items. (Note that **effective** and **total** apply only to stress contour items.)

color — Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots. 24 colors are available; new colors are selected and assigned automatically if needed.

By default, contour colors are graded from blue to red. (Blue is the minimum value, and red is the maximum value.) The number of colors and color types can be changed with the color switches.

activate **off**
on (default)

If **off**, the item is not plotted, but is still assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

average Zone-based contour plots are created based upon volume averaging of values to interpolate to gridpoints. This is less accurate, but faster than calculation based on the **gradient** calculation (see Note on p. 1 - 169).

behind plots the plot item behind the current viewing plane (see **PLOT set plane**).

effective **off** (default)
on

modifies zone stress contour plot to plot effective stress.

fast **off**
on (default)

toggles the fast plotting mode. In fast plotting, the face-sorting is speeded up, at the risk that some faces may be plotted out of sequence.

front plots the plot item in front of the current viewing plane (see **PLOT set plane**).

gpnum **off** (default)
on

gpnum on plots gridpoint numbers at surface face vertices or interface node IDs.

gppos **off** (default)
on

plots the position of gridpoints.

gradient Zone-based contour plots are created based upon gradient calculation to interpolate to gridpoints (see Note on p. 1 - 169). The **tolerance** keyword changes the tolerance for the gradient calculation. (**gradient** is default)

id **off** (default)
on

identification numbers of objects. Numbers are only displayed if their corresponding entities are also displayed.

interval ***v***
auto (default)

sets the contour interval for contour plots. If **auto** is specified instead of ***v***, the interval is calculated automatically for a total of between eight and sixteen contours.

magfac ***value***

controls the gridpoint positions used for plotting, to create a deformed view of the system. ***value*** defines the magnification factor which will be applied to all translational gridpoint displacements. The plotted nodal position is the base position plus ***value*** times the translational grid displacement. Note: In large strain mode, the default magnification factor is 1.0. The user can see the original gridpoint position by resetting the magnification factor to 0.0.

maximum ***v***
auto (default)

sets the maximum value for contour plots to ***v***. If the keyword **auto** is specified, the maximum value for the model is selected for ***v***.

minimum ***v***
auto (default)

sets the minimum value for contour plots to ***v***. If the keyword **auto** is specified, the minimum value for the model is selected for ***v***.

null **off** (default)
on

If **null** is **on**, only null zones for the plot item are plotted.

outline **off**
on (default)
color (default is black)

If **outline** is **on**, an outline of the ball surface will be drawn in the outline color, which can be changed by specifying a color after the **outline** keyword.

plane plots the plot item on the current plotting plane (see **PLOT set plane**).

range keyword . . .

limits the plotting to a specified range. The available keywords are described in Section 1.1.3.

reverse **off** (default)
on

reverses the color scale for contour plots.

shade **off**
on

Color shading, with respect to the light vector, is turned off or on (default is shading **on**). See **PLOT set light**.

tolerance ***v***

specifies the gradient tolerance used for zone-based contour plots (e.g., stress contour plots). The default value is $v = 1 \times 10^{-7}$. If the value is lower, the gradient is recalculated more often to produce a smoother contour. However, this will increase the calculation time for contour generation.

NOTE:

Contours are plotted on the surface of zones. Contour values are calculated as linear interpolations of values stored at surface gridpoints. In order to contour zone-based values (such as stresses and density), values that correspond to locations at a zone centroid must first be brought to the surface gridpoints for plotting. For calculation purposes, zone-based values are assumed to be constant within a zone. Linear interpolation is not an option, because the connectivity between zones is not clear; in many cases, a nonlinear extrapolation is required. For example, the normal stress exactly on a free surface should be zero; but the stress of the zone at the surface is usually nonzero, since the centroid for the surface zone is inside the grid. Because the stress state in the vicinity of the surface can be highly nonlinear, a linear interpolation is not sufficient to produce a zero stress at the surface gridpoint.

Two approaches are provided in *FLAC3D* to extrapolate zone-based values to a surface. The first approach is accessed with the keyword switch **average**. All zones simply contribute their values in a weighted fashion to all gridpoints connected to them, and the average of all contributions to each gridpoint is taken. This approach is simple and fast, but it can also be inaccurate for a constant linear stress field.

A second approach is accessed with the keyword switch **gradient**. In this method, a list of zones in the immediate vicinity of the gridpoint is found, and a best-fit gradient is determined based on those points. If the gradient is determined by using a fourth-order equation, then a constant linear gradient can be calculated. This has a tendency to overdetermine the extrapolated results. If an eighth-order equation is used, then a better fit can generally be found. The gradient method selects a list of zones and first tries to find a fully determined eighth-order equation for the best-fit gradient based on this list. If the gradient is not fully determined, then the algorithm tries to find more zones in the vicinity of the gridpoint. Finally, it will drop the gradient calculation to a fourth-order equation. In certain geometric cases (e.g., a planar wall of zones or a column of single elements), an indeterminate matrix may result. In this case, single-value-decomposition is used to determine the best result, but the contour values returned may still be misleading.

Example 1.4 illustrates the different contouring results that are produced with these two methods. Contour plots are created for three types of grid: a cubic grid; a planar wall of zones; and a column of single elements. Contours of σ_{zz} stress are plotted using the **average** switch (Figure 1.23) and the **gradient** switch (Figure 1.24).

Example 1.4 Stress contour plots for gravity loading of three types of grid

```
gen zone brick size 1 1 4 p0 (7,0,0)
gen zone brick size 1 4 4 p0 (5,0,0)
gen zone brick size 4 4 4
model elas
prop b 3e8 s 2e8
ini den 1000
set grav -10
fix z range z -0.1 0.1
solve
;
plot set title text 'Average zone contour calculation'
plot cont szz out on average
;
plot set title text 'Gradient zone contour calculation(Default)'
plot cont szz out on gradient
```

The **average** method, as illustrated in Figure 1.23, does not produce a uniform gradient for any of the grids, because the weighted average is not as accurate for the gridpoints at the top and bottom of the grids. However, this contour method is fast;

so, for large models, it may prove useful for giving an approximate view of stress distribution quickly.

The **gradient** method, as shown in Figure 1.24, produces reasonable contour plots for all three grid types.

It is important to be aware of the difficulty associated with contouring zone-based variables when viewing contour plots, especially for stress contours. The results plotted may not be representative of what is really calculated. If a plot looks suspicious, try switching between the various calculation methods.

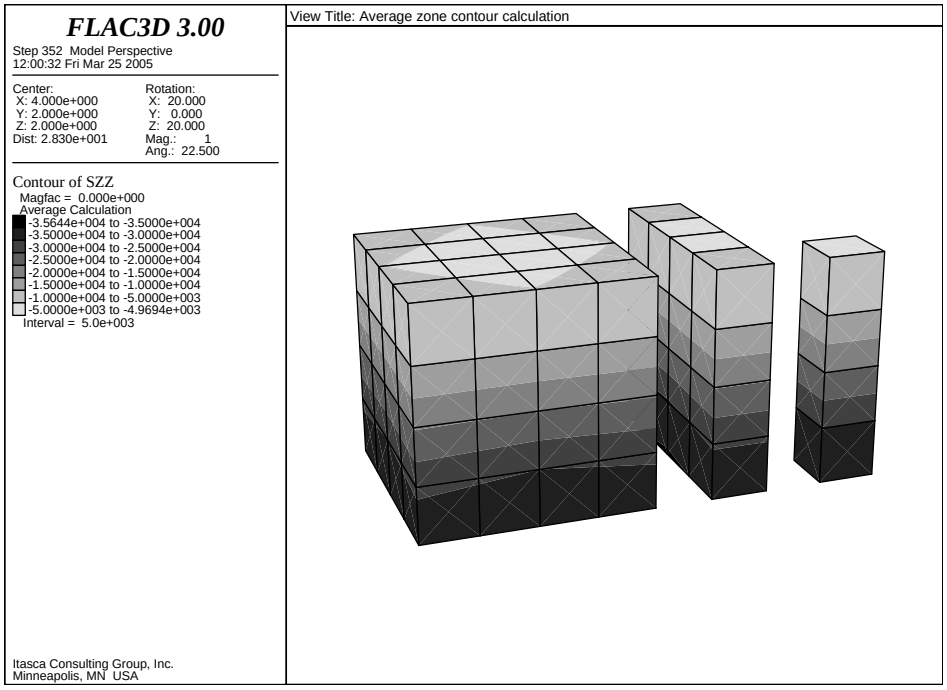


Figure 1.23 Average zone contour calculation

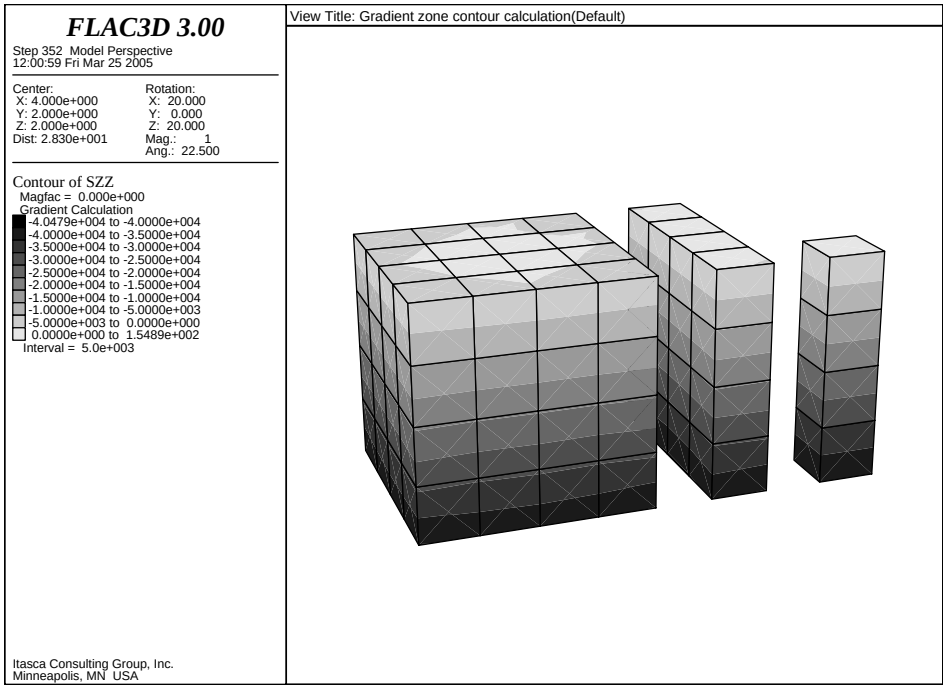


Figure 1.24 Gradient zone contour calculation (default)

displacement <switches>

A displacement vector plot is added to the view list.

The following switches apply.

color — Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots.

activate
off
on (default)

If **off**, the item is not plotted, but is still assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

behind — plots the plot item behind the current viewing plane (see **PLOT set plane**).

front — plots the plot item in front of the current viewing plane (see **PLOT set plane**).

maximum
v
auto (default)

scales plotted displacement vectors to ***v***. If the keyword **auto** is specified, the maximum value for the model is selected for ***v***.

outside
off
on

If **outside** is **on**, gridpoint vectors are plotted only on surfaces facing towards the viewer. (default is **off**)

plane — plots the plot item on the current plotting plane (see **PLOT set plane**).

range keyword . . .

limits the plotting to a specified range. The available keywords are described in Section 1.1.3.

scale ***v***

scales the object as a fraction ***v*** of the screen size.

fap <switches>

Applied force vectors (from applied forces and stresses) are added to the view list.

The following switches apply.

color — Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots.

activate
off
on (default)

If **off**, the item is not plotted, but is still assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

behind — plots the plot item behind the current viewing plane (see **PLOT set plane**).

front — plots the plot item in front of the current viewing plane (see **PLOT set plane**).

maximum
v
auto (default)

scales force vectors, as if the maximum value were ***v***. If the keyword **auto** is specified, the maximum value for the model is selected for ***v***.

outside
off
on

If **outside** is **on**, gridpoint vectors are plotted only on surfaces facing towards the viewer.

plane — plots the plot item on the current plotting plane (see **PLOT set plane**).

range keyword . . .

limits the plotting to a specified range. The available keywords are described in Section 1.1.3.

scale ***v***

scales the object as a fraction ***v*** of the screen size.

flow <switches>

Specific discharge vectors for fluid flow. Flow vectors are plotted at zone centroids.

The following switches apply.

color — Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots.

activate
off
on (default)

If **off**, the item is not plotted, but is still assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

behind — plots the plot item behind the current viewing plane (see **PLOT set plane**).

front — plots the plot item in front of the current viewing plane (see **PLOT set plane**).

maximum
v
auto (default)

scales plotted flow vectors, as if the maximum value were ***v***. If the keyword **auto** is specified, the maximum value for the model is selected for ***v***.

plane — plots the plot item on the current plotting plane (see **PLOT set plane**).

range keyword . . .

limits the plotting to a specified range. The available keywords are described in Section 1.1.3.

scale ***v***

scales the object as a fraction ***v*** of the screen size.

fob <switches>

Unbalanced force vectors at gridpoints are added to the view list.

The following switches apply.

color — Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots.

activate
off
on (default)

If **off**, the item is not plotted, but is still assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

behind — plots the plot item behind the current viewing plane (see **PLOT set plane**).

front — plots the plot item in front of the current viewing plane (see **PLOT set plane**).

maximum
v
auto (default)

scales plotted force vectors, as if the maximum value were ***v***. If the keyword **auto** is specified, the maximum value for the model is selected for ***v***.

outside
off
on

If **outside** is **on**, gridpoint vectors are plotted only on surfaces facing towards the viewer.

plane — plots the plot item on the current plotting plane (see **PLOT set plane**).

range keyword . . .

limits the plotting to a specified range. The available keywords are described in Section 1.1.3.

scale ***v***

scales the object as a fraction ***v*** of the screen size.

fos factor of safety calculated from **SOLVE fos** is added to the plot legend.

gpfix <switches>

Gridpoint fixity conditions are indicated by a line drawn in the direction that the gridpoint is fixed.

The following switches apply.

color Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots.

activate **off**
on (default)

If **off**, the item is not plotted, but is still assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

scale ***v***

scales the object as a fraction ***v*** of the screen size.

grid <switches>

A grid wireframe of all zones is added to the view list.

The following switches apply.

color — Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots.

activate — **off**
on (default)

If **off**, the item is not plotted, but is still assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias — name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

behind — plots the plot item behind the current viewing plane (see **PLOT set plane**).

front — plots the plot item in front of the current viewing plane (see **PLOT set plane**).

magfac — ***value***

controls gridpoint positions used for plotting, to create a deformed view of the grid. ***value*** defines the magnification factor which will be applied to all translational gridpoint displacements. The plotted nodal position is the base position plus ***value*** times the translational grid displacement. Note: In large strain mode, the default magnification factor is 1.0. The user can see the original gridpoint position by resetting the magnification factor to 0.0.

plane — plots the plot item on the current plotting plane (see **PLOT set plane**).

range — keyword . . .

limits the plotting to a specified range. The available keywords are described in Section 1.1.3.

history <-> ***n***. . . <switches>

A plot is made of the history (step number versus value) of a variable recorded in history number ***n***. Multiple histories can be plotted by giving several numbers in sequence. A minus sign in front of a history number reverses the sign of the history plotted.

Optional switches allow the user to manipulate the history. These switches are:

color — Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots. Up to 16 colors may be specified.

activate — **off**
on (default)

If **off**, the item is not plotted, but is still assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias — name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

begin — ***ncb***

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

both — Both a line and marks are used to plot the history record specified last.

default — <**both**> <**l**ine> <**mar**k>

specifies the default line style. Individual settings will override the default.

end — ***nce***

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

line — The history record specified most recently is plotted as a line (default).

mark — The history record specified most recently is plotted as crossed marks.

new clears all histories added to a view.

position ***xl yl xu yu***

describes a rectangle on the screen in which the plot will appear. The screen origin is the bottom-left corner. The values given are fractions of screen size.

skip ***nc***

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

vs ***nh***
step

enables plotting of one or more histories versus another. For example,

```
plot his 2 3 vs 4
```

plots histories 2 and 3 versus history 4; history 4 plots along the abscissa.

If the keyword **step** is given, in place of ***nh***, then the history is plotted versus step number (default).

xlabel 'string'

labels the x-axis with string.

xlog **off**
on

transforms the x-axis to a log scale. (default is linear)

xmaximum ***v*** <**a**uto>

sets the maximum value ***v*** for the abscissa of the plot.

xminimum ***v*** <**a**uto>

sets the minimum value ***v*** for the abscissa of the plot.

ylabel 'string'

labels the y-axis with string.

ylog **off**
on

transforms the *y*-axis to a log scale. (default is linear)

ymaximum ***v*** <**a**uto>

sets the maximum value ***v*** for the ordinate of the plot.

yminimum ***v*** <**a**uto>

sets the minimum value ***v*** for the ordinate of the plot.

interface <keyword> <switches>

An outline of all interface elements is added to the current view. The following optional keywords will plot contours of interface variables.

ndisp relative normal displacement (same as penetration)

nstress normal stress on interface elements

penetration interpenetration magnitude of interface nodes

sdisp relative shear displacement

sstress shear stress on interface elements

The following switches apply.

color Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots.

activate **off**
on (default)

If **off**, the item is not plotted, but is still assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

elnum **off** (default)
on

interface element's ID numbers are displayed. Numbers are only displayed if their corresponding entities are also displayed.

fast **off**
on (default)

toggles the fast plotting mode. In fast plotting, the face-sorting is speeded up at the risk that some faces may be plotted out of sequence.

gpnum **off** (default)
on

gnum on plots gridpoint numbers at surface face vertices or interface node IDs.

gppos **off** (default)
on

plots the position of gridpoints; works only with **interface** keyword.

id **off** (default)
on

identification numbers of objects. Numbers are only displayed if their corresponding entities are also displayed.

interval ***v***
auto (default)

sets the contour interval for contour plots. If **auto** is specified instead of ***v***, the interval is calculated automatically for a total of between eight and sixteen contours.

maximum ***v***
auto (default)

scales plotted vectors, as if the maximum value were ***v***, or sets the maximum value for contour plots to ***v***. If the keyword **auto** is specified, the maximum value for the model is selected for ***v***.

minimum ***v***
auto (default)

sets the minimum value for contour plots to ***v***. If the keyword **auto** is specified, the minimum value for the model is selected for ***v***.

null **off** (default)
on

If **null** is **on**, only null zones for the plot item are plotted.

outline **off**
on (default)
color (default is black)

If **outline** is **on**, an outline of the interface elements will be drawn in the outline color, which can be changed by specifying a color after the **outline** keyword.

reverse **off** (default)
on

reverses the color scale for contour plots.

shade **off**
on

Color shading, with respect to the light vector, is turned off or on (default is shading **on**). See **PLOT set light**.

slip **s**hear
normal

Draws a circle where the interface has slipped in the shear or normal direction.

location <***hn*** . . . > <switches>

The locations of history points (that have locations) are added to the view list. Supplying a list of history numbers ***hn*** will cause only those locations to be shown.

The following switches apply.

color — Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots. Only 1 color may be specified.

activate — **off**
on (default)

If **off**, the item is not plotted, but is still assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias — name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

all — plots all the available locations — overrides the optional ***hn*** list

behind — plots the plot item behind the current viewing plane (see **PLOT set plane**).

front — plots the plot item in front of the current viewing plane (see **PLOT set plane**).

plane — plots the plot item on the current plotting plane (see **PLOT set plane**).

sel keyword

The following SEL plot items are described: **beam**, **cable**, **displacement**, **fob**, **geogrid**, **geom**, **group**, **liner**, **pile**, **recover** and **velocity**. An optional range can be specified to limit the plotting of the SELs. The range can be given in several forms (see Section 1.1.3). If no range is specified, then the command applies to all SELs of that type.

beam keyword <switches> <**ra**nge . . . >

The **sel beam** plot item plots beam responses that include force and moment carried by the beam. The force and moment vectors are expressed in the beamSEL coordinate system — see Figure 1.25 for sign convention. Each component of the force and moment vectors is depicted as a tapered cylinder surrounding the beam, where cylinder radius is proportional to magnitude, and color designates sign.

If a beam is created using the **SEL beam** command, then the nodes of each beamSEL so created will be ordered such that the overall beam direction goes from the **begin** point to the **end** point — i.e., the nodal connectivity of each beamSEL will be ordered such that the direction from end-1 to end-2 corresponds with the direction from the **begin** point to the **end** point.

Force and moment are plotted using the following keywords.

force	keyword	
	fx	force (x-component, beamSEL system)
	fy	force (y-component, beamSEL system)
	fz	force (z-component, beamSEL system)
moment	keyword	
	mx	moment (x-component, beamSEL system)
	my	moment (y-component, beamSEL system)

mz moment (z-component, beamSEL system)

Available switches are:

color Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots. Only one color may be specified.

active **off**
on (default)

If **off**, the item is not plotted, but remains assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

magfac ***value***

controls nodal positions used for plotting, to create a deformed view of the system. The value of ***value*** defines the magnification factor which will be applied to the translational nodal displacements. The plotted nodal position is the base position plus ***value*** times the translational nodal displacement. The base position differs depending on whether one is running in small- or large-strain mode. In small-strain mode, the base position is the reference (or undeformed) position; in large-strain mode, the base position is the configuration corresponding to when the translational nodal displacements were zeroed.
Note that nodal displacements can be set or reset at any time using the **SEL node init** command. The current magnification factor is displayed in the plot legend.

maximum *v*
auto (default)

scales response magnitudes as if the maximum magnitude were ***v***. If the keyword **auto** is specified, then the maximum magnitude of all responses in the plot window is selected for ***v***.

scale *v*

scales the response magnitudes as a fraction ***v*** of the screen size.

cable keyword <switches> <**ra**nge . . . >

The **sel cable** plot item plots cable responses that include: force, stress and yield state of the cable itself; and stress and slip state of the shear coupling springs that represent the grout.

If a cable is created using the **SEL cable** command, then the nodes of each cableSEL so created will be ordered such that the overall cable direction goes from the **begin** point to the **end** point — i.e., the nodal connectivity of each cableSEL will be ordered such that the direction from end-1 to end-2 corresponds with the direction from the **begin** point to the **end** point.

1. Cable response plots

The force and stress quantities are depicted as a tapered cylinder surrounding the cable, where cylinder radius is proportional to magnitude, and color designates sign.

force average axial force in cable (positive: tension; negative: compression)

stress average axial stress in cable (positive: tension; negative: compression)

yield <**c**ompression>

The yield state of each cableSEL is depicted by drawing each cableSEL that is now yielding (using color-1) or has yielded in the past (using color-2). Those cableSELs that have never yielded are not drawn. If **compression** is specified, then the compressive yield state is drawn; otherwise, the tensile yield state is drawn.

2. Grout response plots

grout keyword

slip The grout slip state is depicted by drawing an open circle at each spring that is now slipping (using color-1) or has slipped in the past (using color-2). No circle is drawn if the spring has never slipped.

stress The grout stress is depicted as a tapered cylinder surrounding the cable, where cylinder radius is proportional to magnitude, and color designates sign. Sign convention is positive/negative with respect to average axial direction of cable.

Available switches are:

color Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots. Only one color may be specified.

active **off**
on (default)

If **off**, the item is not plotted, but remains assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

maximum ***v***
auto (default)

scales response magnitudes as if the maximum magnitude were ***v***. If the keyword **auto** is specified, then the maximum magnitude of all responses in the plot window is selected for ***v***. This switchword is not available for cable yield and grout slip.

scale ***value***

scales the response magnitudes as a fraction ***v*** of the screen size.

displacement <switches> <**ra**nge. . . >

A displacement-vector plot is added to the view list. The translational displacement of each node is drawn as an arrow, with length proportional to magnitude, and orientation equal to that of the displacement vector. Note that nodal displacements can be set or reset at any time using the **SEL node init** command.

Available switches are as follows.

color — Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots.

active — **off**
on (default)

If **off**, the item is not plotted, but remains assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias — name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

magfac — ***value***

controls nodal positions used for plotting, to create a deformed view of the system. The value of ***value*** defines the magnification factor which will be applied to the translational nodal displacements. The plotted nodal position is the base position plus ***value*** times the translational nodal displacement. The base position differs depending on whether one is running in small- or large-strain mode. In small-strain mode, the base position is the reference (or undeformed) position; in large-strain mode, the base position is the configuration corresponding to when the translational nodal displacements were zeroed.
Note that nodal displacements can be set or

reset at any time using the **SEL node init** command. The current magnification factor is displayed in the plot legend.

maximum *v*
auto (default)

scales displacement vectors as if the maximum magnitude were ***v***. If the keyword **auto** is specified, then the maximum magnitude of all vectors in the plot window is selected for ***v***.

scale *v*

scales the displacement vectors as a fraction ***v*** of the screen size.

fob <switches> <**ra**nge . . . >

An unbalanced force vector plot is added to the view list. The translational out-of-balance force (FOB) of each node is drawn as an arrow, with length proportional to magnitude, and orientation equal to that of the FOB vector.

Available switches are as follows.

color Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots. Only one color may be specified.

active **off**
on (default)

If **off**, the item is not plotted, but is still assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

fapp The applied nodal force vector is added to the view list. The applied force at each node is drawn as an arrow, with length proportional to magnitude, and orientation equal to that of the applied force vector.

magfac ***value***

controls nodal positions used for plotting, to create a deformed view of the system. The value of ***value*** defines the magnification factor which will be applied to the translational nodal displacements. The plotted nodal position is the base position plus ***value*** times the translational nodal displacement. The base position differs depending on whether one is running in small- or large-strain mode. In small-strain mode, the base position is the

reference (or undeformed) position; in large-strain mode, the base position is the configuration corresponding to when the translational nodal displacements were zeroed.
Note that nodal displacements can be set or reset at any time using the **SEL node init** command. The current magnification factor is displayed in the plot legend.

mapp The applied nodal moment vector is added to the view list. The applied moment at each node is drawn as an arrow, with length proportional to magnitude, and orientation equal to that of the applied moment vector.

maximum ***v***

scales FOB vectors as if the maximum magnitude were ***v***. If the keyword **auto** is specified, then the maximum magnitude of all vectors in the plot window is selected for ***v***.

scale ***v***

scales the FOB vectors as a fraction ***v*** of the screen size.

geogrid keyword <switch> <**ra**nge . . . >

The **sel geogrid** plot item plots stress and yield state in the shear coupling springs. The stresses in the geogrid itself can be viewed with the **sel recover** plot item.

Available keywords are as follows.

coupling keyword

stress The shear stress magnitude in the coupling springs is drawn as a colored contour on the surfaces of all geogridSELs in the plot-item range. The direction in which the stress is acting is not depicted but can be printed (**PRINT sel geogrid coupling stress**) or accessed by the *FISH* function **sg_rstrdir**.

yield The yield state of each shear coupling spring is depicted by drawing an open circle at each spring that is now yielding (using color-1) or has yielded in the past (using color-2). No circle is drawn if the spring has never yielded.

The following switchwords are available.

color Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots. Up to 16 colors may be specified.

active **off**
on (default)

If **off**, the item is not plotted, but remains assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

id **off** (default)
on

displays component-ID numbers of geogrid-SELs. Also see **scale**.

interval *v*
auto (default)

sets the contour interval for contour plots. If **auto** is specified instead of ***v***, the interval is calculated automatically for a total of between 8 and 16 contours.

magfac ***value***

controls nodal positions used for plotting, to create a deformed view of the system. The value of ***value*** defines the magnification factor which will be applied to the translational nodal displacements. The plotted nodal position is the base position plus ***value*** times the translational nodal displacement. The base position differs depending on whether one is running in small- or large-strain mode. In small-strain mode, the base position is the reference (or undeformed) position; in large-strain mode, the base position is the configuration corresponding to when the translational nodal displacements were zeroed.
Note that nodal displacements can be set or reset at any time using the **SEL node init** command. The current magnification factor is displayed in the plot legend.

maximum *v*
auto (default)

scales plotted velocity of force vectors, or stress tensors, as if the maximum value were ***v***, or sets the maximum value of contour plots to ***v***. If the keyword **auto** is specified, the maximum value for the model is selected for ***v***.

minimum ***v***
auto (default)

sets the minimum value for contour plots to ***v***. If the keyword **auto** is specified, the minimum value for the model is selected for ***v***.

outline **off**
on (default)

If **outline** is on, a wire-frame outline is drawn around the boundaries of each shell-type SEL.

plane plots the plot item on the current viewing plane (see **PLOT set plane**).

scale ***v***

scales the plotted entities (IDs) as a fraction ***v*** of the screen size.

shade **off** (default)
on

color shading, with respect to the light vector, is turned on or off (default is **off**). See **PLOT set light**.

geom <switches> <**ra**nge . . . >

The **sel geom** plot item plots the six different SEL types, the nodes and the links. The ID numbers of the nodes and links can also be viewed as well as the ID and CID numbers of the SELs. In addition, the SEL coordinate systems, the node-local coordinate systems and the surface coordinate system (used during stress recovery) can be viewed.

Available switches are as follows.

color — Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots. Up to 16 colors may be specified.

active — **off**
on (default)

If **off**, the item is not plotted, but remains assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

beam — **off**
on

draws all beamSELs in the plot-item range. Each beamSEL is depicted as a straight line between its two end nodes. Also see **magfac** and **shrinkfac**.

cable — **off**
on

draws all cableSELs in the plot-item range. Each cableSEL is depicted as a straight line between its two end nodes. Also see **magfac** and **shrinkfac**.

cid **off** (default)
on

displays component-ID numbers of SELs that are on. Also displays ID numbers of nodes or links that are on. (Nodes and links do not have CID numbers but, for these entities, the CID number is considered equivalent to the ID number.) If **cid** is on, then **id** will be set to off. Also see **scale**.

fill **off** (default)
on

If **fill** is on, shell-type SELs are drawn as filled polygons. Also see **outline**.

geogrid **off**
on

draws all geogridSELs in the plot-item range. Each geogridSEL is depicted as a wire-frame and/or a filled polygon based upon the settings of **outline** and **fill**. Also see **magfac** and **shrinkfac**.

id **off** (default)
on

displays ID numbers of SELs, nodes and links that are on. If **id** is on, then **cid** will be set to off. Also see **scale**.

liner **off**
on

draws all linerSELs in the plot-item range. Each linerSEL is depicted as a wire-frame and/or a filled polygon based upon the settings of **outline** and **fill**. Also see **magfac** and **shrinkfac**.

link **off** (default)
on

draws all links in the plot-item range. An open circle depicts each link location, which is taken equal to that of its associated node. If **link** is on, then **node** will be set to off. Also see **magfac** and **scale**.

magfac ***value***

controls nodal positions used for plotting, to create a deformed view of the system. The value of ***value*** defines the magnification factor which will be applied to the translational nodal displacements. The plotted nodal position is the base position plus ***value*** times the translational nodal displacement. The base position differs depending on whether one is running in small- or large-strain mode. In small-strain mode, the base position is the reference (or undeformed) position; in large-strain mode, the base position is the configuration corresponding to when the translational nodal displacements were zeroed.
Note that nodal displacements can be set or reset at any time using the **SEL node init** command. The current magnification factor is displayed in the plot legend.

node **off**
on (default)

draws all nodes in the plot-item range. An open circle depicts each nodal location. If **node** is on, then **link** will be set to off. Also see **magfac** and **scale**.

nodesys **off** (default)
on

draws the node-local coordinate system at all nodal locations in the plot-item range. The system is drawn even if the nodes are off. A triad depicts the system orientation as follows. The x-axis is drawn as a full-length double line of single thickness; the y-axis is drawn as a half-length single line of single thickness; and the z-axis is drawn as a full-length single line of double thickness. If **nodesys** is on, then both **selsys** and **surfsys** will be set to off.

outline **off**
on (default)

If **outline** is on, shell-type SELs are drawn as wire-frame images. Also see **fill**.

pile **off**
on

draws all pileSELs in the plot-item range. Each pileSEL is depicted as a straight line between its two end nodes. Also see **magfac** and **shrinkfac**.

scale ***v***

scales the plotted entities (nodes, links, CIDs and IDs) as a fraction ***v*** of the screen size.

selsys **off** (default)
on

draws the SEL coordinate system at the centroid of SELs that are on. A triad depicts the system orientation as follows. The x-axis is drawn as a full-length double line of single thickness; the y-axis is drawn as a half-length single line of single thickness; and the z-axis is drawn as a full-length single line of double thickness. If **selsys** is on, then both **nodesys** and **surfsys** will be set to off.

shell **off**
on

draws all shellSELs in the plot-item range. Each shellSEL is depicted as a wire-frame and/or a filled polygon based upon the settings of **outline** and **fill**. Also see **magfac** and **shrinkfac**.

shrinkfac ***value***

controls the size at which SELs are drawn. By shrinking the SELs slightly, the individual SELs can be more easily seen. The SEL size is scaled by ***value*** $\in$ [0.0, 1.0], corresponding with [full size, zero size]. By default, ***value*** equals 0.1.

surfsys **off** (default)
on

draws the surface coordinate system (used during stress recovery) at all nodal locations in the plot-item range. The system is drawn even if the nodes are off. If the surface system is nor valid, then a filled sphere is drawn; otherwise, a triad depicts the system orientation as follows. The *x*-axis is drawn as a full-length double line of single thickness; the *y*-axis is drawn as a half-length single line of single thickness; the *z*-axis is drawn as a full-length single line of double thickness. If **surfsys** is on, then both **selsys** and **nodesys** will be set to off.

group <switches> <**ra**nge . . . >

The **sel group** plot item plots group membership of all SELs. All SELs in the plot-item range are drawn as filled polygons, with different colors indicating their group membership. SELs not in any group are considered part of group NONE and are drawn using color-1. The colors are either those specified for each group (see the **SEL group** command) or are automatically chosen. A legend containing group names and colors is displayed in the caption area.

Available switches are as follows.

color Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots. Up to 16 colors may be specified.

active **off**
on (default)

If **off**, the item is not plotted, but remains assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

magfac ***value***

controls nodal positions used for plotting, to create a deformed view of the system. The value of ***value*** defines the magnification factor which will be applied to the translational nodal displacements. The plotted nodal position is the base position plus ***value*** times the translational nodal displacement. The base position differs depending on whether one is running in small- or large-strain mode. In small-strain mode, the base position is the

reference (or undeformed) position; in large-strain mode, the base position is the configuration corresponding to when the translational nodal displacements were zeroed.
Note that nodal displacements can be set or reset at any time using the **SEL node init** command. The current magnification factor is displayed in the plot legend.

shrinkfac ***value***

controls the size at which SELs are drawn. By shrinking the SELs slightly, the individual SELs can more easily be seen. The SEL size is scaled by ***value*** $\in$ [0.0, 1.0], corresponding with [full size, zero size]. By default, ***value*** equals 0.1.

liner keyword <switches> <**ra**nge . . . >

The **sel liner** plot item plots stress and yield state in both the normal and shear coupling springs. The stresses in the liner itself can be viewed with the **sel recover** plot item.

Available keywords are as follows.

coupling keyword

stress keyword

The stress magnitude in either the normal or shear coupling springs is drawn as a colored contour on the surfaces of all linerSELs in the plot-item range.

normal Stress magnitude in normal coupling springs. Sign convention is positive/negative, indicating separation/overlap.

shear Stress magnitude in shear coupling springs. All values are positive. The direction in which the shear stress is acting is not depicted, but can be printed (**PRINT sel liner coupling stress**) or accessed by the *FISH* function `sl_rstrdir`.

yield **s**hear

The yield state of each shear coupling spring is depicted by drawing an open circle at each spring that is now yielding (using color-1) or has yielded in the past (using color-2). No circle is drawn if the spring has never yielded.

Available switchwords are as follows.

color — Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots. Up to 16 colors may be specified.

active — **off**
on (default)

If **off**, the item is not plotted, but remains assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias — name

changes name of variable in caption legend. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

id — **off** (default)
on

displays component-ID numbers of linerSELs. Also see **scale**.

interval — ***v***
auto (default)

sets the contour interval for contour plots. If auto is specified instead of ***v***, the interval is calculated automatically for a total of between 8 and 16 contours.

magfac — ***value***

controls nodal positions used for plotting, to create a deformed view of the system. The value of ***value*** defines the magnification factor which will be applied to the translational nodal displacements. The plotted nodal position is the base position plus ***value*** times the translational nodal displacement. The base position differs depending on whether one is running in small- or large-strain mode. In small-strain mode, the base position is the

reference (or undeformed) position; in large-strain mode, the base position is the configuration corresponding to when the translational nodal displacements were zeroed. Note that nodal displacements can be set or reset at any time using the **SEL node init** command. The current magnification factor is displayed in the plot legend.

maximum ***v***
auto (default)

scales plotted velocity of force vectors, or stress tensors, as if the maximum value were ***v***, or sets the maximum value of contour plots to ***v***. If the keyword **auto** is specified, the maximum value for the model is selected for ***v***.

minimum ***v***
auto (default)

sets the minimum value for contour plots to ***v***. If the keyword **auto** is specified, the minimum value for the model is selected for ***v***.

outline **off**
on (default)

If **outline** is on, a wire-frame outline is drawn around the boundaries of each shell-type SEL.

plane plots the plot item on the current viewing plane (see **PLOT set plane**).

scale ***v***

scales the plotted entities (IDs) as a fraction ***v*** of the screen size.

shade **off** (default)
on

color shading, with respect to the light vector, is turned on or off (default is off). See **PLOT set light**.

pile keyword <switches> <**ra**nge . . . >

The **sel pile** plot item plots pile responses that include force and moment acting on the pile itself, and stress and yield state in both the normal and shear coupling springs.

If a pile is created using the **SEL pile** command, then the nodes of each pileSEL so created will be ordered such that the overall pile direction goes from the **begin** point to the **end** point — i.e., the nodal connectivity of each pileSEL will be ordered such that the direction from end-1 to end-2 corresponds with the direction from the **begin** point to the **end** point.

1. Force and moment plots

The force and moment vectors are expressed in the pileSEL local coordinate system — see Figure 1.25 for sign convention. Each component of the force and moment vectors is depicted as a tapered cylinder surrounding the pile, where cylinder radius is proportional to magnitude, and color designates sign.

force keyword

fx	force (x-component, pileSEL system)
fy	force (y-component, pileSEL system)
fz	force (z-component, pileSEL system)

moment keyword

mx	moment (x-component, pileSEL system)
my	moment (y-component, pileSEL system)
mz	moment (z-component, pileSEL system)

2. Coupling spring plots

Coupling-spring quantities include the stress and yield state of both the normal and shear springs.

coupling keyword

stress keyword

The coupling spring stresses are depicted as a tapered cylinder surrounding the pile, where cylinder radius is proportional to magnitude, and color designates sign.

normal Stress in normal coupling springs. Sign convention is positive/negative, indicating separation/overlap.

shear Stress is shear coupling springs. Sign convention is positive/negative with respect to average axial direction of pile.

yield keyword

The coupling-spring yield state is depicted by drawing an open circle at each spring that is now yielding (using color-1) or has yielded in the past (using color-2). No circle is drawn if the spring has never yielded.

normal yield state of normal coupling spring

shear yield state of shear coupling spring

Available switches are:

color Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots. Only one color may be specified.

active **off**
on (default)

If **off**, the item is not plotted, but remains assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

magfac ***value***

controls nodal positions used for plotting, to create a deformed view of the system. The value of ***value*** defines the magnification factor which will be applied to the translational nodal displacements. The plotted nodal position is the base position plus ***value*** times the translational nodal displacement. The base position differs depending on whether one is running in small- or large-strain mode. In small-strain mode, the base position is the reference (or undeformed) position; in large-strain mode, the base position is the configuration corresponding to when the translational nodal displacements were zeroed.
Note that nodal displacements can be set or reset at any time using the **SEL node init** command. The current magnification factor is displayed in the plot legend.

maximum ***v***
auto (default)

scales response magnitudes as if the maximum magnitude were ***v***. If the keyword **auto** is specified, then the maximum magnitude of all responses in the plot window is selected for ***v***. This switchword is not available for coupling-spring yield state.

scale ***v***

scales the response magnitudes as a fraction ***v*** of the screen size.

recover keyword <switches> <**ra**nge . . . >

The stress-recovery plot item plots contoured images of displacements, and stress resultants and stresses of all shell-type SELs. The stress resultants and stresses are not computed during normal timestepping, but are instead recovered by the **SEL recover sres** and **SEL recover stress** commands, which require that a consistent surface coordinate system exist (set by the **SEL recover surface** command). Note that the recovered quantities will depend upon the range of SELs over which these quantities are being recovered, because nodal averaging only occurs for SELs within this range. The range of SELs considered, and which define the surface patch, are stored by this plot item.

Each time this plot item is drawn, if the recovered quantities are no longer valid, then the logic of the **SEL recover sres** or **SEL recover stress** command will be invoked for the given range of SELs. If the surface system ever becomes invalid, then the plot item will attempt to regenerate the surface system automatically by invoking the logic of the **SEL recover surface** command using the given value of **surfx** (a switchword of this plot item) for the given range of SELs.

The **regen** switchword controls the automatic regeneration of the surface coordinate system as follows. If **regen** is on, then the surface system will be regenerated (for all SELs in the range associated with the plot item and using its **surfx** vector) each time the plot item is drawn; otherwise, the surface system will only be regenerated if the surface is no longer valid. By setting **regen** to off, the surface system can be set by the user as necessary, and it will not be modified by this plot item unless the surface system becomes invalid.

1. Displacement contours

The displacements are obtained from the displacement values stored at the nodes used by the shell-type SELs in the plot-item range. The following displacement contours are available.

displace magnitude of displacement vector.

xdisplace x-displacement (global system)

ydisplace y-displacement (global system)

zdisplace z-displacement (global system)

2. Stress resultant contours

Stress resultants are expressed in the surface system associated with the shell-type SELs in the plot-item range. The following stress-resultant contours are available.

sres keyword

mx stress resultant M_x (surface system)

my stress resultant M_y (surface system)

mxy stress resultant M_{xy} (surface system)

nx stress resultant N_x (surface system)

ny stress resultant N_y (surface system)

nxy stress resultant N_{xy} (surface system)

qx stress resultant Q_x (surface system)

qy stress resultant Q_y (surface system)

3. Stress contours

Stresses are expressed in the global system and correspond with a shell depth given by the switchword **depth_fac**. The following stresses are contoured based upon the keywords:

stress keyword

xx stress (xx-component, global system)

yy stress (yy-component, global system)

zz stress (zz-component, global system)

xy stress (xy-component, global system)

zy stress (zy-component, global system)

zx stress (zx-component, global system)

pstress keyword

Principal stresses (σ_1, σ_2 and σ_3) are assigned such that compressive stresses are negative and $\sigma_1 \leq \sigma_2 \leq \sigma_3$. Principal stresses correspond with a shell depth given by the switchword **depth_fac**. The principal stresses are contoured based upon the keywords:

1 principal stress σ_1

2 principal stress σ_2

3 principal stress σ_3

The following switchwords are available.

color Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots. Up to 16 colors may be specified.

active **off**
on (default)

If **off**, the item is not plotted, but remains assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

depth_fac ***v***

Depth at which stresses will be recovered by this plot-item. (Applies only to stress contours.) The depth equals ***v*** times $t/2$, where t is shell thickness. The depth factor, ***v***, must be in the range [-1, +1].

id **off** (default)
on

displays component-ID numbers of shell-type SELs. Also see **scale**.

interval ***v***
auto (default)

sets the contour interval for contour plots. If **auto** is specified instead of ***v***, the interval is calculated automatically for a total of between 8 and 16 contours.

magfac ***value***

controls nodal positions used for plotting, to create a deformed view of the system. The value of ***value*** defines the magnification factor which will be applied to the translational nodal displacements. The plotted nodal position is the base position plus ***value*** times the translational nodal displacement. The base position differs depending on whether one is running in small- or large-strain mode. In small-strain mode, the base position is the reference (or undeformed) position, whereas in large-strain mode, the base position is the configuration corresponding to when the translational nodal displacements were zeroed. Note that nodal displacements can be set or reset at any time using the **SEL node init** command. The current magnification factor is displayed in the plot legend.

maximum ***v***
auto (default)

scales plotted velocity of force vectors, or stress tensors, as if the maximum value were ***v***, or sets the maximum value of contour plots to ***v***. If the keyword **auto** is specified, the maximum value for the model is selected for ***v***.

minimum ***v***
auto (default)

sets the minimum value for contour plots to ***v***. If the keyword **auto** is specified, the minimum value for the model is selected for ***v***.

outline **off**
on (default)

If **outline** is on, a wire-frame outline is drawn around the boundaries of each shell-type SEL.

plane plots the plot item on the current viewing plane (see **PLOT set plane**).

regen **off** (default)
on

controls the automatic regeneration of the surface coordinate system as follows. If **regen** is on, then the surface system will be regenerated (for all SELs in the range associated with the plot item and using its **surfx** vector) each time that the plot item is drawn; otherwise, the surface system will only be regenerated if the surface is no longer valid. By setting **regen** to off, the surface system can be set by the user as necessary, and it will not be modified by this plot item unless the surface system becomes invalid. **regen** applies only to stress and stress-resultant contours.

scale ***v***

scales the plotted entities (IDs) as a fraction ***v*** of the screen size.

shade **off** (default)
on

color shading, with respect to the light vector, is turned on or off (default is **off**). See **PLOT set light**.

surefsys **off**
on (default)

The surface system associated with the nodes of all shell-type SELs in the plot-item range is depicted at each such node as follows. If the surface system is not valid, then a filled sphere is drawn; otherwise, a triad aligned with the surface system is drawn. The x-axis is drawn as a full-length double line of single thickness; the y-axis is drawn as a half-length single line of single thickness; the z-axis is drawn as a full-length single line of double thickness. **surefsys** applies only to stress and stress-resultant contours.

surfx ***xx xy xz*** (stress and stress-resultant contours only)

The **surfx** vector is used by the plot-item whenever it attempts to regenerate the surface system for the shell-type SELs in its range — see discussion at the beginning of this plot item. If the **surfx** vector is not specified, then the plot-item attempts to choose (in the given order): (1, 0, 0), (0, 1, 0) and (0, 0, 1). If all of these choices fail (because **surfx** is parallel with the normal of a shell-type SEL in the plot-item range), then **surfx** is set to (1, 0, 0) and an error message will be issued if the plot item attempts to recover stresses or stress resultants. In this case, an approximate **surfx** vector must be specified by the user.

velocity <switches> <**ra**nge . . . >

A velocity-vector plot is added to the view list. The translational velocity of each node is drawn as an arrow, with length proportional to magnitude, and orientation equal to that of the velocity vector.

Available switches are as follows.

color Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots. Only one color may be specified.

active **off**
on (default)

If **off**, the item is not plotted, but remains assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

magfac ***value***

controls nodal positions used for plotting, to create a deformed view of the system. The value of ***value*** defines the magnification factor which will be applied to the translational nodal displacements. The plotted nodal position is the base position plus ***value*** times the translational nodal displacement. The base position differs depending on whether one is running in small- or large-strain mode. In small-strain mode, the base position is the reference (or undeformed) position, whereas in large-strain mode, the base position is the configuration corresponding to when the translational nodal displacements were zeroed. Note that nodal displacements can be set or reset at any time using the **SEL node init** command. The current magnification factor is displayed in the plot legend.

maximum ***v***
auto (default)

scales velocity vectors as if the maximum magnitude were ***v***. If the keyword **auto** is specified, then the maximum magnitude of all vectors in the plot window is selected for ***v***.

scale ***v***

scales the velocity vectors as a fraction ***v*** of the screen size.

sketch <switches>

A wireframe of surface faces facing the viewer is added to the view list. (No hidden-line removal is done in order to speed up the plotting.)

The following switches apply.

color Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots.

activate **off**
on (default)

If **on**, the plot item is plotted when **PLOT show** or **PLOT hard** is given. If **off**, the item is not plotted, but is still assigned to the plot view.

alias name

changes name of variable in legend caption. name can be a single token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

stensor keyword. . . <switches>

The three-component principal stress tensor (total stress at the zone centroid) is added to the view list. The stress tensor is plotted on the exposed zone surfaces by default. The following keywords are available.

effective **off**
on

modifies stress tensor plot to show effective stress (default is **off**).

local **off**
on

principal stresses plotted as crosses with magnitude and direction of the cross lines based upon the principal stress acting in the plane on which crosses are plotted (default is **off**).

transparent **off**
on

removes (**on**) or adds (**off**) the fill color from the zone, exposing or hiding elements behind the foreground object. The transparent view may appear cluttered because of the visibility of background object. However, choosing **off** could cause the fill color to mask tensors protruding beyond zone boundaries in adjacent zones.

The following switches apply.

color Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots.

activate **off**
on (default)

If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given. If **off**, the item is not plotted, but is still assigned to the plot view.

alias name

changes name of variable in legend caption. name can be a single token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

behind plots the plot item behind the current viewing plane (see **PLOT set plane**).

front plots the plot item in front of the current viewing plane (see **PLOT set plane**).

maximum ***v***
auto (default)

scales plotted velocity or force vectors, or stress tensors, as if the maximum value were ***v***, or sets the maximum value for contour plots to ***v***. If the keyword **auto** is specified, the maximum value for the model is selected for ***v***.

range keyword. . .

limits the plotting to a specified range. The available keywords are described in Section 1.1.3.

surface <switches>

A color-rendered surface plot is added to the view list. If the optional switch **id** is turned **on**, then zone numbers are also plotted. (The default is **id off**.)

The following switches apply.

color — Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots.

activate — **off**
on (default)

If **off**, the item is not plotted, but is still assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias — name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

behind — plots the plot item behind the current viewing plane (see **PLOT set plane**).

fast — **off**
on (default)

toggles the fast plotting mode. In fast plotting, the face-sorting is speeded up at the risk that some faces may be plotted out of sequence.

front — plots the plot item in front of the current viewing plane (see **PLOT set plane**).

gpnum — **off** (default)
on

gpnum on plots gridpoint numbers at surface face vertices or interface node IDs.

gppos **off** (default)
on

plots the position of gridpoints. Note: The *x*-,*y*-,*z*-coordinates of any position on the surface of a model will be displayed by double-clicking the left-mouse button on the position of the surface plot. If **SET log on**, the coordinates are copied to the log file.

id **off** (default)
on

identification numbers of objects. Numbers are only displayed if their corresponding entities are also displayed.

magfac ***value***

controls gridpoint positions used for plotting to create a deformed view of the surface. ***value*** defines the magnification factor which will be applied to all translational gridpoint displacements. the plotted nodal position is the base position plus ***value*** times the translational grid displacement. Note: In large strain mode, the default magnification factor is 1.0. The user can see the original gridpoint position by resetting the magnification factor to 0.0.

null **off** (default)
on

If **null** is **on**, only null zones for the plot item are plotted.

outline **off**
on (default)
color (default is black)

If **outline** is **on**, an outline of the ball surface will be drawn in the outline color, which can be changed by specifying a color after the **outline** keyword. **outline** is **on**, by default.

plane plots the plot item on the current plotting plane (see **PLOT set plane**).

range keyword . . .

limits the plotting to a specified range. The available keywords are described in Section 1.1.3.

shade **off**
on

Color shading, with respect to the light vector, is turned off or on (default is shading **on**). See **PLOT set light**.

table ***n*** . . . <switches>

A plot of table number ***n*** is made (see the **TABLE** command). Multiple tables can be plotted by giving several numbers in sequence.

Optional switches can be used to allow the user to manipulate the table. These keywords are:

color — Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots. Up to 16 colors may be specified.

activate **off**
on (default)

If **off**, the item is not plotted, but is still assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

begin ***ncb***

Tables will be plotted beginning with the ***ncb*** item in the table.

both Both a line and marks are used to plot the last specified table item.

default <**l**ine> <**mar**k> <**both**>

specifies the default line style. Individual settings will override the default. (**mark** is default)

end ***nce***

Tables will be plotted ending with the ***nce*** item in the table.

line The last specified table item is plotted as a line.

mark The last specified table item is plotted as crossed marks (default).

new clears all tables added to a view.

position ***xl yl xu yu***

describes a rectangle on the screen in which the plot will appear. The screen origin is the bottom-left corner. The values given are fractions of screen size.

xlabel 'string'

labels the *x*-axis with string.

xlog **off**
on

transforms the *x*-axis to a log scale. (default is linear)

xmaximum ***v*** <**a**uto>

sets the maximum value ***v*** for the abscissa of the plot.

xminimum ***v*** <**a**uto>

sets the minimum value ***v*** for the abscissa of the plot.

ylabel 'string'

labels the *y*-axis with string.

ylog **off**
on

transforms the *y*-axis to a log scale. (default is linear)

ymaximum ***v*** <**a**uto>

sets the maximum value ***v*** for the ordinate of the plot.

yminimum ***v*** <**a**uto>

sets the minimum value ***v*** for the ordinate of the plot.

track keyword color

The tracks of particles flowing through the grid are plotted. The following keywords apply.

head

The current position of the particle is plotted as a tiny "*x*."

tail

The path traced by the particle is plotted as a solid line. By default only the tail is plotted.

The color switch applies to this command. Table 1.7 lists the color switches that may be used; e.g., **PLOT track** red plots the tracks in red.

velocity <switches>

A velocity-vector plot is added to the view list.

The following switches apply.

color Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots. Only 1 color may be specified.

activate **off**
on (default)

If **off**, the item is not plotted, but is still assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

maximum ***v***
auto (default)

scales plotted velocity vectors, as if the maximum value were ***v***. If the keyword **auto** is specified, the maximum value for the model is selected for ***v***.

range keyword . . .

limits the plotting to a specified range. The available keywords are described in Section 1.1.3.

scale ***v***

scales the object as a fraction ***v*** of the screen size.

volume <switches>

A color-rendered plot of the 3D volumes, created with the **GENERATE surface** command, is added to the view list.

The following switches apply.

color — Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots.

activate — **off**
on (default)

If **off**, the item is not plotted, but is still assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias — name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

range — keyword . . .

limits the plotting to a specified range. The available keywords are described in Section 1.1.3.

water <switches>

The phreatic surface, created with the **WATER table face** command, is added to the view list.

The following switches apply.

color — Table 1.7 lists the color switches that may be used, and the corresponding screen color and fill shade for b/w hardcopy plots.

activate
off
on (default)

If **off**, the item is not plotted, but is still assigned to the plot view. If **on**, this plot item is plotted when **PLOT show** or **PLOT hard** is given.

alias name

changes name of variable in legend caption. name can be a single-token *FISH* variable. A sequence of words can be given for name if they are contained within single quotes.

behind — plots the plot item behind the current viewing plane (see **PLOT set plane**).

front — plots the plot item in front of the current viewing plane (see **PLOT set plane**).

link
off
on (default)

causes objects to be plotted in a depth sort with all other objects that have **link on**.

outline
off
on (default)
color (default is black)

If **outline** is **on**, an outline of the ball surface will be drawn in the outline color, which can be changed by specifying a color after the **outline** keyword.

plane — plots the plot item on the current plotting plane (see **PLOT set plane**).

range keyword . . .

limits the plotting to a specified range. The available keywords are described in Section 1.1.3.

PRINT keyword <keyword> . . . <**ra**nge . . . >

Output is displayed according to the keyword(s), below. If a **range** is specified (see Section 1.1.3), then the output will be restricted to the given range. Furthermore, the range will be remembered on subsequent **PRINT** commands (i.e., there is no need to repeat the **range**). The range may be reset by giving the command **PRINT range** without parameters.

The following keywords may be used.

apply values and types of applied conditions at zone faces

attach Attached gridpoint and face data are displayed. This information includes the weighting factors for the gridpoints.

creep creep calculation-mode information

directory prints name of current working directory

dynamic dynamic calculation-mode information

fish all *FISH* variables and their current values

fishcall all current associations between ID numbers and *FISH* functions (see Table 2.3 in the ***FISH* volume**)

fluid fluid-flow calculation-mode information

generate keyword

grid generation data. The following keywords are available to display selected grid generation data.

point list of all reference points

surface list of all surface elements of 3D volumes

volume list of all closed 3D volumes

gp <keyword>

gridpoint data. An optional keyword can be specified to display selected gridpoint data. The following keywords apply.

damp gridpoint damping parameters

displacement gridpoint displacements

extra ***i***

extra gridpoint variable for extra array index ***i***

force gridpoint forces

information general gridpoint information, gridpoint mass, fixity condition and applied-velocity condition (This is the default option if no keyword is given.)

position position coordinates of gridpoint

pp gridpoint pore pressure

temperature gridpoint temperature

velocity gridpoint velocities

group summary of existing groups selected for the model

history summary data on stored histories (identification, minimum and maximum values). Each history has a sequence number that may be used for identification.

information values of global variables and parameter settings

interface <keyword>

summary data on interfaces. An optional keyword may be specified to display selected interface data. The following keywords apply.

cellspace description of the surface-face cell space

i keyword

Selected data can be displayed for interface ***i*** using the following keywords.

area normal vector and characteristic area of interface node

ctolerance contact tolerance for new contacts

displacement displacement at nodes

element nodes, area and normal vector of element

host information on host face

join elements joined along edges

model interface model name

position location of interface nodes

prestress	normal prestress imposed by the **INTERFACE *i* nstress** command
property	name
	value of property name (See **INTERFACE property** for a list of property names.)
shear	accumulated shear displacement vector
small	accumulated displacement of interacting faces, in small-strain mode
state	current state of interface node
stress	shear and normal stresses and shear stress direction
target	target face information including penetration
velocity	velocity at nodes

macro list of assigned macros

memory <keyword>

describes *FLAC3D*'s memory usage. By default, the approximate amount of memory currently in use is printed. The following optional keywords apply.

save	total memory required to save the current state of the model.
system	total memory currently in use and total memory allocated by the system. *Warning:* this could potentially take a long time to execute under Windows if virtual memory is being used. The system may not release all the memory when a **NEW** command is issued, showing a large discrepancy between the memory *FLAC3D* thinks it's using and the memory the system has allocated to *FLAC3D*. In this situation it is better to exit from *FLAC3D* and restart it. Note that at least 4 to 6 MB should be left for Windows overhead; otherwise, virtual RAM management will drastically reduce *FLAC3D*'s performance.

model list of available material models

rlist <name>

list of all named ranges selected for the model. If a range name is given, a detailed description of the named range is displayed.

sel keyword . . .

information about the structural elements.

beam keyword . . .

information about beams. The following keywords apply.

apply applied distributed loads

connect nodal connectivity

elemtype beamSEL type

force nodal forces: components are displayed in terms of the beam local coordinate systems. These are the forces exerted by the nodes on the beamSEL.

group group to which beamSEL belongs

id beam ID

length length

local beamSEL local coordinate system

moment moment acting at ends of beamSEL

nforce <[**tr**anslation, **ro**tation]> <[**g**lobal, **l**ocal]>

generalized nodal forces (acting on the beamSEL). The translational or rotational components can be displayed in either the global or the local beamSEL system.

position centroid location of beamSEL

property keyword

The following keywords are available.

density density, ρ
emod Young's modulus, E
nu Poisson's ratio, ν
pmoment plastic moment, M^P
thexp thermal expansion coefficient, α_t
xcarea cross-sectional area, A
xciy 2^{nd} moment with respect to local y-axis, I_y
xciz 2^{nd} moment with respect to local z-axis, I_z
xcj polar moment of inertia, J
ydirection y-axis vector components

surfarea surface area along beam axis (assuming a circular cross-section)

volume volume

cable keyword . . .

information about cables. The following keywords apply.

connect nodal connectivity

elemtype cableSEL type

force nodal forces are displayed in terms of the cable local coordinate systems. These are the forces exerted by the nodes on the cableSEL.

group group to which cableSEL belongs

grout keyword

The following keywords are available.

confinement confining stress

displacement displacement

slip shear state

stress stress state

id cable ID

length length

local cableSEL local coordinate system

nforce <[**tr**anslation, **ro**tation]> <[**g**lobal, **l**ocal]>

generalized nodal forces (acting on the cableSEL). The translational or rotational components can be displayed in either the global or the local cableSEL system.

position centroid location of cableSEL

property keyword

The following keywords are available.

density	density, ρ
emod	Young's modulus, E
gr_coh	grout cohesive strength, c_g
gr_fric	grout friction angle, ϕ_g
gr_k	grout stiffness, k_g
gr_per	grout exposed perimeter, p_g
slide	sliding flag
slide_tol	sliding tolerance
thexp	thermal expansion coefficient, α_t
xcarea	cross-sectional area, A
ycomp	compressive yield strength, F_c
yten	tensile yield strength, F_t

stress axial stress in cableSEL

surfarea surface area along cable axis (length times grout exposed perimeter)

volume volume

yield indicator of whether or not the cableSEL itself has yielded

geogrid keyword <**ra**nge . . . >

Print properties and response of each geogridSEL in the range.

apply **pre**ssure

uniform applied pressure acting on geogridSEL

connect nodal connectivity

coupling keyword

confinement confining stress

displacement displacement magnitude in coupling springs

stress stress magnitude in coupling springs

yield yield state of coupling springs

elemtype geogridSEL type

group group of which geogridSEL is a part

id ID number of geogridSEL

local local coordinate system of geogridSEL

nforce <[**tr**anslation, **ro**tation]> <[**g**lobal, **l**ocal]>

generalized nodal forces (acting on the geogridSEL). The translational or rotational components can be displayed in either the global or the local geogridSEL system.

position centroid position

property keyword

The following keywords are available.

cs_scoh coupling spring cohesion (stress units)

cs_sfric coupling spring friction angle (degrees)

cs_sk coupling spring stiffness per unit area

density density (needed if dynamic mode or gravity is active)

			isotropic	isotropic material properties: E and v, where E is Young's modulus and v is Poisson's ratio
			orthotropic	orthotropic material properties: $\bar{e}_{11}, \bar{e}_{12}, \bar{e}_{22}, \bar{e}_{33}$
			slide	large-strain sliding flag
			slide_tol	large-strain sliding tolerance
			thexp	thermal expansion coefficient
			thickness	geogrid thickness
		surfarea	surface area	
		volume	volume (surface area times thickness)	
	info	general information relating to the structural-element logic		
	liner	keyword <**ra**nge. . . >		
		Print properties and responses of each linerSEL in the range.		
		apply	**pre**ssure	
			uniform applied pressure acting on the linerSEL	
		connect	nodal connectivity	
		coupling	keyword	
			displacement	**nor**mal, **s**hear
				displacement in normal or shear coupling springs
			stress	**nor**mal, **s**hear
				stress in normal or shear coupling springs
			yield	**s**hear
				yield state of shear coupling springs

elemtype linerSEL type

group group of which linerSEL is a part

id ID number of linerSEL

local local coordinate system of linerSEL

nforce <[**tr**anslation, **ro**tation]> <[**g**lobal, **l**ocal]>

generalized nodal forces (acting on the linerSEL). The translational or rotational components can be displayed in either the global or the local linerSEL system.

position centroid position

property keyword

The following keywords are available.

cs_ncut normal coupling spring tensile strength

cs_nk normal coupling spring stiffness per unit area

cs_scoh shear coupling spring cohesion (stress units)

cs_scohres shear coupling spring residual cohesion (stress units)

cs_sfric shear coupling spring friction angle (degrees)

cs_sk shear coupling spring stiffness per unit area

density density (needed if dynamic mode or gravity is active)

isotropic isotropic material properties: E and v, where E is Young's modulus and v is Poisson's ratio

orthotropic orthotropic material properties: $\bar{e}_{11}, \bar{e}_{12}, \bar{e}_{22}, \bar{e}_{33}$

slide large-strain sliding flag

slide_tol large-strain sliding tolerance

thexp thermal expansion coefficient

thickness liner thickness

surfarea surface area

volume volume (surface area times thickness)

link keyword . . . ...

information about links. The following keywords apply.

attach attachment conditions

constit keyword property

information associated with constitutive models for deformable attachment directions. The available keywords are as follows.

lindeform Property keywords for the linear model are as follows.

area area
disp displacement
force force
k stiffness
yield yield state

nydeform Property keywords for the normal-yield model are as follows.

area area
disp displacement
force force
gap gap information
k stiffness
ycomp compressive-yield force
yield yield state
ytens tensile-yield force

slide sliding flag

slide_tol sliding tolerance

source source node — node from which the link emanates

target link type: node-to-zone or node-to-node

node keyword. . .

information about structural-element nodes. The following keywords apply.

apply <**fo**rce> or <**mom**ent>

applied force: translational (default) or rotational components are displayed in terms of the global or nodal local coordinate systems.

disp <**rot** or **t**ran> <**gl**obal or **l**ocal>

displacement: translational (default) or rotational components are displayed in terms of the global or nodal local (default) coordinate systems.

fixity velocity fixity in terms of nodal local coordinate system

fob <**rot** or **t**ran> <**gl**obal or **l**ocal>

unbalanced force: translational (default) or rotational components are displayed in terms of the global or nodal local (default) coordinate systems.

ldamp local-damping factor

link link-present flag

local nodal local coordinate system

local_fix fixity state

mass <**rot** or **t**ran>

mass: translational (default) or rotational components are displayed in terms of the nodal local coordinate systems.

pos current position

current denotes current position of the node. This gets updated even in small-strain mode.

reference (default)

denotes the reference position of the node. (Position when **sel** is first created/moved before cycling.) Reference position gets updated only in large-strain mode.

stiffness <**rot** or **t**ran>

stiffness sum: translational (default) or rotational components are displayed in terms of the nodal local coordinate system.

vel <**rot** or **t**ran> <**gl**obal or **l**ocal>

velocity: translational (default) or rotational components are displayed in terms of the global or nodal local (default) coordinate systems.

pile keyword. . .

information about piles. The following keywords apply.

apply applied distributed loads

connect nodal connectivity

coupling keyword

The following keywords are available.

confinement confining stress

displacement keyword

normal normal displacement

shear shear displacement

stress keyword

normal normal stress

shear shear stress

yield keyword

normal normal yield state

shear shear yield state

elemtype pileSEL type

force nodal forces: translational components are displayed in terms of the pile local coordinate systems. These are the forces exerted by the nodes on the pileSEL.

group group to which pileSEL belongs

id pile ID

length length

local pileSEL local coordinate system

moment moment acting at end of pileSEL

nforce <[**tr**anslation, **ro**tation]> <[**g**lobal, **l**ocal]>

generalized nodal forces (acting on the pileSEL). The translational or rotational components can be displayed in either the global or the local pileSEL system.

position centroid location of pileSEL

property keyword

The following keywords are available.

cs_ncoh normal coupling spring cohesive strength, c_n

cs_nfric normal coupling spring friction angle, ϕ_n

cs_ngap normal coupling spring gap-use flag, g

cs_nk normal coupling spring stiffness, k_n

cs_scoh shear coupling spring cohesive strength, c_s

cs_sfric shear coupling spring friction angle, ϕ_s

cs_sk shear coupling spring stiffness, k_s

density density, ρ

emod Young's modulus, E

nu Poisson's ratio, ν

per exposed perimeter, p

pmoment plastic moment, M^P
slide sliding flag
slide_tol sliding tolerance
thexp thermal expansion coefficient, α_t
xcarea cross-sectional area, A
xciy 2^{nd} moment with respect to local y-axis, I_y
xciz 2^{nd} moment with respect to local z-axis, I_z
xcj polar moment of inertia, J
ydirection y-direction vector components

The following keywords apply if the rockbolt logic is active (**SEL pile property rockbolt on**).

cs_cfincr flag to activate incremental confining stress logic (default: **off**)
cs_cftable number of table relating effective confining stress factor to deviatoric stress
cs_sctable number of table relating cohesion of shear coupling spring to relative shear displacement
cs_sftable number of table relating friction angle of shear coupling spring to relative shear displacement
rockbolt flag to activate rockbolt logic (default: **off**)
tfstrain tensile failure strain (non-dimensional)
tyield axial tensile yield strength (force units), [F]

surfarea surface area along pile axis (length times exposed perimeter)

volume volume

recover keyword

Print stresses and stress resultants (as well as surface system) for shell-type SELs (geogrids, liners and shells). This command prints these quantities, which must be computed first using the stress-recovery procedure (see the **SEL recover** command).

The following keywords apply.

depth depth factor used during stress recovery

pstress principal stresses at specified depth

sres keyword

mx	stress resultant M_x
my	stress resultant M_y
mxy	stress resultant M_{xy}
nx	stress resultant N_x
ny	stress resultant N_y
nxy	stress resultant N_{xy}
qx	stress resultant Q_x
qy	stress resultant Q_y

stress keyword

xx	xx-stress component
yy	yy-stress component
zz	zz-stress component
xy	xy-stress component
yz	yz-stress component
zx	zx-stress component

surface surface coordinate system used during stress recovery

shell keyword. . .

information about shells. The following keywords apply.

apply **pre**ssure

applied pressure

connect nodal connectivity

elemtype shellSEL type

group group to which shellSEL belongs

id shell ID

local shellSEL local coordinate system

nforce <[**tr**anslation, **ro**tation]> <[**g**lobal, **l**ocal]>

generalized nodal forces (acting on the shellSEL). The translational or rotational components can be displayed in either the global or the local shellSEL system.

position centroid location of shellSEL

property keyword

The following keywords are available.

density	density, ρ
isotropic	isotropic properties
orthotropic	orthotropic properties
thexp	thermal expansion coefficient, α_t
thickness	thickness, t

surfarea surface area

volume volume

type type of structural element, including CID and ID number

table <***n***>

summary of existing tables, with number of items and maximum and minimum values. If table number ***n*** is given, table values are displayed.

tet <keyword>

tetrahedra data. An optional keyword can be specified to display selected tetrahedra data. The following keywords apply.

gp gridpoint connection

information general tetrahedra information is displayed: overlay-type numbers; tetrahedron volume; and centroid coordinates (This is the default display if no keyword is given.)

principal principal stresses for tetrahedra

stress stress components for tetrahedra

thermal thermal calculation-mode information

water water-table coordinates

zone <keyword>

zone data. Optional keywords can be specified to display selected zone data. The following keywords apply.

density zone mass density

extra ***i***

extra zone variable for extra array index ***i***

fdensity zone fluid density

fsi full strain-increment tensor

fsr full strain-rate tensor

gp ID number of gridpoint associated with zones

information general zone information: type number (0 = brick, 1 = wedge, 2 = pyramid), constitutive model, group, centroid coordinates (This is the default display if no keyword is given.)

join zone face (join) connections

pp zone pore pressure (average from gridpoint values)

principal <**eff**ective>
<**to**tal>

principal stress components for zones. The optional keyword **effective** or **total** can be given to display either effective or total principal stresses. The default is **total**.

property keyword

material properties assigned to zones. Values are displayed for the **property** keyword. The **PROPERTY** keywords available for each constitutive model are listed in the **PROPERTY** command section.

state <keyword>

plasticity state indicators. The meanings of the plasticity state indicators for each plasticity model are listed in the **PROPERTY** command section.

The following keywords apply.

any prints plasticity state for state set in any tetrahedral sub-zone (default)

average prints plasticity state if more than 50% of tetrahedral sub-zones are at this state

strain shear strain increment, shear strain rate, volumetric strain increment, volumetric strain rate

stress <**eff**ective>
<**to**tal>

stress components for zones. The optional keyword **effective** or **total** can be given to display either effective or total stresses. The default is total.

volume zone volume

FISH Variables

One or more *FISH* symbol names can be given as keywords; their values will be displayed. If the symbol name is that of a **DEFINE**d *FISH* function, then it will be executed (and all functions that it invokes will be executed) before its value will be displayed. If the symbol is a simple *FISH* variable, then its current value will be displayed.

PROPERTY keyword ***value*** <keyword . . . > <. . . > <**ra**nge . . . >

This command assigns properties for a constitutive model identified by the **MODEL** command. There are 10 basic constitutive models in *FLAC3D*: null, isotropic elastic, transversely isotropic elastic, orthotropic elastic, Mohr-Coulomb plasticity, Drucker-Prager plasticity, ubiquitous-joint, strain-hardening/softening, Cam-clay and bilinear strain-hardening/softening ubiquitous-joint. Except for the null model, all of these models require the specific properties described below. In addition, the fluid-flow model requires additional properties, also listed here. The optional creep and thermal models also require specific properties that are included under this command as well. Finally, the properties for the optional dynamic pore-pressure generation model, the Finn model, are listed. The ten basic models are described in Section 2 in **Theory and Background**, the fluid-flow model in Section 1 in **Fluid-Mechanical Interaction**, and the optional models for creep and thermal in Sections 2 and Section 1 in **Optional Features**, respectively. The dynamic pore-pressure generation model is described in Section 3 in **Optional Features**.

The **PROPERTY** keywords for each model type are listed separately. The model type must be defined prior to displaying or plotting properties. If properties that are not consistent with the chosen model are given, a warning message that informs the user that the unneeded properties were not accepted will be given. If a required property is *not* specified, the default will be used. Property values are zero, by default, unless otherwise specified.

Three optional keyword phrases are available to modify properties. The phrase must be given immediately *following* the property value. The available keyword phrases are:

gauss_dev ***s***

A Gaussian distribution is used to assign the property values randomly, with a mean of ***value*** and standard deviation of ***s***.

gradient ***gx gy gz***

Property values are assigned with a gradient defined by

$$value\_modified = value + gx \times x + gy \times y + gz \times z$$

in which ***x, y, z*** is the position vector.

uniform_dev ***s***

A uniform distribution is used to assign the property values randomly, with a mean value of ***value*** and a standard deviation of ***s***.

NOTE: Isotropic elastic behavior is defined in most of the models by means of the elastic bulk modulus, K, and the shear modulus, G. This is the recommended approach based on the reasons stated in Section 3.8 in the **User's Guide** (note 13). Alternatively, for convenience, the elastic behavior can be prescribed via the keywords **young** for Young's modulus, E, and **poisson** for Poisson's ratio, ν.

Elastic Mechanical Models

Isotropic Elastic — **MODEL elastic**

(1) **bu**lk elastic bulk modulus, *K*
(2) **sh**ear elastic shear modulus, *G*

See Section 2.4.1 in **Theory and Background** for details.

Transversely Isotropic Elastic — **MODEL anisotropic**

(1)	**dd**	dip direction of plane of isotropy
(2)	**dip**	dip angle of plane of isotropy
(3)	**e1**	Young's modulus in the plane of isotropy
(4)	**e3**	Young's modulus normal to the plane of isotropy
(5)	**g13**	shear modulus for any plane normal to the plane of isotropy
(6)	**nu12**	Poisson's ratio characterizing lateral contraction in the plane of isotropy when tension is applied in the plane
(7)	**nu13**	Poisson's ratio characterizing lateral contraction in the plane of isotropy when tension is applied normal to the plane

See Section 2.4.3 in **Theory and Background** for details.

Orthotropic Elastic — **MODEL orthotropic**

(1) **dd**	dip direction of plane defined by axes 1′-2′
(2) **dip**	dip angle of plane defined by axes 1′-2′
(3) **e1**	Young's modulus in direction 1′
(4) **e2**	Young's modulus in direction 2′
(5) **e3**	Young's modulus in direction 3′
(6) **g12**	shear modulus in planes parallel to axes 1′-2′
(7) **g13**	shear modulus in planes parallel to axes 1′-3′
(8) **g23**	shear modulus in planes parallel to axes 2′-3′
(9) **nu12**	Poisson's ratio characterizing lateral contraction in direction 1′ when tension is applied in direction 2′
(10) **nu13**	Poisson's ratio characterizing lateral contraction in direction 1′ when tension is applied in direction 3′
(11) **nu23**	Poisson's ratio characterizing lateral contraction in direction 2′ when tension is applied in direction 3′
(12) **nx**	x-component of unit normal to plane defined by axes 1′-2′
(13) **ny**	y-component of unit normal to plane defined by axes 1′-2′
(14) **nz**	z-component of unit normal tp plane defined by axes 1′-2′
(15) **rot**	rotation angle between the 1′ axis and the dip-direction vector, defined positive clockwise from the dip-direction vector

See Section 2.4.2 in **Theory and Background** for details.

Elastic-Plastic Mechanical Models

Drucker-Prager — **MODEL drucker**

(1)	**bu**lk	elastic bulk modulus, K
(2)	**ks**hear	material parameter, k_ϕ
(3)	**qd**il	material parameter, q_ψ
(4)	**qv**ol	material parameter, q_ϕ
(5)	**sh**ear	elastic shear modulus, G
(6)	**ten**sion	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 2.5.1 in **Theory and Background** for details.

Note that the default tension limit is zero for a material with $q_\phi = 0$, and k_ϕ / q_ϕ otherwise. The value assigned for the tension limit remains constant when tensile failure occurs.

Hoek-Brown — **MODEL hoekbrown**

(1)	**at**able	number of table relating a to e_3^p
(2)	**b**ulk	bulk modulus, K
(3)	**cit**able	number of table relating σ_{ci} to e_3^p
(4)	**hba**	Hoek-Brown parameter, a
(5)	**hbs**	Hoek-Brown parameter, s
(6)	**hbmb**	Hoek-Brown parameter, m_b
(7)	**hbsigci**	Hoek-Brown parameter, σ_{ci}
(8)	**hbs3cv**	Hoek-Brown parameter, σ_3^{cv}
(9)	**hb_e3plas**	accumulated plastic strain, e_3^p
(10)	**hb_ind**	plasticity indicator (as Mohr Coulomb)
(11)	**mt**able	number of table relating to m_b to e_3^p
(12)	**mult**able	number of table relating a multiplier to σ_3
(13)	**sh**ear	shear modulus, G
(14)	**st**able	number of table relating s to e_3^p

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 shear in the past

See Section 2.5.8 in **Theory and Background** for details.

Mohr-Coulomb — **MODEL mohr**

(1)	**bu**lk	elastic bulk modulus, K
(2)	**c**ohesion	cohesion, c
(3)	**di**lation	dilation angle, ψ
(4)	**fric**tion	internal angle of friction, ϕ
(5)	**sh**ear	elastic shear modulus, G
(6)	**ten**sion	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 2.5.2 in **Theory and Background** for details.

Note that the default tension limit is zero for a material with no friction, and $c/tan\phi$ otherwise. The value assigned for the tension limit remains constant when tensile failure occurs.

Ubiquitous-Joint — **MODEL ubiquitous**

(1)	**bu**lk	elastic bulk modulus, K
(2)	**c**ohesion	cohesion of matrix, c
(3)	**di**lation	dilation angle of matrix, ψ
(4)	**fric**tion	internal angle of friction of matrix, ϕ
(5)	**jc**ohesion	joint cohesion, c_j
(6)	**jdd**irection	dip direction of weakness plane
(7)	**jdil**ation	joint dilation angle, ψ_j
(8)	**jdip**	dip angle of weakness plane
(9)	**jf**riction	joint friction angle, ϕ_j
(10)	**jnx**	x-component of unit normal to weakness plane
(11)	**jny**	y-component of unit normal to weakness plane
(12)	**jnz**	z-component of unit normal to weakness plane
(13)	**jt**ension	joint tension limit, σ_j^t
(14)	**sh**ear	elastic shear modulus, G
(15)	**ten**sion	tension limit of matrix, σ^t

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 2.5.3 in **Theory and Background** 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 $c_j / tan\phi_j$ 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 — **MODEL ssoftening**

(1)	**bu**lk	elastic bulk modulus, K
(2)	**c**ohesion	cohesion, c
(3)	**ct**able	number of table relating cohesion to plastic shear strain
(4)	**di**lation	dilation angle, ψ
(5)	**dt**able	number of table relating dilation angle to plastic shear strain
(6)	**fric**tion	angle of internal friction, ϕ
(7)	**ft**able	number of table relating friction angle to plastic shear strain
(8)	**sh**ear	elastic shear modulus, G
(9)	**ten**sion	tension limit, σ^t
(10)	**tt**able	number of table relating tension limit to plastic tensile strain

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 **cohesion**, **dilation**, **friction** or **tension** keywords).

The following calculated properties can be printed, plotted or accessed via *FISH*.

(1)	**es**_plastic	plastic shear strain
(2)	**et**_plastic	plastic tensile strain

See Section 2.5.4 in **Theory and Background** for details.

Bilinear, Strain-Hardening/Softening Ubiquitous-Joint — **MODEL subiquitous**

(1) **bij**oint = 0 for joint linear model (default);
= 1 for joint bilinear model

(2) **bim**atrix = 0 for matrix linear model (default);
= 1 for matrix bilinear model

(3) **bu**lk elastic bulk modulus, K

(4) **c2**table number of table relating matrix cohesion c_2 to matrix plastic shear strain

(5) **cj**table number of table relating joint cohesion c_{j1} to joint plastic shear strain

(6) **cj2**table number of table relating joint cohesion c_{j2} to joint plastic shear strain

(7) **c**ohesion matrix cohesion, c_1

(8) **co2** matrix cohesion, c_2

(9) **ct**able number of table relating matrix cohesion c_1 to matrix plastic shear strain

(10) **d2**table number of table relating matrix dilation ψ_2 to matrix plastic shear strain

(11) **di2** matrix dilation angle, ψ_2

(12) **di**lation matrix dilation angle, ψ_1

(13) **dj**table number of table relating joint dilation ψ_{j1} to joint plastic shear strain

(14) **dj2**table number of table relating joint dilation ψ_{j2} to joint plastic shear strain

(15) **dt**able number of table relating matrix dilation angle ψ_1 to matrix plastic shear strain

(16) **f2**table number of table relating matrix friction angle ϕ_2 to matrix plastic shear strain

(17) **fj**table number of table relating joint friction angle ϕ_{j1} to joint plastic shear strain

(18) **fj2**table number of table relating joint friction angle ϕ_{j2} to joint plastic shear strain

(19) **fr2** matrix friction angle, ϕ_2

(20) **fric**tion matrix friction angle, ϕ_1

(21) **ft**able number of table relating matrix friction ϕ_1 angle to matrix plastic shear strain

(22) **jc2** joint cohesion, c_{j2}

(23) **jc**ohesion joint cohesion, c_{j1}

(24) **jdd**irection dip direction of weakness plane

(25) **jdil**ation	joint dilation angle, ψ_{j1}
(26) **jdip**	dip angle of weakness plane
(27) **jd2**	joint dilation angle, ψ_{j2}
(28) **jf**riction	joint friction angle, ϕ_{j1}
(29) **jf2**	joint friction angle, ϕ_{j2}
(30) **jnx**	x-component of unit normal to weakness plane
(31) **jny**	y-component of unit normal to weakness plane
(32) **jnz**	z-component of unit normal to weakness plane
(33) **jt**ension	joint tension limit, σ_j^t
(34) **sh**ear	elastic shear modulus, G
(35) **ten**sion	matrix tension limit, σ^t
(36) **tj**table	number of table relating joint tension limit σ_j^t to joint plastic tensile strain
(37) **tt**able	number of table relating matrix tension limit σ^t to matrix plastic tensile strain

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

The following calculated properties can be printed, plotted or accessed via *FISH*.

(1) **es**_plastic	plastic shear strain
(2) **et**_plastic	plastic tensile strain
(3) **etj**_plastic	joint plastic tensile strain
(4) **esj**_plastic	joint plastic shear strain

See Section 2.5.5 in **Theory and Background** 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 1.8 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
	jddirection		dip direction of weakness plane
	jdip		dip angle of weakness plane
	jnx		x-comp. of unit normal to weakness plane
	jny		y-comp. of unit normal to weakness plane
	jnz		z-comp. of unit normal to weakness plane
	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 —**MODEL doubleyield**

(1)	**b**ulk	maximum elastic bulk modulus, K
(2)	**cap_p**ressure	current intersection of volumetric yield surface (cap) with pressure (mean stress) axis, p_c
(3)	**c**ohesion	cohesion, c
(4)	**cpt**able	number of table relating cap pressure to plastic volume strain
(5)	**ct**able	number of table relating cohesion to plastic shear strain
(6)	**di**lation	dilation angle, ψ
(7)	**dt**able	number of table relating dilation angle to plastic shear strain
(8)	**ev**_plastic	accumulated plastic volumetric strain
(9)	**f**riction	angle of internal friction, ϕ
(10)	**ft**able	number of table relating friction angle to plastic shear strain
(11)	**mu**ltiplier	multiplier on current plastic cap modulus to give elastic bulk and shear moduli, R
(12)	**s**hear	maximum elastic shear modulus, G
(13)	**t**ension	tension limit, σ^t
(14)	**tt**able	number of table relating tensile limit to plastic tensile strain

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 **cohesion**, **dilation**, **friction**, **tension** or **cap_pressure** keywords).

The following calculated properties can be printed, plotted or accessed via *FISH*.

(15) **es**_plastic accumulated plastic shear strain
(16) **et**_plastic accumulated plastic tensile strain
(17) **ev**_plastic accumulated plastic volumetric strain

See Section 2.5.6 in **Theory and Background** for details.

Modified Cam-clay — **MODEL cam-clay**

(1)	**bulk_b**ound	maximum elastic bulk modulus, K_{max}
(2)	**cv**	initial specific volume, v_0 (by default, calculated internally)
(3)	**ka**ppa	slope of elastic swelling line, κ
(4)	**l**ambda	slope of normal consolidation line, λ
(5)	**mm**	frictional constant, M
(6)	**mpc**	preconsolidation pressure, p_{c0}
(7)	**mp1**	reference pressure, p_1
(8)	**mv_l**	specific volume at reference pressure, p_1, on normal consolidation line, v_λ
(9)	**p**oisson	Poisson's ratio, ν
(10)	**sh**ear	elastic shear modulus, G

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 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 on Poisson's ratio is specified, then the shear modulus remains constant; Poisson's ratio will change as the bulk modulus changes.

The following calculated properties can be printed, plotted or accessed via *FISH*.

(1)	**bu**lk	elastic bulk modulus, K
(2)	**cam_cp**	current mean effective stress
(3)	**cam_ev**	accumulated total volumetric strain
(4)	**camev_p**	accumulated plastic volumetric strain
(5)	**cq**	current mean deviatoric stress

See Section 2.5.7 in **Theory and Background** for details.

Finn (dynamic pore-pressure generation) — **MODEL finn**

(1)	**bu**lk	elastic bulk modulus, K
(2)	**c**ohesion	cohesion, c
(3)	**ct**able	number of table relating cohesion to plastic shear strain
(4)	**di**lation	dilation angle, ψ
(5)	**dt**able	number of table relating dilation angle to plastic shear strain
(6)	**ff_c1**	constant, C_1
(7)	**ff_c2**	constant, C_2
(8)	**ff_c3**	constant, C_3
(9)	**ff_c4**	constant, C_4
(10)	**ff_l**atency	minimum number of timesteps between reversals
(11)	**ff_s**witch	= 0 for Martin et al. (1995) formula, and 1 for Byrne (1991) formula
(12)	**fric**tion	angle of internal friction, ϕ
(13)	**ft**able	number of table relating friction angle to plastic shear strain
(14)	**sh**ear	elastic shear modulus, G
(15)	**ten**sion	tension limit, σ^t
(16)	**tt**able	number of table relating tension limit to plastic tensile strain

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)	**es**_plastic	plastic shear strain
(2)	**et**_plastic	plastic tensile strain
(3)	**ff_c**ount	number of shear strain reversals detected
(4)	**ff_e**vd	internal volume strain, ϵ_{vd}, of Eq. (3.59) in **Optional Features**

See Section 3.4.4 in **Optional Features** for details.

Creep Models

Classical Viscoelastic (Maxwell Substance) — **MODEL viscous**

(1) **bu**lk elastic bulk modulus, K
(2) **sh**ear elastic shear modulus, G
(3) **vis**cosity dynamic viscosity, η

See Section 2.2.1 in **Optional Features** for details.

Burger's Model — **MODEL burger**

(1)	**bu**lk	elastic bulk modulus, K
(2)	**ksh**ear	Kelvin shear modulus, G^K
(3)	**kvis**cosity	Kelvin viscosity, n^K
(4)	**msh**ear	Maxwell shear modulus, G^M
(5)	**mvis**cosity	Maxwell viscosity, n^M

See Section 2.2.2 in **Optional Features** for details.

Power Law — **MODEL power**

(1)	**a_1**	power-law constant, A_1
(2)	**a_2**	power-law constant, A_2
(3)	**bu**lk	elastic bulk modulus, K
(4)	**n_1**	power-law exponent, n_1
(5)	**n_2**	power-law exponent, n_2
(6)	**rs_1**	reference stress, σ_1^{ref}
(7)	**rs_2**	reference stress, σ_2^{ref}
(8)	**sh**ear	elastic shear modulus, G

See Section 2.2.3 in **Optional Features** for details.

WIPP Model — **MODEL wipp**

(1) **act**_energy activation energy, Q
(2) **a**_**w**ipp WIPP model constant, A
(3) **b**_**w**ipp WIPP model constant, B
(4) **bu**lk elastic bulk modulus, K
(5) **d**_**w**ipp WIPP model constant, D
(6) **e**_**dot**_star critical steady-state creep rate, $\dot{\epsilon}_{ss}^{*}$
(7) **gas**_c gas constant, R
(8) **n**_**w**ipp WIPP model exponent, n
(9) **sh**ear elastic shear modulus, G
(10) **t**emp zone temperature, T

The following calculated properties can be printed, plotted or accessed via *FISH*.

(11) **e**_**prime** accumulated primary creep strain
(12) **e**_**rate** accumulated primary creep strain rate

See Section 2.2.4 in **Optional Features** for details.

Burger-Creep Viscoplastic Model — **MODEL cvisc**

(1)	**b**ulk	elastic bulk modulus, K
(2)	**c**ohesion	cohesion, c
(3)	**d**ensity	mass density, ρ
(4)	**di**lation	dilation angle, ψ
(5)	**f**riction	angle of internal friction, ϕ
(6)	**ksh**ear	Kelvin shear modulus, G^K
(7)	**kvis**cosity	Kelvin viscosity, η^K
(8)	**msh**ear	elastic shear modulus, G^M
(9)	**t**ension	tension limit σ^t
(10)	**mvis**cosity	Maxwell dynamic viscosity, η^M

The following calculated properties can be printed, plotted or accessed via *FISH*.

(11)	**es**_plastic	accumulated plastic shear strain
(12)	**et**_plastic	accumulated plastic tensile strain

See Section 2.2.5 in **Fluid-Mechanical Interaction** for details.

Power Law Viscoplastic Model — **MODEL cpow**

(1)	**a_1**	power-law constant, A_1
(2)	**a_2**	power-law constant, A_2
(3)	**bu**lk	elastic bulk modulus, K
(4)	**c**ohesion	cohesion, c
(5)	**dil**ation	dilation angle, ψ
(6)	**f**riction	angle of internal friction, ϕ
(7)	**n_1**	power-law exponent, n_1
(8)	**n_2**	power-law exponent, n_2
(9)	**rs_1**	reference stress, σ_1^{ref}
(10)	**rs_2**	reference stress, σ_2^{ref}
(11)	**sh**ear	elastic shear modulus, G
(12)	**t**ension	tension limit, σ^t

See Section 2.2.6 in **Optional Features** for details.

WIPP-Creep Viscoplastic Model — **MODEL pwipp**

(1) **act**_energy activation energy, Q
(2) **a**_**w**ipp WIPP model constant, A
(3) **b**_**w**ipp WIPP model constant, B
(4) **bu**lk elastic bulk modulus, K
(5) **d**_**w**ipp WIPP model constant, D
(6) **e**_**dot**_star critical steady-state creep rate, $\dot{\epsilon}^{*}_{ss}$
(7) **gas**_c gas constant, R
(8) **ks**hear material parameter, k_{ϕ}
(9) **n**_**w**ipp WIPP model exponent, n
(10) **qd**il material parameter, q_k
(11) **qv**ol material parameter, q_{ϕ}
(12) **sh**ear elastic shear modulus, G
(13) **t**emp zone temperature, T
(14) **ten**sion 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**_**p**rime primary creep strain
(2) **e**_**r**ate primary creep rate
(3) **es**_plastic accumulated plastic shear strain
(4) **et**_plastic accumulated plastic tensile strain

See Section 2.2.7 in **Fluid-Mechanical Interaction** for details.

Crushed Salt Model — **MODEL cwipp**

(1) **act**_energy activation energy, Q
(2) **a**_**w**ipp WIPP model constant, A
(3) **b**_**f** final, intact salt, bulk modulus, K_f
(4) **b**_**w**ipp 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) **bu**lk elastic bulk modulus, K
(9) **d**_**f** final, intact salt, density, ρ_f
(10) **d**_**w**ipp WIPP model constant, D
(11) **e**_**dot**_star critical steady-state creep rate, $\dot{\epsilon}^*_{ss}$
(12) **gas**_c gas constant, R
(13) **n**_**w**ipp WIPP model exponent, n
(14) **rh**o density, ρ
(15) **s**_**f** final, intact salt, shear modulus, G_f
(16) **sh**ear elastic shear modulus, G
(17) **t**emp 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

Fluid Flow

Isotropic Fluid Flow — **MODEL fl_isotropic**

(1)	**per**meability	isotropic permeability, k
(2)	**poro**sity	porosity, n (n = 0.5, by default)

Anisotropic Fluid Flow* — **MODEL fl_anisotropic**

(1)	**fdd**	dip direction of $k1 - k2$ plane
(2)	**fdip**	dip angle of $k1 - k2$ plane
(3)	**frot**	rotation angle between $k1$-axis and dip vector
(4)	**h1**	permeability in $k1$-direction
(5)	**h2**	permeability in $k2$-direction
(6)	**h3**	permeability in $k3$-direction

Mechanical-Fluid Flow Coupling

(1)	**biot_c**	Biot coefficient (grain compressibility), α (α = 1, by default)

Thermal-Fluid Flow Coupling

(1)	**u_thc**	undrained thermal coefficient, β

The permeability used in *FLAC3D* is defined as the conventional hydraulic conductivity (units: [L/T] — e.g., m/sec) divided by the unit weight of water (units: [F/L^3] — e.g., Pa/m). Permeability must be nonzero if **SET flow on** applies.

* See Figure 1.1 in **Fluid-Mechanical Interaction** for definitions of the principal directions of the permeability tensor.

Thermal

Isotropic Heat Conduction — **MODEL th_isotropic**

(1) **cond**uctivity isotropic thermal conductivity, K
(2) **sp**ec_heat specific heat, C_v

Thermal-Mechanical Coupling

(1) **thex**p coefficient of linear thermal expansion, α_t

Mass density must also be initialized for the model grid using the command **INITIAL density**.

Isotropic Advection-Conduction — **MODEL th_ac**

(1) **sp**ec_heat specific heat, C_p
(2) **thex**p coefficient of linear thermal expansion, α_t
(3) **tdd** principal thermal conductivity plane dip direction angle
(4) **tdip** principal thermal conductivity plane dip angle
(5) **trot** principal thermal conductivity plane rotation angle
(6) **tk1** principal thermal conductivity value, $tk1$
(7) **tk2** principal thermal conductivity value, $tk2$
(8) **tk3** principal thermal conductivity value, $tk3$
(9) **tkxx** xx-component of thermal conductivity matrix
(10) **tkyy** yy-component of thermal conductivity matrix
(11) **tkzz** zz-component of thermal conductivity matrix
(12) **tkxy** xy-component of thermal conductivity matrix
(13) **tkxz** xz-component of thermal conductivity matrix
(14) **tkyz** yz-component of thermal conductivity matrix

Anisotropic Heat Conduction — **MODEL th_anisotropic**

(1) **cond**uctivity matrix thermal conductivity, k_s^T
(2) **econd**uct effective conductivity, k^T
(3) **esp**ec_heat effective specific heat over matrix density, C^T/ρ
(4) **f_qx** x-component of specific discharge
(5) **f_qy** y-component of specific discharge
(6) **f_qz** z-component of specific discharge
(7) **f_rho** temperature-dependent fluid density, ρ_w
(8) **f_t**hexp fluid volumetric thermal expansion coefficient, β_f
(9) **f_t0** reference temperature, T_0

(10) **lcond**uct	fluid thermal conductivity, k_w^T
(11) **lsp**ec_heat	fluid specific heat, c_w
(12) **sp**ec_heat	matrix specific heat, c_s
(13) **thex**p	matrix linear thermal expansion coefficient, α

QUIT stops execution of *FLAC3D* (a synonym of **STOP**).

RANGE **na**me rangename keyword . . . <**not**> <**any**>

A range can be created directly with the **RANGE** command. The range is a volume of the model (either connected or disconnected) that is defined by one or more range elements. The range must first be named by specifying a rangename following the **name** keyword (see below). The following keyword phrases can be used to define range elements.

annulus **c**enter ***xc yc zc*** **r**adius ***r1 r2***

(***xc, yc, zc***) is the center of a spherical region; the range is between radii ***r1*** and ***r2***.

cid ***imin imax***

beginning and ending component identification numbers for structural element components (see Section 1 in **Structural Elements**)

cylinder **end1** ***x1 y1 z1*** **end2** ***x2 y2 z2*** **r**adius ***r***

cylindrical range with one end of the cylinder axis (**end1**) at location (***x1, y1, z1***), the other end (**end2**) at location (***x2, y2, z2***), and with a cylinder radius of ***r***

direction ***v1, v2, v3*** <**ang**le ***value***>

zone surface with outward normal defined by the vector (***v1, v2, v3***). By default, the tolerance is 90 degrees. This can be changed with the optional **angle** keyword followed by the tolerance angle ***value***.

group name

group name, identified by the **GROUP** command, of zones or structural elements

id ***il iu***

beginning and ending identification numbers for zones, gridpoints, structural elements, interface elements or nodes

model keyword

zones with corresponding constitutive model (see the **MODEL** command for keyword names)

name rangename

The created range element will be stored in a list under the specified range name. The name can then be used directly following the **range** keyword.

name Once a range has been named, the name can be specified as a range element in another range.

plane keyword

range above or below a specified plane. The plane is defined by the keywords:

above half-space above the plane (default)

below half-space below the plane

dd ***dd***

dip direction angle, measured in the global xy-plane clockwise from the positive y-axis

dip ***dip***

dip angle, measured in the negative z-direction from the global xy-plane

distance ***d***

points within distance ***d*** of the plane

normal ***xn yn zn***

unit normal vector ***xn, yn, zn*** of the plane

origin ***x y z***

one point on the plane at location (***x, y, z***)

The location of the plane is defined by **origin** and either **dd** and **dip** or **normal**.

sphere **c**enter ***x y z*** **r**adius ***r***

spherical range with centroid at location (***x, y, z***) and with radius ***r***

volume ***n***

volume range defined by the volume ***n***, created with the **GENERATE surface** command

x ***xl xu***

lower and upper limits for the x-coordinate

y ***yl yu***

lower and upper limits for the *y*-coordinate

z ***zl zu***

lower and upper limits for the *z*-coordinate

If multiple range keyword phrases are given for the same **RANGE** command, the range is the intersection of the separate range elements.

Two additional keywords are available to modify the definition of a range element.

any If **any** is given after a range element, any object that falls within the range element is considered part of the range regardless of other range elements that define the range.

not If **not** is given after a range element, the meaning of the element is changed to signify the set *not* inside that range element.

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 filename. If no extension is specified, ".SAV" is assumed. If no filename is given, the default file "FLAC3D.SAV" will be restored.

The following files are unaffected by the **RESTORE** command:

log file	(see **SET log**)
plot file	(see **SET output**)

These files remain open, if open already, and their filenames are not changed when a **RESTORE** command is given. New filenames can be specified after the **RESTORE** command if required.

Also, the echo mode (**SET echo**) is unaffected by **RESTORE**. This mode can be turned on or off as needed.

All other conditions and values are taken from the **RESTORE**d save file. For example, *FISH* functions and variables and histories are restored from the save file; existing functions, variables and histories are lost.

RETURN This command should be the last in a remote input file (see the **CALL** command). It returns command control to the previous input source — for example, another file or to interactive keyboard mode. By default, **RETURN** is assumed if not given at the end of an input file.

SAVE <filename>

All information required to restart the problem is saved to the named file, filename. If no extension is specified, the default extension ".SAV" is used. The default filename is "FLAC3D.SAV" if no filename is given.

Note that all currently defined *FISH* functions and variables are saved on the file.

SEL keyword <keyword> ***value***

The commands and keywords governing the structural elements in *FLAC3D* are shown in Table 1.9.

Table 1.9 Major keywords for the* SEL *command

beam	**apply**
	begin . . . end . . .
	id
	nseg
	property
beamsel	**cid**
	id
	nodes
cable	**begin . . . end . . .**
	id
	nseg
	pretension
	property
cablesel	**cid**
	id
	nodes
delete	**beam**
	cable
	geogrid
	liner
	link
	node
	pile
	sel
	shell
geogrid	**apply**
	crossdiag
	elemtype
	group
	id
	property
	range

The structural elements are described in detail in Section 1 in **Structural Elements**.

Table 1.9 Major keywords for the **SEL** *command (cont.)*

geogridsel	**cid**
	elemtype
	id
	nodes
group	**range**
liner	**apply**
	crossdiag
	elemtype
	group
	id
	property
	range
linersel	**cid**
	elemtype
	id
	nodes
link	**attach**
	constit
	id
	net
	range
	target
node	**apply**
	fix
	free
	id
	init
	ldamp
	local
pile	**apply**
	begin . . . end . . .
	id
	nseg
	property

The structural elements are described in detail in Section 1 in **Structural Elements**.

Table 1.9 Major keywords for the **SEL** *command (cont.)*

pilesel	**cid**
	id
	nodes
recover	**sres**
	stress
	surface
set	**damp**
	liner
	link
	safety_fac
	scale_rmass
	v20ndcmd
shell	**apply**
	crossdiag
	elemtype
	group
	id
	property
	range
shellsel	**cid**
	elemtype
	id
	nodes

The structural elements are described in detail in Section 1 in **Structural Elements**.

Beam and BeamSEL

This section describes the **SEL beam** and **SEL beamsel** commands. The **SEL beam** command allows one to create a collection of beamSELs that lie between a beginning and ending point and specify properties of beamSELs, while the **SEL beamsel** command allows one to create individual beamSELs.

There are two forms of the **SEL beam** command: one form for the creation of the beam; and the other for specifying beam properties and applied loads.

Command Form 1: Beam Creation

beam <**id** ***id***> **b**egin ***x,y,z*** **end** ***x,y,z*** <**n**seg ***nseg***>

creates a beam. The beam consists of a collection of ***nseg*** (default = 1) beamSELs that lie along a straight line between the **begin** and **end** locations. New nodes associated with the beam will also be created (see the **id** keyword). The nodal connectivity of each new beamSEL will be ordered such that the direction from end-1 to end-2 corresponds with the direction from the **begin** point to the **end** point. The beam will be attached to the grid at its nodes such that the translational degrees-of-freedom are rigidly connected to the grid and the rotational degrees-of-freedom are free. If no attachment to the grid is desired, then, after creating and positioning the beam, the links may be deleted with the **SEL delete link** command. The following keyword is available.

id ***id***

specifies the beam ID that will be assigned to all new beamSELs created by this command. This keyword must immediately follow the **beam** keyword. If this value is not given, then the next available ID will be used. The ID controls potential joining of the two beam end-nodes to an existing beam as follows. At each end-node location, a new node is created unless all of the following conditions are fulfilled: (1) the ID is given; (2) an existing node is found "near" to the potential new node location; and (3) the found node is part of a beam with ID equal to ***id***.

Command Form 2: Specifying Beam Properties

beam <**id** ***id***> keyword <**ra**nge>

apply **y**dist ***yd*** **z**dist ***zd***

assigns uniform applied distributed loads (force per unit length) to all beamSELs in the range and with an ID number of ***id***; if ***id*** is not given, then all beamSELs in the range are considered. Positive loads act in the positive y- or z-directions of the beamSEL system and maintain this orientation during large-strain motion. Point loads are applied at the nodes using the **SEL node apply** command.

property keyword ***value*** <keyword ***value***> . . .

assigns the specified property to all beamSELs in the range and with an ID number of ***id***; if ***id*** is not given, then all beamSELs in the range are considered. The following properties are available.

density density (needed if dynamic mode or gravity are active)

emod Young's modulus

nu Poisson's ratio

pmoment plastic moment capacity

thexp thermal expansion coefficient

xcarea cross-sectional area

xciy second moment with respect to beamSEL y-axis

xciz second moment with respect to beamSEL z-axis

xcj polar moment of inertia

ydirection vector (***Yx, Yy, Yz***) whose projection onto the beamSEL cross-section defines the y-axis of the beamSEL system

beamsel <**cid** *cid*> <**id** *id*> **no**des *nd1 nd2*

creates a beamSEL given an existing set of nodes. (To create a collection of beamSELs that lie between a beginning and ending point, or to assign properties to beamSELs, use the **SEL beam** command.) The new beamSEL will not be attached to the grid; if you wish to attach it to the grid, then, after creating it, position its nodes with the **SEL node init** {**x,y,z**}**pos** command (which will create links and set appropriate attachment conditions for all nodes that are moved into a zone). The following keywords are available.

cid ***cid***

specifies the component-ID number of the beamSEL. Each SEL has a unique component-ID number. If this value is not given, then the new beamSEL is assigned the next available component-ID number.

id ***id***

specifies the ID number of the beamSEL. All SELs that comprise a particular entity will possess the same ID number — e.g., all beamSELs that comprise beam 6 will have an ID of 6. If this value is not given, then the new beamSEL is assigned an ID number of 0.

nodes ***nd1 nd2***

specifies the ID numbers (***nd1, nd2***) of two nodes that will define the beamSEL. These nodes must already exist — nodes can be created with the **SEL node** command. Ordering of the nodes defines the beamSEL coordinate system as follows. The positive x-direction lies along the line from ***nd1*** to ***nd2***, and the y-direction is found by projecting the global y- or x-directions onto the beamSEL cross-section. The y-direction can also be modified by the **SEL beam prop ydir** command.

Cable and CableSEL

This section describes the **SEL cable** and **SEL cablesel** commands. The **SEL cable** command allows one to create a collection of cableSELs that lie between a beginning and ending point and specify properties of cableSELs, while the **SEL cablesel** command allows one to create individual cableSELs.

There are two forms of the **SEL cable** command: one form for the creation of the cable; and the other for specifying cable properties.

Command Form 1: Cable Creation

cable <**id** ***id***> **b**egin ***x,y,z*** **end** ***x,y,z*** <**n**seg ***nseg***>

creates a cable. The cable consists of a group of ***nseg*** (default = 1) cableSELs that lie along a straight line between the **begin** and **end** locations. New nodes associated with the cable will also be created (see the ***id*** keyword). The nodal connectivity of each new cableSEL will be ordered such that the direction from end-1 to end-2 corresponds with the direction from the **begin** point to the **end** point. The cable will be attached to the grid at its nodes with link attachment conditions appropriate to a cable — see Table 1.29 in **Structural Elements**. The following keyword is available.

id ***id***

specifies the cable ID that will be assigned to all new cableSELs created by this command. This keyword must immediately follow the **cable** keyword. If this value is not given, then the next available ID will be used. The ID controls potential joining of the two cable end-nodes to an existing cable as follows. At each end-node location, a new node is created unless all of the following conditions are fulfilled: (1) the ID is given; (2) and existing node is found "near" to the potential new node location; and (3) the found node is part of a cable with ID equal to ***id***.

Command Form 2: Specifying Cable Properties

cable <**id** ***id***> keyword <**ra**nge>

pretension ***value***

applies given pre-tension force to all cableSELs in the range and with an ID number of ***id***; if ***id*** is not given, then all cableSELs in the range are considered. A positive pre-tension force places a cableSEL into tension. The given force is added to the current force being carried by each cableSEL.

property keyword ***value*** <keyword ***value***> . . .

assigns the specified property to all cableSELs in the range and with an ID number of ***id***; if ***id*** is not given, then all cableSELs in the range are considered. The following properties are available.

density density (needed if dynamic mode or gravity is active)

emod Young's modulus

gr_coh grout cohesive strength (force) per unit length

gr_fric grout friction angle (degrees)

gr_k grout stiffness per unit length

gr_per grout exposed perimeter

slide large-strain sliding flag

slide_tol large-strain sliding tolerance

thexp thermal expansion coefficient

xcarea cross-sectional area

ycomp compressive yield strength (force)

ytens tensile yield strength (force)

cablesel <**cid** *cid*> <**id** *id*> **no**des *nd1 nd2*

creates a cableSEL given an existing set of nodes. (To create a collection of cableSELs that lie between a beginning and ending point, or to assign properties to cableSELs, use the **SEL cable** command.) The new cableSEL will not be attached to the grid; if you wish to attach it to the grid, then, after creating it, position its nodes with the **SEL node init** {**x,y,z**}**pos** command (which will create links and set appropriate attachment conditions for all nodes that are moved into a zone). The following keywords are available.

cid ***cid***

specifies the component-ID number of the cableSEL. Each SEL has a unique component-ID number. If this value is not given, then the new cableSEL is assigned the next available component-ID number.

id ***id***

specifies the ID number of the cableSEL. All SELs that comprise a particular entity will possess the same ID number — e.g., all cableSELs that comprise cable 6 will have an ID of 6. If this value is not given, then the new cableSEL is assigned an ID of 0.

nodes ***nd1 nd2***

specifies the ID numbers (***nd1, nd2***) of two nodes that will define the cableSEL. These nodes must already exist — nodes can be created with the **SEL node** command. Ordering of the nodes defines the cableSEL coordinate system as follows. The positive x-direction lies along the line from ***nd1*** to ***nd2***, and the y-direction is found by projecting the global y- or x-directions onto the cableSEL cross-section.

delete keyword <**ra**nge . . . >

deletes SELs, nodes and links. Useful range elements include: **id**, **cid**, **seltype** and **selid**. If a SEL is deleted, then any dangling nodes (not being used by any SEL) and dangling links (not being used by any node) are also deleted.

beam deletes all beamSELs in the range.

cable deletes all cableSELs in the range.

geogrid deletes all geogridSELs in the range.

liner deletes all linerSELs in the range.

link deletes all links in the range.

node deletes all nodes in the range. If a deleted node is used by a link, then that link is also deleted. If a node is used by at least one SEL, then it cannot be deleted; instead, the SEL must be deleted.

pile deletes all pileSELs in the range.

sel deletes all SELs in the range.

shell deletes all shellSELs in the range.

Geogrid and GeogridSEL

This section describes the **SEL geogrid** and **SEL geogridsel** commands. The **SEL geogrid** command allows one to create a collection of geogridSELs that lie upon existing zone faces and specify properties of geogridSELs, while the **SEL geogridsel** command allows one to create individual geogridSELs.

There are two forms of the **SEL geogrid** command: one form for the creation of the geogrid; and the other for specifying geogrid properties and applied loads.

Command Form 1: Geogrid Creation

geogrid <**id** ***id***> <**c**rossdiag> <**ele**mtype ***ename***> <**gro**up ***gname***> <**ra**nge . . . >

creates a geogrid. The geogrid consists of a collection of geogridSELs that are attached to the set of 3- or 4-sided zone-faces that are specified with the optional keywords: **group** and **range**. The **range** defines the collection of zone faces upon which geogridSELs will be created. If **group** is not given, then only zone faces on the surface are considered. New nodes associated with the geogrid will also be created (see the **id** keyword). The nodes of each new geogridSEL will be ordered counterclockwise with respect to the outside of the specified zone faces, thereby making the geogridSEL z-axis point outward. The geogrid will be attached to the grid at its nodes with link attachment conditions appropriate to a geogrid — see Table 1.29 in **Structural Elements**. Note that, after creating the geogrid with this command, the zones may be deleted, and the geogrid may be positioned by moving its nodes with the **SEL node init** {**x**,**y**,**z**}**pos** command. The following keywords are available.

id ***id***

specifies the geogrid ID that will be assigned to all new geogridSELs created by this command. This keyword must immediately follow the **geogrid** keyword. If this value is not given, then the next available ID will be used. The ID controls potential joining of the geogrid nodes to an existing geogrid, as follows. At each potential new node location, a new node is created unless all of the following conditions are fulfilled: (1) the ID is given; (2) an existing node is found "near" to the potential new node location; and (3) the found node is part of a geogrid with ID equal to ***id***.

crossdiag

specifies the mesh configuration. If **crossdiag** is specified, then a cross-diagonal mesh will be created with a node lying at the centroid of all four-sided zone faces; otherwise, a crosshatch mesh will be created.

When connecting geogridSELs to the grid, it is best to locate nodes at gridpoints. If the finite-element type can resist bending (DKT, DKT-CST or DKT-(CST Hybrid) elements), then incompatibilities can develop along geogridSEL faces and zone faces, because the transverse (out-of-plane) displacement field varies cubically across geogridSEL faces, but varies only linearly across zone faces. Thus, when connecting bending-resistant geogridSELs rigidly to zone faces, it is best to utilize a crosshatch mesh, because the middle node of the cross-diagonal mesh will be constrained to translate according to the linear displacement field of the zone face, and this overconstrains the geogridSELs and can produce near-zero moments at these midnodes when present in a nonzero moment field.

elemtype ***ename***

specifies the finite element used by each geogridSEL. If no element type is specified, then the element type will be CST. Note that the type of finite element associated with each geogridSEL is created and cannot subsequently be altered. The finite element type of existing geogridSELs is printed by the **PRINT sel geogrid elemtype** command. The value of ***ename*** must be one of the following 3-noded finite elements:

cst CST (6 degrees-of-freedom) finite element. This is a membrane element — it resists membrane but does not resist bending loading.

csth CST Hybrid (9 degrees-of-freedom) finite element. This is a membrane element — it resists membrane but does not resist bending loading.

dkt DKT (9 degrees-of-freedom) finite element. This is a plate-bending element — it resists bending but does not resist membrane loading.

dkt_cst DKT-CST (15 degrees-of-freedom) finite element. This is a shell element that combines the DKT and CST elements to resist both bending and membrane loading.

dkt_csth DKT-(CST Hybrid) (18 degrees-of-freedom) finite element. This is a shell element that combines the DKT and CST Hybrid elements to resist both bending and membrane loading.

group ***gname***

specifies that new geogridSELs be created on the zone faces that lie within the optional range and are on the surface of the given group, even if this surface is internal to the grid. This allows one to create geogridSELs that lie upon zone faces that are internal to the grid.

Command Form 2: Specifying Geogrid Properties

geogrid <**id** ***id***> keyword <**ra**nge>

apply **p**ressure ***p***

assigns uniform applied pressure ***p*** to all geogridSELs in the range and with an ID number of ***id***; if ***id*** is not given, then all geogridSELs in the range are considered. Positive pressure acts in the positive z-direction of the geogridSEL system and maintains this orientation during large-strain motion.

property keyword ***value*** <keyword ***value***> . . .

assigns the specified property to all geogridSELs in the range and with an ID number of ***id***; if ***id*** is not given, then all geogridSELs in the range are considered. The geogridSEL itself remains elastic and can have either isotropic or orthotropic material properties. The following properties are available.

cs_scoh coupling spring cohesion (stress units)

cs_sfric coupling spring friction angle (degrees)

cs_sk coupling spring stiffness per unit area

density density (needed if dynamic mode or gravity is active)

isotropic isotropic material properties: E and v where E is Young's modulus and v is Poisson's ratio

orthotropic orthotropic material properties: $\bar{e}_{11}$, $\bar{e}_{12}$, $\bar{e}_{22}$, $\bar{e}_{33}$

slide large-strain sliding flag

slide_tol large-strain sliding tolerance

thexp thermal expansion coefficient

thickness geogrid thickness

geogridsel <**cid** ***cid***> <**id** ***id***> <**ele**mtype ***ename***> **no**des ***nd1 nd2 nd3***

creates a geogridSEL given an existing set of nodes. (To create a collection of geogridSELs that lie upon existing zone faces, or to assign properties to geogridSELS, use the **SEL geogrid** command.) The new geogridSEL will not be attached to the grid; if you wish to attach it to the grid, then, after creating it, position its nodes with the **SEL node init** {**x,y,z**}**pos** command (which will create links and set appropriate attachment conditions for all nodes that are moved into a zone). The following keywords are available.

cid ***cid***

specifies the component-ID number of the geogrid-SEL. Each SEL has a unique component-ID number. If this value is not given, then the new geogridSEL is assigned the next available component-ID number.

id ***id***

specifies the ID number of the geogridSEL. All SELs that comprise a particular entity will possess the same ID number — e.g., all geogridSELs that comprise geogrid 6 will have an ID of 6. If this value is not given, then the new geogridSEL is assigned an ID number of 0.

elemtype ***ename***

specifies the finite element used by each geogridSEL. If no element type is specified, then the element type will be CST. Note that the type of finite element associated with each geogridSEL is created and cannot subsequently be altered. The finite-element type of existing geogridSELs is printed by the **PRINT sel geogrid elemtype** command. The value of ***ename*** must be one of the following 3-noded finite elements:

cst CST (6 degrees-of-freedom) finite element. This is a membrane element — it resists membrane but does not resist bending loading.

csth CST Hybrid (9 degrees-of-freedom) finite element. This is a membrane element — it resists membrane but does not resist bending loading.

dkt DKT (9 degrees-of-freedom) finite element. This is a plate-bending element — it resists bending but does not resist membrane loading.

dkt_cst DKT-CST (15 degrees-of-freedom) finite element. This is a shell element that combines the DKT and CST elements to resist both bending and membrane loading.

dkt_csth DKT-(CST Hybrid) (18 degrees-of-freedom) finite element. This is a shell element that combines the DKT and CST Hybrid elements to resist both bending and membrane loading.

nodes ***nd1 nd2 nd3***

specifies the ID numbers (***nd1, nd2, nd3***) of three nodes that will define the geogridSEL. These nodes must already exist — nodes can be created with the **SEL node** command. Ordering of the nodes defines the geogridSEL coordinate system as follows. The nodes are oriented counterclockwise with respect to the positive z-direction, and the positive x-direction is parallel with the edge from ***nd1*** to ***nd2***.

group *gname* <color> <**ra**nge . . . >

assigns all SELs in the range to the group with the name of ***gname***.

SEL groups are sets of SELs identified by group name. Groups may not overlap, and each SEL may only belong to one group. A new group name will replace an old name. The **SEL group** command assigns a name (and color) to all SELs within the optional **range**. The command **PRINT group** lists the existing group names. The groups can be plotted with the **sel group** plot item. (This command is identical to the **GROUP** command, except it allows specification of SEL groups, not zone groups.)

Liner and Linersel

This section describes the **SEL liner** and **SEL linersel** commands. The **SEL liner** command allows one to create a collection of linerSELs that lie upon existing zone faces and specify properties of linerSELs, while the **SEL linersel** command allows one to create individual linerSELS.

There are two forms of the **SEL liner** command: one form for the creation of the liner, and the other for specifying liner properties and applied loads.

Command Form 1: Liner Creation

liner <**id** ***id***> <**c**rossdiag> <**ele**mtype ***ename***> <**gro**up ***gname***> <**ra**nge . . . >

creates a liner. The liner consists of a collection of linerSELs that are attached to the set of 3- or 4-sided zone faces that are specified with the optional keywords **group** and **range**. The **range** defines the collection of zone faces upon which linerSELs will be created. If **group** is not given, then only zone faces on the surface are considered. New nodes associated with the liner will also be created (see the **id** keyword). The nodes of each new linerSEL will be ordered counterclockwise with respect to the outside of the specified zone faces, thereby making the linerSEL z-axis point outward. The liner will be attached to the grid at its nodes with link attachment conditions appropriate to a liner — see Table 1.29 in **Structural Elements**. Note that, after creating the liner with this command, the zones may be deleted, and the liner may be positioned by moving its nodes with the **SEL node init** {**x,y,z**}**pos** command. The following keywords are available.

id ***id***

specifies the liner ID that will be assigned to all new linerSELs created by this command. This keyword must immediately follow the **liner** keyword. If this value is not given, then the next available ID will be used. The ID controls the potential joining of the liner nodes to an existing liner as follows. At each potential new node location, a new node is created unless all of the following conditions are fulfilled: (1) the ID is given; (2) an existing node is found "near" to the potential new node location; and (3) the found node is part of a liner with ID equal to ***id***.

crossdiag

specifies the mesh configuration. If **crossdiag** is specified, then a cross-diagonal mesh will be created with a node lying at the centroid of all four-sided zone faces; otherwise, a crosshatch mesh will be created.

When connecting linerSELs to the grid, it is best to locate nodes at gridpoints. If the finite-element type can resist bending (DKT, DKT-CST, or DKT-(CST Hybrid) elements), then incompatibilities can develop along linerSEL faces and zone faces because the transverse (out-of-plane) displacement field varies cubically across linerSEL faces, but varies only linearly across zone faces. Thus, when connecting bending-resistant linerSELs rigidly to zone faces, it is best to utilize a crosshatch mesh, because the middle node of the cross-diagonal mesh will be constrained to translate according to the linear displacement field of the zone face, and this overconstrains the linerSELs and can produce near-zero moments at these midnodes when present in a nonzero moment field.

elemtype ***ename***

specifies the finite element used by each linerSEL. If no element type is specified, then the element type will be DKT-CST. Note that the type of finite element associated with each linerSEL is set when the linerSEL is created and cannot subsequently be altered. The finite-element type of existing linerSELs is printed by the **PRINT sel liner elemtype** command. The value of ***ename*** must be one of the following 3-noded finite elements:

cst CST (6 degrees-of-freedom) finite element. This is a membrane element — it resists membrane but does not resist bending loading.

csth CST Hybrid (9 degrees-of-freedom) finite element. This is a membrane element — it resists membrane but does not resist bending loading.

dkt DKT (9 degrees-of-freedom) finite element. This is a plate-bending element — it resists bending but does not resist membrane loading.

dkt_cst DKT-CST (15 degrees-of-freedom) finite element. This is a shell element that combines the DKT and CST elements to resist both bending and membrane loading.

dkt_csth DKT-(CST Hybrid) (18 degrees-of-freedom) finite element. This is a shell element that combines the DKT and CST Hybrid elements to resist both bending and membrane loading.

group ***gname***

specifies that new linerSELs be created on the zone faces that lie within the optional range and are on the surface of the given group, even if this surface is internal to the grid. This allows one to create linerSELs that lie upon zone faces that are internal to the grid.

Command Form 2: Specifying Liner Properties

liner <**id** ***id***> keyword <**ra**nge>

apply **p**ressure ***p***

assigns uniform applied pressure ***p*** to all linerSELs in the range and with an ID number of ***id***; if ***id*** is not given, then all linerSELs in the range are considered. Positive pressure acts in the positive z-direction of the linerSEL system and maintains this orientation during large-strain motion.

property keyword ***value*** <keyword ***value***> . . .

assigns the specified property to all linerSELs in the range and with an ID number of ***id***; if ***id*** is not given, then all linerSELs in the range are considered. The linerSEL itself remains elastic and can have either isotropic or orthotropic material properties. The following properties are available.

cs_ncut normal coupling spring tensile strength (stress units)

cs_nk normal coupling spring stiffness per unit area

cs_scoh shear coupling spring cohesion (stress unit)

cs_scohres shear coupling spring residual cohesion (stress units)

cs_sfric shear coupling spring friction angle (degrees)

cs_sk shear coupling spring stiffness per unit area

density density (needed if dynamic mode or gravity is active)

isotropic isotropic material properties: E and v where E is Young's modulus and v is Poisson's ratio

orthotropic orthotropic material properties: $\bar{e}_{11}$, $\bar{e}_{12}$, $\bar{e}_{22}$, $\bar{e}_{33}$

slide	large-strain sliding flag
slide_tol	large-strain sliding tolerance
thexp	thermal expansion coefficient
thickness	liner thickness

linersel <**cid** *cid*> <**id** *id*> <**ele**mtype *ename*> **no**des *nd1 nd2 nd3*

creates a linerSEL given an existing set of nodes. (To create a collection of linerSELs that lie upon existing zone faces, or to assign properties to linerSELs, use the **SEL liner** command.) The new linerSEL will not be attached to the grid; if you wish to attach it to the grid, then, after creating it, position its nodes with the **SEL node init** {**x,y,z**}**pos** command (which will create links and set appropriate attachment conditions for all nodes that are moved into a zone). The following keywords are available.

cid *cid*

specifies the component-ID number of the linerSEL. Each SEL has a unique component-ID number. If this value is not given, then the new linerSEL is assigned the next available component-ID number.

id *id*

specifies the ID number of the linerSEL. All SELs that comprise a particular entity will possess the same ID number — e.g., all linerSELs that comprise liner 6 will have an ID of 6. If this value is not given, then the new linerSEL is assigned an ID number of 0.

elemtype *ename*

specifies the finite element used by each linerSEL. If no element type is specified, then the element type will be DKT-CST. Note that the type of finite element associated with each linerSEL is set when the linerSEL is created and cannot subsequently be altered. The finite-element type of existing linerSELs is printed by the **PRINT sel liner elemtype** command. The value of ***ename*** must be one of the following 3-noded finite elements:

cst CST (6 degrees-of-freedom) finite element. This is a membrane element — it resists membrane but does not resist bending loading.

csth CST Hybrid (9 degrees-of-freedom) finite element. This is a membrane element — it resists membrane but does not resist bending loading.

dkt DKT (9 degrees-of-freedom) finite element. This is a plate-bending element — it resists bending but does not resist membrane loading.

dkt_cst DKT-CST (15 degrees-of-freedom) finite element. This is a shell element that combines the DKT and CST elements to resist both bending and membrane loading.

dkt_csth DKT-(CST Hybrid) (18 degrees-of-freedom) finite element. This is a shell element that combines the DKT and CST Hybrid elements to resist both bending and membrane loading.

nodes ***nd1 nd2 nd3***

specifies the ID numbers (***nd1, nd2, nd3***) of three nodes that will define the linerSEL. These nodes must already exist — nodes can be created with the **SEL node** command. Ordering of the nodes defines the linerSEL coordinate system as follows. The nodes are oriented counterclockwise with respect to the positive z-direction, and the positive x-direction is parallel with the edge from ***nd1*** to ***nd2***.

link keyword <**ra**nge . . . >

A link is an object that links a source node to a target entity that may be either another node or a zone. Each link utilizes the local system of its source node, and all link properties are specified with respect to this local system. Links implement the interactions that occur between the different types of SELs and the grid. In most cases, it will not be necessary to create or modify links, because they will be created and their properties set automatically by the SELs that utilize them.

However, if one wishes to introduce a plastic hinge with full rotational freedom (such that two different rotation angles can develop on each side of the hinge point), then one must create two separate nodes at this point and create a node-to-node link between them and specify a normal-yield spring in the appropriate rotational degree-of-freedom, and set the stiffness and yield strength of this spring equal to that of the plastic hinge. For these, and other more complex situations, we provide the following interface to the link logic.

Whenever a link attempts to establish a connection to a target zone, it will search for a non-null zone for which the source node lies within a distance δ of the zone's boundary. The value of δ is obtained from the global value of zone tolerance (see the **SEL set link zone_tol** command) multiplied by zone size, where zone size is the maximum x-, y- or z-dimension of the zone bounding box. But note that such a nearby zone will be used only if the source node does not lie within or on the boundary of any non-null zone. If the source node lies within the δ-boundary of a zone, then the weighting functions used to transfer information from the link to the zone will correspond with the location on the zone surface that is nearest to the node location.

There are two forms of the **SEL link** command: one form for the creation of a link, and the other for specifying properties of the link. There are two methods to create links.

Command Form 1: Creating a Single Link

link <**id** ***id***> ***sid*** <**ta**rget [**no**de **t**gt_num ***tid***], [**zo**ne <**t**gt_num ***tid***>]>

creates a new link that links a source node ***sid*** with a target entity that may be either another node or a zone. The ***id*** is the ID number of the new link. The ***sid*** is the ID number of an existing node that will become the source node of the new link. The optional keyword **target** is used to identify the target entity. The default target is **zone**.

For zone targets, if ***tid*** is not specified, then a non-null zone that is within a distance δ of the source node is utilized; otherwise, if ***tid*** is specified, then that zone is used if it is non-null and the source node is within a distance δ of its boundary. (See discussion above for definition of δ.)

For node targets, ***tid*** must be specified and the two nodes must be near to one another for a link to be created. Near is defined as being within a distance δ of one another, where δ is the global value of node tolerance (see the **SEL set link node_tol** command). If the source and target entities cannot be identified, then an error is notified and nothing changes. The attachment condition of the new link are set to "rigid" for all six directions.

Command Form 2: Creating a Network of Links

link

net <**ra**nge . . . >

For every node in the optional **range**, if it does not already have a link, and if it is within a distance δ of a non-null zone, then a link is created between the node and the zone. (See discussion above for definition of δ.) The attachment conditions of all new links are set to "rigid" for all six directions.

The following keywords allow to specify link properties.

attach keyword1 keyword2 <**r**ange . . . >

sets attachment conditions for the specified link directions. The current attachment condition will be replaced by the condition specified here. The direction is given by the following values of keyword1:

xdirection translational x-direction, also degree-of-freedom 1

xrdirection rotational x-direction, also degree-of-freedom 4

ydirection translational y-direction, also degree-of-freedom 2

yrdirection rotational y-direction, also degree-of-freedom 5

zdirection translational z-direction, also degree-of-freedom 3

zrdirection rotational z-direction, also degree-of-freedom 6

The attachment condition is given by the following values of keyword2:

free free

lindeform deformable linear spring

nydeform deformable normal-yield spring

rigid rigid

constit keyword1 ***dof*** keyword2 ***value*** <**r**ange . . . >

sets property keyword2 of deformable spring type keyword1 in direction ***dof***, where ***dof*** ∈ {1, 2, . . . , 6}. The appropriate deformable spring must already be present; if it is not, then this command does nothing. Deformable springs can be created using the **SEL link attach** command. Two deformable spring types are available:

lindeform ***dof*** keyword2 ***value***

The linear spring properties are given by the following values of keyword2:

area ***value*** (default is 1.0)

area

k ***value*** (default is 1.0)

stiffness per unit area

nydeform ***dof*** keyword2 ***value***

The normal-yield spring properties are given by the following values of keyword2:

area ***value*** (default is 1.0)

area

gap **off** (default)
on

gap-use flag. A gap may form in both the positive and negative directions whenever yielding is occurring in these directions. The gap is always being tracked, but only affects behavior if gap-use flag is on. If the gap-use flag is on, then the gap must close before forces will develop in the alternate direction.

k	***value*** (default is 1.0) stiffness per unit area
ycomp	***value*** (default is 1e20) compressive yield strength (force units)
ytens	***value*** (default is 1e20) tensile yield strength (force units)

node

The **SEL node** command allows one to both create and specify properties of nodes. Node properties include applied point loads, velocity-fixity conditions, velocity, displacement, position and local-damping factor, as well as specification of the node-local system.

There are two coordinate systems associated with each node: the global system and the local system.

1. The global system can be used to specify generalized velocity and displacement boundary conditions, nodal positions and applied loads. The global system does not change throughout the course of a simulation.

2. The local system is used to specify attachment conditions that control how the node interacts with the grid. Also, the equations of motion are solved in these local directions; therefore, one may **fix** or **free** velocities in these directions only. Also note that any degrees-of-freedom of a node with a RIGID attachment condition (such as the node-local y- and z-axes of a node used by a cable, which are rigidly attached to the zone in which the cable is embedded) cannot be **fix**ed or **free**d, because the velocity of these degrees-of-freedom comes from the entity to which it is attached.

The orientation of the local system is set automatically at the start of a set of cycles (or when the **CYCLE 0** command is executed) based on the type of SELs that use the node. For beams and shells, the local system is aligned with the global system. For cables and piles, the local system is oriented such that: (1) the x-axis is aligned with the average axial direction of all cableSELs or pileSELs using the node; and (2) the yz-axes are arbitrarily oriented in the cable or pile cross-sectional plane — see Figure 1.25. For geogrids and liners, the local system is oriented such that: (1) the z-axis is aligned with the average normal direction of all geogridSELs or linerSELs using the node; and (2) the xy-axes are arbitrarily oriented in the geogrid or liner tangent plane — see Figure 1.26. (If more than one type of SEL is using a node, then the orientation of the local system will correspond with the first SEL type in the following list: liner, geogrid, pile, cable, shell and beam. Thus, if a node is being used by both a geogrid and a cable, then the orientation of the local system will correspond with that of a geogrid.) When running in large-strain mode, the orientation of the local system is updated automatically, based on the motion of the SELs that use the node (unless this behavior is inhibited by the **fix lsys** keyword).

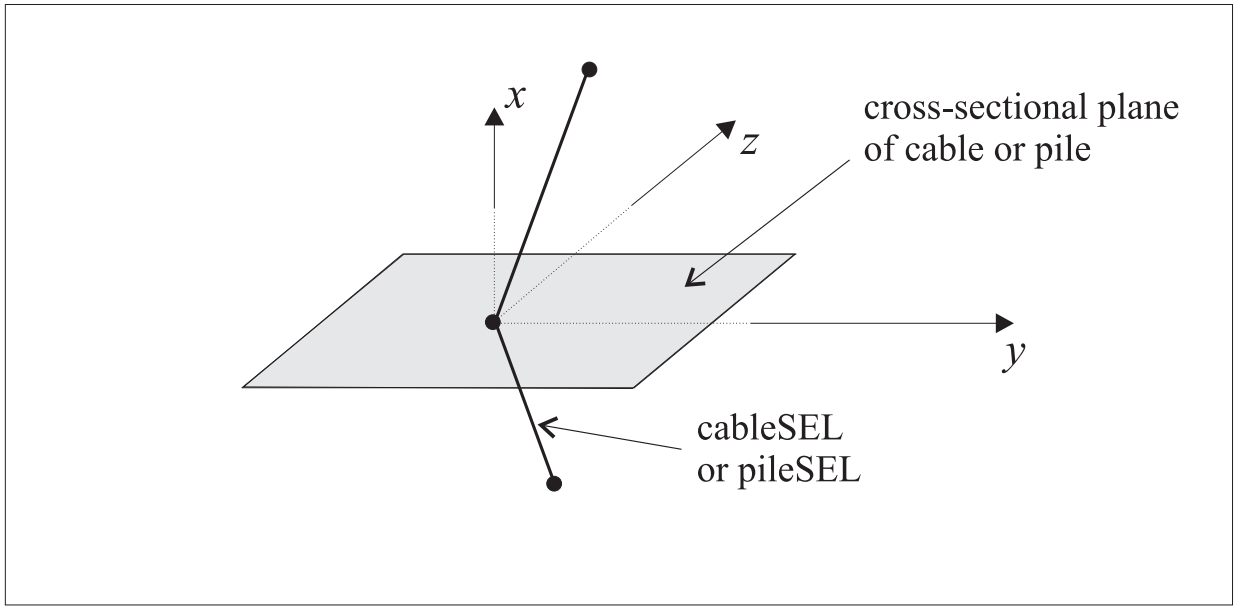


Figure 1.25 Orientation of the node-local system for a node used by cableSELs or pileSELs

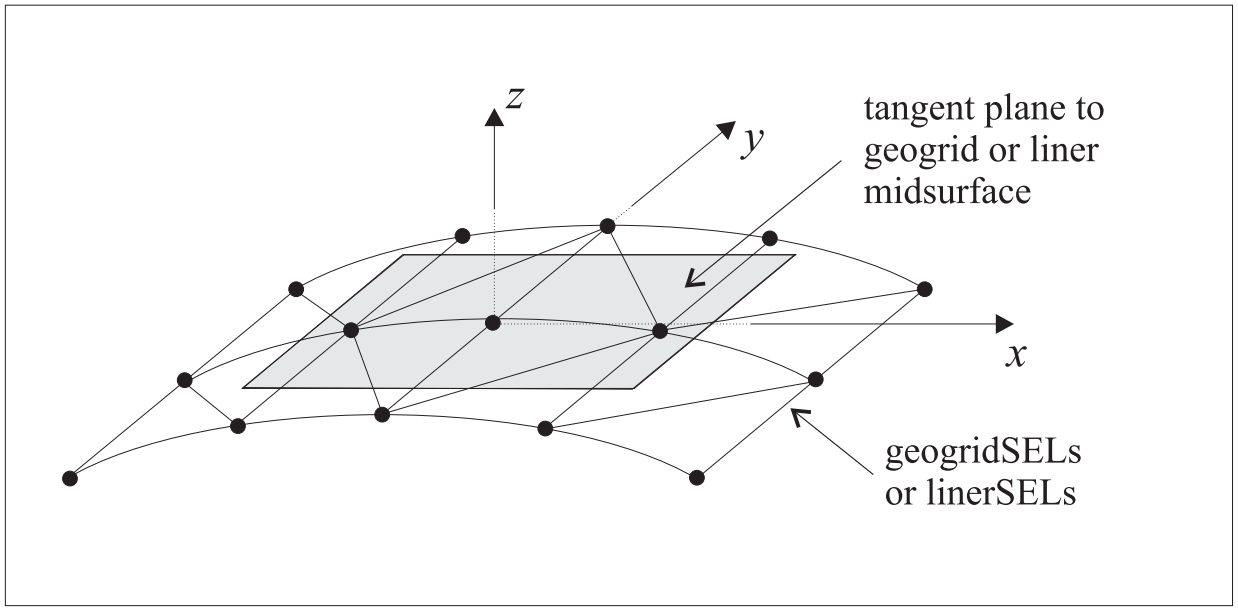


Figure 1.26 Orientation of the node-local system for a node used by geogrid-SELs or linerSELs

There are two forms of the **SEL node** command: one form for the creation of the node, and the other for specifying node properties and applied loads.

Command Form 1: Node Creation

node <**id** *id*> *x, y, z*

creates a node at location (*x, y, z*). Note that nodes are created automatically when creating SELs, using the SEL-creation commands: **SEL** type, where type = {**beam, cable, pile, shell, geogrid** or **liner**}. If individual SELs are to be created using the commands: **SEL** type, where type = {**beamsel, cablesel, pilesel, shellsel, geogridsel** or **linersel**}, then nodes must first be created manually.

id *id*

specifies the ID number of the node. Each node has a unique ID number. If this value is not given, then the new node is assigned the next available ID number.

Command Form 2: Specifying Node Properties

node keyword <**ra**nge . . . >

apply keyword . . .

sets generalized point loads for all nodes in the range. Generalized point loads consist of an applied force and an applied moment. Pressure loading can be applied to shell surfaces using the **SEL** type **apply** commands, where type = {**shell, geogrid, liner**}. Distributed loads can be applied to beam and pile surfaces using the **SEL** type **apply** commands, where type = {**beam, pile**}. The following keywords are available to define generalized point loads.

force *Fx, Fy, Fz*

applied force vector (in terms of system defined by the **system** keyword)

moment *Mx, My, Mz*

applied moment vector (in terms of system defined by the **system** keyword)

remove [**fo**rce, **m**oment]

An applied condition remains in effect until it is removed with this command.

system [**g**lobal, **l**ocal]

A coordinate system in which the force and moment vectors are expressed. The default system is global. If the system is set to **local**, then in large-strain mode, the force and moment vectors will remain aligned with the local system as it rotates.

fix keyword <keyword> . . .

fixes velocity in the specified directions in terms of the node-local system for all nodes in the range, thereby preventing these velocities from changing. If a fixed displacement is required, the appropriate velocities should be initialized to zero. Degrees-of-freedom with a RIGID attachment condition cannot be **fix**ed — the RIGID condition overrides the fix condition. Also allows node-local system to be fixed.

lsys inhibits automatic updating of the node-local system based on the SELs that use the node. The default behavior is free. Consult the **local** keyword for more information.

x translational velocity in x-direction (node-local system)

xrot rotational velocity about x-axis (node-local system)

y translational velocity in y-direction (node-local system)

yrot rotational velocity about y-axis (node-local system)

z translational velocity in z-direction (node-local system)

zrot rotational velocity about z-axis (node-local system)

free keyword <keyword> . . .

frees velocities in the specified directions in terms of the node-local system for all nodes in the range; also allows node-local system to be freed.

lsys allows automatic updating of the node-local system based on the SELs that use the node. This is the default behavior. Consult the **local** keyword for more information.

x translational velocity in x-direction (node-local system)

xrot rotational velocity about x-axis (node-local system)

y translational velocity in y-direction (node-local system)

yrot rotational velocity about y-axis (node-local system)

z translational velocity in z-direction (node-local system)

zrot rotational velocity about z-axis (node-local system)

init keyword <[**add**, **mul**tiply]> ***v*** <[**gra**d ***gx, gy, gz***> <keyword ***v*** . . . >

initializes nodal quantities of velocity, displacement and position of all nodes in the range. Note that all quantities must be specified in the global system.

The following optional keyword phrases are available to modify the behavior of this command.

add ***v***

the existing value of the parameter (of all nodes in the range) is set to its current value plus the given value. For example, one can add 0.1 to all node x-displacements with the command **SEL node init xdisp add 0.1**.

grad ***gx, gy, gz***

The value installed in each node in the range is $v_f = v + xg_x + yg_y + zg_z$, where the **grad** keyword is given immediately after the value ***v***, and (x, y, z) is the reference position of the node. If the **multiply** keyword is also present, then the given gradient is applied to the multiplier.

multiply ***v***

the existing value of the parameter (of all nodes in the range) is multiplied by the given value rather than being set to the given value. For example, one can multiply all node x-displacements by a factor of 1.5 with the command **SEL node init xdisp multiply 1.5**.

The following keywords are available to initialize nodal quantities.

xdisp translational displacement (x-component, global system)

xpos x-coordinate of node (global system) to be used for positioning of a structural-element grid; must be used after creating the node, but before executing any cycles. Any node that is moved into a zone (see the **SEL link** command) will have its link deleted (if one is present) and will have a new link created with attachment conditions corresponding to the type of structural element using the node (see Table 1.29 in **Structural Elements**). If more than one SEL is using the node, then the attachment condition will correspond with the first SEL type in the following list: liner, geogrid, pile, cable, shell and beam. Thus, if a node is being used by both a geogrid and a cable, then the attachment condition after moving this node will correspond with that of a geogrid.

xrdisp rotational displacement (x-component, global system). Beware that the rotational displacement is not a true vector quantity.

xrvel rotational velocity (x-component, global system)

xvel translational velocity (x-component, global system)

ydisp translational displacement (y-component, global system)

ypos y-coordinate of node (global system) — see description for **xpos** keyword

yrdisp rotational displacement (y-component, global system). Beware that the rotational displacement is not a true vector quantity.

yrvel rotational velocity (y-component, global system)

yvel translational velocity (y-component, global system)

zdisp translational displacement (z-component, global system)

zpos z-coordinate of node (global system) — see description for **xpos** keyword

zrdisp rotational displacement (z-component, global system). Beware that the rotational displacement is not a true vector quantity.

zrvel rotational velocity (z-component, global system)

zvel translational velocity (z-component, global system)

ldamp ***dfac*** sets the local-damping factor to ***dfac*** for all nodes in the range — see the **SEL set damp** command. By default, ***dfac*** equals 0.8.

local **x**dir ***Xx, Xy, Xz*** **y**dir ***Yx, Yy, Yz***

sets the orientation of the node-local system for all nodes in the range. The ***x*** and ***y*** vectors define the x- and y-directions as follows. The x-direction will be parallel with ***x***. The y-direction will be equal to the projection of ***y*** onto the plane with normal in the x-direction. The z-direction will be equal to the cross-product of the x- and y-directions. At the start of a set of cycles, the orientation of the node-local system is set automatically based on the type of SELs that use the node (see the discussion above); however, this behavior can be overridden by setting the orientation

of the node-local system with this keyword, and using the **fix lsys** keyword.

Pile and PileSEL

This section describes the **SEL pile** and **SEL pilesel** commands. The **SEL pile** command allows one to create a collection of pileSELs that lie between a beginning and ending point, and specify properties of pileSELs, while the **SEL pilesel** command allows one to create individual pileSELs and apply loads.

There are two forms of the **SEL pile** command: one form for the creation of the pile, and the other for specifying pile properties.

Command Form 1: Pile Creation

pile <**id** ***id***> **b**egin ***x,y,z*** **end** ***x,y,z*** <**n**seg ***nseg***>

creates a pile. The pile consists of a collection of ***nseg*** (default = 1) pileSELs that lie along a straight line between the **begin** and **end** locations. New nodes associated with the pile will also be created (see the **id** keyword). The nodal connectivity of each new pileSEL will be ordered such that the direction from end-1 to end-2 corresponds with the direction from the **begin** point to the **end** point. The pile will be attached to the grid at its nodes with link attachment conditions appropriate to a pile — see Table 1.29 in **Structural Elements**. The following keyword is available.

id ***id***

specifies the pile ID that will be assigned to all new pileSELs created by this command. This keyword must immediately follow the **pile** keyword. If this value is not given, then the next available ID will be used. The ID controls potential joining of the two pile end-nodes to an existing pile as follows. At each end-node location, a new node is created unless all of the following conditions are fulfilled: (1) the ID is given; (2) an existing node is found "near" to the potential new node location; and (3) the found node is part of a pile with ID equal to ***id***.

Command Form 2: Specifying Pile Properties

pile <**id** ***id***> keyword <**ra**nge>

apply **y**dist ***yd*** **z**dist ***zd***

assigns uniform applied distributed loads (force per unit length) to all pileSELs in the range and with an ID number of ***id***; if ***id*** is not given, then all pileSELs in the range are considered. Positive loads act in the positive *y*- or *z*-directions of the pileSEL system and maintain this orientation during large-strain motion. Point loads are applied at the nodes using the **SEL node apply** command.

property keyword ***value*** <keyword ***value***> . . .

assigns the specified property to all pileSELs in the range and with an ID number of ***id***; if ***id*** is not given, then all pileSELs in the range are considered. The following properties are available.

cs_cfincr shear direction incremental confining stress flag (rockbolt logic)

cs_cftable shear direction incremental confining stress factor table number (rockbolt logic)

cs_ncoh normal coupling spring cohesion (force / unit length)

cs_nfric normal coupling spring friction angle (degrees)

cs_ngap normal coupling spring gap-use flag

cs_nk normal coupling spring stiffness per unit length

cs_scoh shear coupling spring cohesion(force / unit length)

cs_sctable shear direction cohesive strength table number (rockbolt logic)

cs_sfric shear coupling spring friction angle

cs_sftable shear direction friction angle table number (rockbolt logic)

cs_sk	shear coupling spring stiffness per unit length
density	density (needed if dynamic mode or gravity are active)
emod	Young's modulus
nu	Poisson's ratio
perimeter	exposed perimeter
pmoment	plastic moment capacity
rockbolt	rockbolt flag (activates rockbolt logic)
slide	large-strain sliding flag
slide_tol	large-strain sliding tolerance
tfstrain	tensile failure strain (non-dimensional, rockbolt logic)
thexp	thermal expansion coefficient
tyield	axial tensile yield strength (force units, rockbolt logic)
xcarea	cross-sectional area
xciy	second moment with respect to pileSEL *y*-axis
xciz	second moment with respect to pileSEL *z*-axis
xcj	polar moment of inertia
ydirection	vector (***Yx, Yy, Yz***) whose projection onto the pileSEL cross-section defines the *y*-axis of the pileSEL system

pilesel <**cid** *cid*> <**id** *id*> **no**des *nd1 nd2*

creates a pileSEL given an existing set of nodes. (To create a collection of pileSELs that lie between a beginning and ending point, or to assign properties to pileSELs, use the **SEL pile** command.) The new pileSEL will not be attached to the grid; if you wish to attach it to the grid, then, after creating it, position its nodes with the **SEL node init** {**x,y,z**}**pos** command (which will create links and set appropriate attachment conditions for all nodes that are moved into a zone). The following keywords are available.

cid *cid*

specifies the component-ID number of the pileSEL. Each SEL has a unique component-ID number. If this value is not given, then the new pileSEL is assigned the next available component-ID number.

id *id*

specifies the ID number of the pileSEL. All SELs that comprise a particular entity will possess the same ID number — e.g., all pileSELs that comprise pile 6 will have an ID of 6. If this value is not given, then the new pileSEL is assigned an ID number of 0.

nodes *nd1 nd2*

specifies the ID numbers (***nd1, nd2***) of two nodes that will define the pileSEL. These nodes must already exist — nodes can be created with the **SEL node** command. Ordering of the nodes defines the pileSEL coordinate system as follows. The positive x-direction lies along the line from ***nd1*** to ***nd2***, and the y-direction is found by projecting the global y- or x-directions onto the pileSEL cross-section. The y-direction can also be modified by the **SEL pile prop ydir** command.

recover keyword <**ra**nge . . . >

The stress-recovery procedure can be applied to all shell-type SELs to compute both the stress resultants and the stresses acting at a specified depth. The **range** keyword identifies the patch of shell-type SELs to which the command will apply. Note that the recovered quantities will depend upon the range of SELs over which these quantities are being recovered, because nodal averaging only occurs for SELs within this range.

sres Recover the eight stress resultants for all shell-type structural elements (shellSELs, geogridSELs and linerSELs) in the optional range. The stress resultants are expressed in terms of the surface coordinate system. This command assumes that a consistent surface coordinate system has been established for the group of shell-type SELs in the range — see the **surface** keyword. The bending and membrane stress resultants (M_x, M_y, M_{xy}, N_x, N_y and N_{xy}) vary linearly over each element, whereas the transverse-shear stress resultants (Q_x and Q_y) are constant over each element. The recovery procedure first computes the average values of bending and membrane stress resultants (by averaging at the nodes the contributions from each shell-type SEL in the range), and then spatially differentiates this average bending field over each element to obtain the transverse-shear stress resultants.

Stress resultants can be queried after recovery by the command **PRINT sel recover sres**, and by the *FISH* function **`sst_sres`**. The validity of the stress resultants for a particular shell-type SEL can be queried by the *FISH* function **`sst_sresvalid`**.

stress <**d**epth_factor ***v***>

Recover stress tensor (expressed in global coordinate system) at specified depth in all shell-type SELs in the optional range. The depth equals ***v*** times $t/2$, where t is shell thickness. The depth factor, v, must be in the range [-1, +1]. v equal to +1 / -1 corresponds with the outer/inner shell surface (outer surface defined by positive surface system z-direction), and v equal to zero corresponds with the shell midsurface. If v is not specified, it defaults to +1.

Stresses are recovered at the three nodal points and centroid of each element. If we designate the shell

midsurface by xy-axes, then: (1) stress components σ_{xx}, σ_{yy} and σ_{xy} vary linearly over each element; (2) stress components σ_{zy} and σ_{zx} are constant over each element; and (3) σ_{zz} = 0 over each element. The stresses are derived from the internal element forces (see **PRINT sel shell nforce**). If the stress resultants are not valid when this command is executed, then an attempt is made to first recover them; if this attempt fails, then an error message is displayed indicating the problem — usually the inconsistency of the surface system, which must then be established manually using the **SEL recover surface** command.

Stresses and principal stresses can be queried after recovery by the commands **PRINT sel recover stress** and **PRINT sel recover pstress**, and by the *FISH* functions **sst_str** and **sst_pstr**. The depth at which these stresses have been recovered can be queried by the command **PRINT sel recover depth_factor**, and by the *FISH* function **sst_depfac**. The validity of the stresses and principal stresses for a particular shell-type SEL can be queried by the *FISH* function **sst_strvalid**.

surface **s**urfX ***Xx Xy Xz***

The **surfx** vector (***Xx, Xy, Xz***) enables a surface coordinate system to be generated for all nodes used by the shell-type SELs in the optional range. The surface coordinate system, $x'y'z'$, has the following properties: (1) z' is normal to the surface; (2) x' is the projection of the given **surfx** vector onto the surface; and (3) y' is orthogonal to x' and z'. The z'-direction is found at each node by taking the average normal direction of all shell-type SELs in the range are considered. If the **surfx** vector is aligned at z' at any node, then processing stops and an error message is displayed. To proceed, designate a different **surfx** vector, or restrict the range of shell-type SELs considered.

The surface coordinate system can be queried by the command **PRINT sel recover surface** and the *FISH* function **nd_ssys**. It can also be set for an individual node with the *FISH* function **nd_ssysx**. It can be visualized by the **sel recover** and **sel geom** plot items by setting the **surfsys** switchword to **on**. If the surface system is valid at a node, then the $x'y'z'$ directions

will be drawn as a unit-triad; otherwise, if the surface system is not valid at a node, then a filled sphere will be drawn.

The validity of the surface system at a particular node can be queried by the *FISH* function **nd_svalid**. The surface system at a node automatically becomes invalid under the following conditions: (1) large-strain update; or (2) creation or deletion of a shell-type SEL that uses the node. Validity must be re-established by the **SEL recover surface** command.

set keyword

modifies global program settings that control the SEL logic. The values of these settings can be printed using the **PRINT sel info** command. The following keywords are available.

damp keyword

combined sets the damping scheme used by the SEL logic to combined-local damping. The local-damping constants are stored separately for each node, and set by the command **SEL node ldamp**.

local sets the damping scheme used by the SEL logic to local damping. The local-damping constants are stored separately for each node, and set by the command **SEL node ldamp**.

rayleigh ***beta alpha***

sets the damping scheme used by the SEL logic to Rayleigh damping. The stiffness-proportional constant is ***beta***, and the mass-proportional constant is ***alpha***.

liner **g**ap_factor ***g***

sets the gap factor used during large-strain sliding of liners. During large-strain sliding, if the gap that forms between a node used by a linerSEL and the zone to which it is attached becomes greater than ***g*** times the zone size (where zone size is the maximum *x*-, *y*- or *z*-dimension of the zone bounding box), and the normal spring is in tension, then the simulation stops and an error message is displayed. If this large gap is physically reasonable, then the simulation can be continued by either increasing ***g*** or reducing the liner tensile strength such that the normal spring breaks. By default, ***g*** equals 0.1.

link keyword

node_tol ***ntol***

sets the tolerance used when establishing node-to-node links. A node-to-node link can only be established if the distance between the two nodes is less than ***ntol***. By default, ***ntol*** equals 1×10^{-5}.

zone_tol ***ztol***

sets the tolerance used when establishing node-to-zone links. A node is considered to be in a zone if it is within a distance δ of the zone boundary, where δ equals ***ztol*** times zone size (taken as the maximum x-, y- or z-dimension of the zone bounding box). Nodes that are used by SELs with their sliding flag on will utilize **slide_tol** instead of **zone_tol** when attempting to update the zone interpolation point of an existing link. By default, ***ztol*** equals 1×10^{-5}.

safety_fac ***sfac***

sets the factor by which the timestep necessary for solution stability of the SEL computations will be multiplied. The actual timestep used will be the minimum of the SEL timestep and the timesteps for the grid and other active modules — e.g., fluid, thermal or creep. In static mode, the final safety factor is one-half of ***sfac***; in dynamic mode, the final safety factor is ***sfac***. By default, ***sfac*** equals 1.0.

scale_rmass **off** (default)
on

This value applies to dynamic analysis only. If set to off, then the rotational degree-of-freedom masses are computed using a sphere with a volume equal to that contributed by SELs using the node. This is described as fully dynamic mode in the SEL chapter. If set to on, then the rotational degree-of-freedom masses are scaled based on rotational stiffnesses. This is described as partially dynamic mode in the SEL chapter.

v20ndcmd **off** (default)
on

The **SEL node** command was modified in Version 2.1 such that data files created using Version 2.0 and earlier may not behave as desired for newer versions of *FLAC3D*. If this occurs, we strongly suggest that you modify the data files to correspond with the new logic (see the **SEL node** command in this manual); however, you *may* be able to get the old data files to run correctly by setting this flag to on. When this flag is on, the behavior of the **SEL node init**, **SEL node fix/free** and **SEL node apply** commands reverts to that of Version 2.0. (It may be necessary to execute a **CYCLE 0** command, before issuing the old commands in order to automatically orient the node-local system, because this system affects the behavior of the Version 2.0 command.)

Shell and ShellSEL

This section describes the **SEL shell** and **SEL shellsel** commands. The **SEL shell** command allows one to create a collection of shellSELs that lie upon existing zone faces and specify properties of shellSELs, while the **SEL shellsel** command allows one to create individual shellSELs.

There are two forms of the **SEL shell** command: one form for the creation of the shell, and the other for specifying shell properties and applied loads.

shell <**id** *id*> <**c**rossdiag> <**ele**mtype *ename*> <**gro**up *gname*> <**ra**nge . . . >

creates a shell. The shell consists of a collection of shellSELs that are attached to the set of 3- or 4-sided zone faces that are specified with the optional keywords: **group** and **range**. The **range** defines the collection of zone faces upon which shellSELs will be created. If **group** is not given, then only zone faces on that surface are considered. New nodes associated with the shell will also be created (see the **id** keyword). The nodes of each new shellSEL will be ordered counterclockwise with respect to the outside of the specified zone faces, thereby making the shellSEL z-axis point outward. The shell will be attached to the grid at its nodes such that the translational degrees-of-freedom are rigidly connected to the grid, and the rotational degrees-of-freedom are free. Note that, after creating the shell with this command, the zones may be deleted, and the shell may be positioned by moving its nodes with the **SEL node init** {**x,y,z**}**pos** command. Also, if no attachment to the grid is desired, then after creating and positioning the shell, the links may be deleted with the **SEL delete link** command. The following keywords are available.

id *id*

specifies the shell ID that will be assigned to all new shellSELs created by this command. This keyword must immediately follow the **shell** keyword. If this value is not given, then the next available ID will be used. The ID controls potential joining of the shell nodes to an existing shell as follows. At each potential new node location, a new node is created unless all of the following conditions are fulfilled: (1) the ID is given; (2) and existing node is found “near” to the potential new node location; and (3) the found node is part of a shell with ID equal to ***id***.

crossdiag

specifies the mesh configuration. If **crossdiag** is specified, then a cross-diagonal mesh will be created with a node lying at the centroid of all four-sided zone faces; otherwise, a crosshatch mesh will be created.

When connecting shellSELs to the grid, it is best to locate nodes at gridpoints. If the finite-element type can resist bending (DKT, DKT-CST or DKT-(CST hybrid) elements), then incompatibilities can develop along shellSEL faces and zone faces because the transverse (out-of-plane) displacement field varies cubically across shellSEL faces, but varies only linearly across zone faces. Thus, when connecting bending-resistant shellSELs rigidly to zone faces, it is best to utilize a crosshatch mesh, because the middle node of the cross-diagonal mesh will be constrained to translate according to the linear displacement field of the zone face, and this overconstrains the shellSELs and can produce near-zero moments at these midnodes when present in a nonzero moment field.

elemtype ***ename***

specifies the finite element used by each shellSEL. If no element type is specified, then the element type will be DKT-CST. Note that the type of finite element associated with each shellSEL is set when the shellSEL is created and cannot subsequently be altered. The finite-element type of existing shellSELs is printed by the **PRINT sel shell elemtype** command. The value of ***ename*** must be one of the following 3-noded elements.

cst CST (6 degrees-of-freedom) finite element. This is a membrane element — it resists membrane but does not resist bending loading.

csth CST Hybrid (9 degrees-of-freedom) finite element. This is a membrane element — it resists membrane but does not resist bending loading.

dkt DKT (9 degrees-of-freedom) finite element. This is a plate-bending element — it resists bending but does not resist membrane loading.

dkt_cst DKT-CST (15 degrees-of-freedom) finite element. This is a shell element that combines the DKT and CST elements to resist both bending and membrane loading.

dkt_csth DKT-(CST Hybrid) (18 degrees-of-freedom) finite element. This is a shell element that combines the DKT and CST Hybrid elements to resist both bending and membrane loading.

group ***gname***

specifies that new shellSELs be created on the zone faces that lie within the optional range and are on the surface of the given group, even if this surface is internal to the grid. This allows one to create shellSELs that lie upon zone faces that are internal to the grid.

shell <**id** ***id***> keyword <**ra**nge>

apply **p**ressure ***p***

assigns uniform applied pressure ***p*** to all shellSELs in the range and with an ID number of ***id***; if ***id*** is not given, then all shellSELs in the range are considered. Positive pressure acts in the positive z-direction of the shellSEL system and maintains this orientation during large-strain motion.

property keyword ***value*** <keyword ***value***> . . .

assigns the specified property to all shellSELs in the range and with an ID number of ***id***; if ***id*** is not given, then all shellSELs in the range are considered. The shellSEL itself remains elastic and can have either isotropic or orthotropic material properties. The following properties are available.

density density (needed if dynamic mode or gravity is active)

isotropic isotropic material properties: E and v, where E is Young's modulus and v is Poisson's ratio

orthotropic orthotropic material properties: $\bar{e}_{11}$, $\bar{e}_{12}$, $\bar{e}_{22}$, $\bar{e}_{33}$

thexp thermal expansion coefficient

thickness shell thickness

shellsel <**cid** *cid*> <**id** *id*> <**ele**mtype *ename*> **no**des *nd1 nd2 nd3*

creates a shellSEL given an existing set of nodes. (To create a collection of shellSELs that lie upon existing zone faces, or to assign properties to shellSELs, use the **SEL shell** command.) The new shellSEL will not be attached to the grid; if you wish to attach it to the grid, then, after creating it, position its nodes with the **SEL node init** {**x**,**y**,**z**}**pos** command (which will create links and set appropriate attachment conditions for all nodes that are moved into a zone). The following keywords are available.

cid *cid*

specifies the component-ID number of the shellSEL. Each SEL has a unique component-ID number. If this value is not given, then the new shellSEL is assigned the next available component-ID number.

elemtype *ename*

specifies the finite element used by each shellSEL. If no element type is specified, then the element type will be DKT-CST. Note that the type of element associated with each shellSEL is set when the shellSEL is created and cannot subsequently be altered. The finite-element type of existing shellSELs is printed by the **PRINT sel shell elemtype** command. The value of ***ename*** must be one of the following 3-noded finite elements.

cst CST (6 degrees-of-freedom) finite element. This is a membrane element — it resists membrane but does not resist bending loading.

csth CST Hybrid (9 degrees-of-freedom) finite element. This is a membrane element — it resists membrane but does not resist bending loading.

dkt DKT (9 degrees-of-freedom) finite element. This is a plate-bending element — it resists bending but does not resist membrane loading.

dkt_cst DKT-CST (15 degrees-of-freedom) finite element. This is a shell element that combines the DKT and CST elements to resist both bending and membrane loading.

dkt_csth DKT-(CST Hybrid) (18 degrees-of-freedom) finite element. This is a shell element that combines the DKT and CST Hybrid elements to resist both bending and membrane loading.

id ***id***

specifies the ID number of the shellSEL. All SELs that comprise a particular entity will possess the same ID number — e.g., all shellSELs that comprise shell 6 will have an ID of 6. If this value is not given, then the new shellSEL is assigned an ID number of 0.

nodes ***nd1 nd2 nd3***

specifies the ID numbers (***nd1, nd2, nd3***) of three nodes that will define the shellSEL. These nodes must already exist — nodes can be created with the **SEL node** command. Ordering of the nodes defines the shellSEL coordinate system as follows. The nodes are oriented counterclockwise with respect to the positive z-direction, and the positive x-direction is parallel with the edge from ***nd1*** to ***nd2***.

SET keyword <keyword ***value*** . . . >

This command is used to set parameters in a *FLAC3D* model. The parameters are divided into two categories: control conditions and model conditions. The keywords available in these categories are summarized in Table 1.10, below.

Table 1.10 Summary of* SET *keywords

Control Conditions	Model Conditions
case	**creep**
cust1	**dynamic**
cust2	**fluid**
directory	**gravity**
echo	**large**
geom_rep	**mechanical**
geometry	**ratio**
hist_rep	**small**
log	**thermal**
logfile	
memory	
mouse	
movie	
output	
pagelength	
pagination	
pcxout	
pinterval	
plot	
safe	
track	

The keyword definitions are as follows.

case **off**
on

Case sensitivity in *FLAC3D* and in *FISH* is **off**, by default. To make *FLAC3D* and *FISH* case-sensitive, use the command **SET case on**.

creep keyword <keyword ***value*** . . . >

This command sets parameters for a time-dependent creep material analysis (only available with the creep model option — see Section 2 in **Optional Features**). The following keywords apply.

age ***t***

t is the creep time limit for the creep calculation using the **SOLVE** command.

dt ***t***
auto on/off

t defines the creep timestep. If not specified, the creep timestep will be set to the value given for **mindt**. The automatic calculation of creep timestep is turned on and off with the **auto on** and **auto off** keywords. By turning on this option, the timestep will be updated automatically. The automatic timestep calculation is controlled by the **SET creep** keywords: **lfob**, **ufob**, **latency**, **lmul**, **umul**. The default is **auto off**.

latency ***value***

The minimum number of creep timesteps that must elapse before the timestep is changed to ***value***. The default is ***value*** = 1.

lfob ***value***

The creep timestep will be increased if the ratio of the maximum unbalanced force to the average gridpoint force falls below ***value***. The default is ***value*** = 10^{-3}.

lmul ***value***

The creep timestep will be multiplied by ***value*** if the unbalanced force ratio falls below **lfob**. **lmul** must be greater than 1. The default is ***value*** = 1.01.

maxdt ***value***

The maximum creep timestep allowed is set to ***value***. The default is 10^{20} unless **mindt** is set, in which case **maxdt** = max (**mindt**, 10^{20}).

mindt *value*

The minimum creep timestep allowed is set to ***value***. The default is ***value*** = 10^{20}.

off The creep calculation process is turned **off**. The creep process is **on**, by default, when the **CONFIG creep** command is given. Specify **SET creep on** to start the creep calculation.

on The creep calculation process is turned **on**. The creep process is **on**, by default, when the **CONFIG creep** command is given. Specify **SET creep off** to suppress the creep calculation.

ufob *value*

The creep timestep will be decreased if the ratio of the maximum unbalanced force to the average gridpoint force exceeds ***value***. The default is ***value*** = 5×10^{-3}.

umul *value*

The creep timestep will be multiplied by ***value*** if the unbalanced force ratio exceeds **ufob**. **umul** must be less than 1. The default is ***value*** = 0.90.

step *value*

The maximum number of steps to be taken when the **SOLVE** command is issued is set to ***value***. (By default, unlimited stepping is allowed.)

time ***t***

Creep time is initialized to ***t***. This is useful if the problem time is different from the time the creep calculation is begun. The default is ***t*** = 0.

cust1 ***str1***

sets the first line of customer information, which is displayed in the lower-left corner of all plot views and in the HELP — ABOUT dialog box, to ***str1***. If this string contains spaces, then it must be enclosed by single quotes, as in ***'str1'***, in which case any occurrence of a single quote or a backslash character must be preceded by a backslash character.

cust2 ***str2***

sets the second line of customer information, which is displayed in the lower-left corner of all plot views and in the HELP — ABOUT dialog box, to ***str2***. If this string contains spaces, then it must be enclosed by single quotes, as in ***'str2'***, in which case any occurrence of a single quote or a backslash character must be preceded by a backslash character.

directory name

changes working directory to that specified by name.

dynamic keyword <keyword ***value*** . . . >

This command sets parameters for a dynamic analysis (only available with the dynamic model option — see Section 3 in **Optional Features**). Dynamic analysis parameters for the structural element logic are set separately via the **SEL set** command. The following keywords apply.

age ***t***

t is the dynamic time limit for the dynamic calculation using the **SOLVE** command.

damping keyword

This command selects the damping type for the dynamic analysis. (Damping is described in Section 3 in **Optional Features**.) The following keywords apply.

avisc ***an al***

artificial viscosity. This damping only applies to the main grid.

combine <***value***>

combined local damping (default for creep modeling). The damping ***value*** is 0.8, by default.

local <***value***>

local damping. The damping ***value*** is 0.8, by default.

rayleigh *frac freq* <**mas**s> <**st**iffness>

For dynamic calculations, a certain fraction of critical damping is usually required over a given frequency range. This type of damping is known as Rayleigh damping, where ***frac*** = the fraction of critical damping operating at center frequency of ***freq***. (NOTE: Input frequencies for the program are in cycle/sec or Hertz — *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 left out, 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 mesh deformation occurs. In such a case, lower the timestep with the **SET dynamic dt** command.

dt ***t***

The value ***t*** defines the dynamic timestep. By default, *FLAC3D* calculates the dynamic timestep automatically for the explicit solution scheme. This keyword allows the user to choose a different timestep. If *FLAC3D* determines that the user-selected value is too large for numerical stability, the timestep will be reduced to a suitable value when dynamic calculation steps are taken. The calculation will not revert to the user-selected value until another **SET dynamic dt** command is issued.

multistep **off**
on

turns multi-stepping on or off. Multi-stepping speeds up calculations in dynamic models which have a large zone size or modulus contrast. Areas of the grid with critical timesteps greater than the global critical timestep are updated less frequently, thus saving execution time. The logic is general in the sense that *all* zones, gridpoints and structures are included. The

user does not need to do anything beyond switching it on.

Sub-stepping only works when dynamic mode is in operation (**SET dyn on**), and is effective only when the grid is nonuniform or there is a contrast in material properties. In this case, zones and gridpoints are each given a multiplier (which can be determined with a *FISH* function) that is used to control the frequency of respective calculations. For example, if a zone's multiplier is 4, then that zone's calculations will be done at every fourth timestep. Multipliers are integers, and are powers of two (2, 4, 8, 16, etc.).

When the usual wavelength criterion (10 times zone size) is obeyed, the results with sub-stepping are almost identical to those without, except for an increase in calculation speed.

off turns the dynamic calculation process **off**. The dynamic process is **on**, by default, when the **CONFIG dynamic** command is given. Specify **SET dynamic on** to turn the dynamic calculation on.

on turns the dynamic calculation process **on**. The dynamic process is **on**, by default, when the **CONFIG dynamic** command is given. Specify **SET dynamic off** to suppress the dynamic calculation.

time ***t***

Dynamic time is initialized to ***t***. This is useful if the problem time is different from the time the dynamic calculation is begun.

echo **off**
on

SET echo on causes *FLAC3D* to echo lines when the lines come from a **CALL**ed file, or from a *FISH* **COMMAND – ENDCOMMAND** section, to the screen and to the log file (if it is open). This is the default setting. **SET echo off** prevents the input lines from being displayed; the lines are still written to the log file.

fluid keyword <keyword ***value***> . . .

This command sets parameters for a fluid-flow analysis (see Section 1 in **Fluid-Mechanical Interaction**). The following keywords apply.

age ***t***

The value ***t*** is the fluid-flow time limit for the fluid-flow calculation using the **SOLVE** command.

biot **off**
on

The fluid-flow calculation uses the Biot coefficient, α, and the Biot modulus, M, if **on**. If **off**, then the fluid modulus, K_f, and porosity, n, are used and $\alpha = 1$. The default is **off**.

dt ***t***

The value ***t*** defines the fluid-flow timestep. This *must* be specified for the implicit solution scheme. By default, *FLAC3D* calculates the fluid-flow timestep automatically for the explicit solution scheme. This keyword allows the user to choose a different timestep. If *FLAC3D* determines that the user-selected value is too large for numerical stability, the timestep will be reduced to a suitable value when fluid-flow steps are taken. The calculation will not revert to the user-selected value until another **SET flow dt** command is issued.

implicit **off**
on

The implicit solution scheme in the fluid flow model is turned **on** or **off**. The default is **off**.

off The fluid flow calculation may be turned **off** for a mechanical-only calculation or a thermal-only calculation.

on The fluid flow process is **on**, by default, when the **CONFIG fluid** command is given.

ratio *value*

The fluid-flow ratio limit is set to ***value*** for the **SOLVE** command. By default, **ratio** is defined as the maximum unbalanced flow magnitude for all the gridpoints in the model divided by the average applied flow magnitude for all the gridpoints. (Different forms of **ratio** can be specified; see the **SET ratio** command.) When the ratio falls below ***value*** during the calculation process, the fluid-flow calculation will stop. By default, the ratio limit is set to 1.0×10^{-5}.

step *value*

value is the timestep limit when the **SOLVE** command is issued. (By default, unlimited stepping is allowed.)

substep *value* <**au**to>

The maximum number of fluid-flow sub-steps in a coupled fluid flow-mechanical calculation, or a coupled thermal-mechanical-fluid flow calculation, is set to ***value***. (The default is ***value*** = 1.)

The fluid calculation is identified as the slave component in the fluid-flow mechanical process or in the thermal-mechanical fluid-flow process when the optional keyword **auto** is given.

geom_rep ***n***

Geometry terms for tetrahedral sub-zones are updated every ***n*** steps (default ***n*** = 10). For rapid rotation under large-strain mode, ***n*** should be set to a low number (which gives greater accuracy but slower calculations).

geometry *value*

This command protects against an illegal geometry error in large-strain mode. The code signals an illegal geometry error if the volume of any tetrahedral sub-zone becomes less than zero. The user input ***value*** is the ratio of the volume of a tetrahedral sub-zone to the zone volume. If a value greater than zero is specified for the ratio, then the code will exit cycling and not signal an error. In this way, a run can be made to determine a failure state without a bad geometry error occurring. (Note: Be aware that zone geometry is checked only every 10 steps in large-strain mode. So, if the tolerance is set too close to zero, an error may be signaled before exiting at the selected ratio. Also note that the typical minimum sub-zone volumes are 0.2

of the full-zone volume. So, the maximum value selected for the ratio should be less than 0.2 to avoid premature termination of cycling.) By default, ***value*** = 0.0.

gravity *<**gx** **gy**> **gz***

Gravitational accelerations are specified for the *x*-, *y*- and *z*-directions. If only one value is given, (0, 0, ***-value***) is assumed.

hist_rep ***n***

Histories are sampled every ***n*** timesteps. The default is ***n*** = 10. The synonym **HISTORY nstep** may also be used.

large large-strain mode (coordinates are updated) — also applies to the structural element logic.

log **off**
on

on opens an ASCII file "FLAC3D.LOG" on the default disk drive. If a file "FLAC3D.LOG" already exists, the option is given to overwrite or append to the existing file. Any text that is displayed to the screen from this point on is written to the log file. This is particularly useful for keeping a record of interactive sessions. The file may be edited to create batch data files.

off turns off the logging function. It does not close the log file. If **SET log on** is given at some later stage in the session, subsequent screen output will be appended to the file.

logfile filename

The log filename can be specified directly by the user. The log file must still be turned **on** to activate writing.

mechanical keyword <keyword ***value***> . . .

This command sets parameters for a static mechanical analysis. The following keywords apply.

damp keyword

This command selects the damping type for the static, mechanical process. The following keywords apply.

combined <***value***>

combined local damping (default is 0.8).

local <***value***>

local damping (default is 0.8). This is the default damping mode.

force ***value***

The out-of-balance force limit is set to ***value*** for the **SOLVE** command. When the maximum out-of-balance force falls below this limit, the mechanical calculation will stop. (By default, the out-of-balance force limit is zero.)

off The mechanical calculation may be turned **off** for a thermal-only calculation or a fluid flow-only calculation.

on The mechanical process is **on**, by default.

ratio ***value***

The force ratio limit is set to ***value*** for the **SOLVE** command. By default, **ratio** is defined as the maximum unbalanced force magnitude for all gridpoints in the model divided by the average applied force magnitude for all the gridpoints. (Different forms of **ratio** can be specified; see the **SET ratio** command.) When the ratio falls below ***value*** during the calculation process, the mechanical calculation will stop. By default, the ratio limit is set to 1.0×10^{-5}.

step ***value***

The maximum number of steps to be taken when the **SOLVE** command is issued is set to ***value***. (By default, unlimited stepping is allowed.)

substep ***value*** <**au**to>

The maximum number of mechanical sub-steps in a coupled fluid flow-mechanical calculation or a coupled thermal-mechanical calculation is set to ***value***. (The default is ***value*** = 100.) The mechanical calculation is identified as the slave component in the fluid flow-mechanical process or in the thermal-mechanical process when the optional keyword **auto** is given.

memory keyword

controls memory-allocation system used for model memory. By default, memory management occurs automatically. Memory is allocated from the operating system and deallocated (or returned to the operating system) based upon model requirements. In most cases, the automatic behavior will be adequate; however, in certain special cases, one may wish to modify the memory-allocation behavior using the following keywords.

add ***m***

requests an additional ***m*** megabytes of memory from the operating system and makes this available to the model.

lock **off**
on

If **lock** is set to **on**, then no more memory will be made available to the model; if the model requires more memory than has been allocated, an error condition will be triggered. By default, **lock** is equal to **off**.

size ***b***

sets the allocation-block size equal to ***b*** megabytes. When the model requires more memory, a memory block of size ***b*** is requested from the operating system. By default, ***b*** is equal to 1.0 megabyte.

mouse ***n***

The movement of a mouse can be reduced by increasing the value of ***n***. By default, ***n*** = 1.0.

movie **avi** <keyword>
dcx <keyword>

This command sets the graphics hardcopy device characteristics for movies (also see the **MOVIE** command). Either the AVI or DCX file type may be specified. Available keywords are:

file fname

sets the name of the file to receive screen images (defaults to “FLAC3D.DCX” or “FLAC3D.AVI” if not specified).

size ***iw***, ***ih***

specifies the size (width = ***iw*** and height = ***ih***) of a single image in pixels. The default size is (***iw,ih***) = (640,480). To make the movie images appear the same relative size as those appearing in a plot window, keep the ratio of ***iw*** to ***ih*** the same as the aspect ratio of the plot window (i.e., if the plot window is twice as wide as it is tall, then make ***iw*** two times ***ih*** in the **size** keyword).

step ***n***

A new screen plot is added to the movie file every ***n*** steps. This allows movie frames to be saved without interrupting a **CYCLE** or **SOLVE** command. The plot generated is the same as the current plot (see the **PLOT current viewid** command).

frameperiod ***n***

sets the length of time (in milliseconds) that a frame is displayed before the next frame appears (this keyword only applies to AVI files and does not apply to DCX files). A higher number implies a slower, longer-playing movie. The default is 100 milliseconds (this can be overridden with the AVI player options).

output port

sends plotted output to the device connected to port, in which 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 the named file that can be specified by the user. The default is to send output to file "FLAC3D.PS."

pagelength ***n***

The number of lines displayed per page can be changed to ***n***. The default is ***n*** = 25.

pagination **off**
on

If **off**, text pagination on the screen is turned off. The default is **on**.

pcxout filename

sets filename for PCX output generated by the <F2> keystroke in graphics screen mode. (The default filename is "FLAC3D.PCX.")

pinterval ***n***

The number of cycles calculated between screen refreshes in plot mode while cycling is set to ***n*** (default is ***n*** = 1).

plot keyword

controls the type of graphics hardcopy output (see the **PLOT hard** command). Output types include: Windows printer, Windows enhanced metafile, bitmap, JPEG, PCX and PostScript. The default output type is a Windows printer. The type, and settings for each type, are specified with the following keywords.

bitmap <**si**ze ***iw ih***>

sets the target resolution of the Windows bitmap to ***iw*** by ***ih*** pixels. By default, ***iw*** and ***ih*** are set equal to 1024 and 768 pixels, respectively.

bmp **si**ze ***iw ih***

sets the target resolution of the Windows bitmap to ***iw*** by ***ih*** pixels. By default, ***iw*** and ***ih*** are set equal to 1024 and 768 pixels, respectively.

jpg <**qual**ity ***iq***>

iq sets the quality of the JPG output plot
2 — maximum quality (least compression)
255 — minimum quality (maximum compression)
By default, ***iq*** is set equal to 50.

size ***iw ih***

sets the target resolution of the JPEG format output to ***iw*** by ***ih*** pixels. By default, ***iw*** and ***ih*** are set equal to 1024 and 768 pixels, respectively.

metafile <keyword>

The graphics hardcopy device is set to be a Windows enhanced metafile. The settings can be modified with the following optional keyword.

size ***iw ih***

sets the target resolution of the Windows enhanced metafile to ***iw*** by ***ih*** pixels. By default, ***iw*** and ***ih*** are equal to 1024 and 768 pixels, respectively.

pcx <**si**ze *iw ih*>

sets the target resolution of the PCX format output to ***iw*** by ***ih*** pixels. By default, ***iw*** and ***ih*** are set equal to 1024 and 768 pixels, respectively.

postscript <keyword>

The graphics hardcopy device is set to be a PostScript file. The settings can be modified with the following optional keywords.

a3 sets page size to ISO A3 standard 297 × 420 mm.

a4 sets page size to ISO A4 standard 210 × 297 mm.

color sets output mode to be color.

custom ***w h***

sets page size to user-defined ***w*** × ***h*** inches.

gray sets output mode to be black and white. (This is the default output mode.)

landscape sets page orientation to landscape mode. (This is the default page orientation.)

letter sets page size to US standard 8.5 × 11 inches. (This is the default page size.)

portrait sets page orientation to portrait mode.

position ***ox oy***

sets the location of the lower left-hand corner of the image to be at an x- and y-offset of ***ox*** and ***oy*** inches, respectively, from the lower left-hand corner of the page. By default, both ***ox*** and ***oy*** are equal to 0.5 inches.

size ***w h***

sets the width and height of the image, as it will appear on the page, to ***w*** and ***h*** inches, respectively. By default, ***w*** and ***h*** are equal to 10.0 and 7.25 inches, respectively.

tabloid sets page size to US standard — 11 × 17 inches.

windows <keyword>

The graphics hardcopy device is set to be a Windows printer. The settings can be modified with the following optional keywords.

size ***w h***

sets the width and height of the image, as it will appear on the page, to ***w*** and ***h*** inches, respectively. By default, ***w*** and ***h*** are determined automatically and set equal to the size of the paper to which one is printing.

position ***ox oy***

sets the location of the lower left-hand corner of the image to be at an x- and y-offset of ***ox*** and ***oy*** inches, respectively, from the lower left-hand corner of the page. By default, both ***ox*** and ***oy*** are equal to 0.5 inches.

ratio keyword

The ratio limit for mechanical, thermal and fluid-flow calculations using the **SOLVE** command can be calculated in three ways, as defined by the following keywords.

average The ratio is defined to be the average unbalanced mechanical force (or heat-flux or fluid-flow) magnitude for all the gridpoints in the model divided by the average applied mechanical force (or heat flux or fluid flow) magnitude for all the gridpoints (default).

local The ratio is defined to be the maximum value of the ratio of the unbalanced mechanical force (or heat-flux or fluid-flow) magnitude to the applied mechanical force (or heat-flux or fluid-flow) magnitude for all the gridpoints in the model.

maximum The ratio is defined to be the maximum unbalanced mechanical force (or heat-flux or fluid-flow) magnitude for all the gridpoints in the model divided by the average applied mechanical force (or heat flux or fluid flow) magnitude for all the gridpoints.

In increasing order of stringency, the conditions are:

average ensures a majority of zones are in equilibrium.

maximum ensures that all unbalanced forces are below a certain value, compared to average forces for the whole model.

local ensures that each local unbalanced force is less than some fraction of the local forces, averaged for each gridpoint. This criterion can be misleading if some zones have very small stresses, because even a large ratio for one gridpoint may not be very important given that the associated zone contributes almost nothing to the overall response.

safe **off**
on

If **on**, *FISH* variables must be prefixed with an @ sign and macro variables with a # symbol. These symbols unambiguously identify the variables as being *FISH* or macro names, avoiding conflicts with reserved words. The symbols are ignored when **safe** is **off**. The default is **off**.

small small strain (coordinates are not updated); this is the default.

thermal keyword <keyword ***value***> . . .

This command sets parameters for a thermal analysis (only available with the thermal model option — see Section 1 in **Optional Features**). The following keywords apply.

age ***t***

The value ***t*** is the thermal "heating time" limit for the thermal calculation using the **SOLVE** command.

dt ***t***

The value ***t*** defines the thermal timestep. This timestep *must* be specified for the implicit solution scheme. By default, *FLAC3D* calculates thermal timestep automatically for the explicit solution scheme. This keyword allows the user to choose a different timestep. If *FLAC3D* determines that the user-selected value 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 thermal dt** command is issued.

implicit **off**
on

The implicit solution scheme in the thermal model is turned **on** or **off**. The default is **off**.

off The thermal calculation may be turned **off** to do a fluid flow or a mechanical calculation only.

on The thermal process is **on**, by default, when the **CONFIG thermal** command is given.

ratio ***value***

The heat-flux ratio limit is set to ***value*** for the **SOLVE** command. By default, **ratio** is defined as the maximum unbalanced heat-flux magnitude for all the gridpoints in the model divided by the average applied heat-flux magnitude for all the gridpoints. (Different forms of **ratio** can be specified; see the **SET ratio** command.) When the ratio falls below ***value*** during the calculation process, the thermal calculation will stop. By default, the ratio limit is set to 1.0×10^{-5}.

step ***value***

The maximum number of steps to be taken when the **SOLVE** command is issued is set to ***value***. (By default, unlimited stepping is allowed.)

substep ***value*** <**au**to>

The maximum number of thermal sub-steps in a coupled calculation is set to ***value***. (The default is ***value*** = 100.) The thermal calculation is identified as the slave component in the thermal-mechanical process or in the thermal-mechanical-fluid flow process when the optional keyword **auto** is given.

track **off**
on

Turns particle tracking on or off. Particle tracking is off by default. See the related **TRACK** command.

SOLVE <keyword ***value***> <keyword ***value***> . . .

This command controls the automatic timestepping for the following calculation processes:

mechanical static
mechanical creep
mechanical dynamic
fluid-flow
thermal
coupled thermal mechanical
coupled fluid-flow mechanical
coupled thermal mechanical fluid-flow

For mechanical static calculations, the steady-state solution is detected, by default, when the unbalanced force ratio reaches 1×10^{-5}. (This can be changed with the **SET mechanical** command.) For transient analyses, a calculation is performed until the limiting conditions, as defined by the following keywords, are reached.

age ***t***

In **CONFIG creep** mode, ***t*** is the creep time limit for the mechanical creep calculation (only available with the creep model option — see Section 2 in **Optional Features**).

In **CONFIG dynamic** mode, ***t*** is the dynamic time limit for the dynamic calculation (only available with the dynamic analysis option — see Section 3 in **Optional Features**).

In **CONFIG fluid** mode, ***t*** is the fluid-flow time limit for the fluid flow calculation. (See Section 1 in **Fluid-Mechanical Interaction**.)

In **CONFIG thermal** mode, ***t*** is the thermal "heating time" limit for the thermal calculation (only available with the thermal option — see Section 1 in **Optional Features**).

For coupled processes, ***t*** is the maximum time of all processes involved.

NOTE: The units for time, ***t***, will depend on the input properties.

clock ***t***

The value ***t*** is the computer runtime limit, in minutes. By default, there is no limit on computer runtime.

elastic performs a mechanical calculation in two steps: first, assuming elastic behavior; and then using the actual strength values of the material. The cohesion and tensile strength for all materials in the model are set to high values for the first step. (At present, only Mohr-Coulomb materials can be present in the model when using **SOLVE elastic**.) For the second step, the cohesion and tensile strength are reset to their original values.

fishhalt ***FISH function***

The specified *FISH* function is called during every iteration of the solve loop. If the *FISH* function evaluates to 0 (zero), then solving continues; otherwise, solving terminates.

force ***value***

out-of-balance force limit (by default, ***value*** = 0)

fos <keyword> . . .

performs an automatic search for factor of safety. The procedure is described in Section 3.8 in the **User's Guide** (Note 12), and applies only when the Mohr-Coulomb model is installed in all non-null zones.

The following keywords apply:

associated associated flow rule is applied, whereby the dilation angle is set equal to the friction angle, as modified by the **fos** procedure. (By default, the non-associated flow applies.)

file filename

sets the filename of the save file produced for the last non-equilibrium state. (By default the save file is named "fosmode.fsv.")

include keyword . . .

exclude keyword . . .

Various items are included or excluded from consideration during the **fos** search, according to the list of keywords that follow **include** or **exclude**, respectively. The following keywords are allowed, denoting items that may be modified during the **fos** search.

cohesion cohesion

friction friction angle

interface interface friction and cohesion

tension tension cutoff

By default, friction and cohesion are *included*, and tension and interfaces are *excluded*.

ratio ***value***

ratio limit for the active calculation process (by default, the limit is 1.0×10^{-5})

step ***value***

total step limit (by default, no limit is placed on the stepping)

STEP ***n***

executes ***n*** timesteps. If the <Esc> key is pressed during execution, *FLAC3D* will return control to the user after the current step is completed. The synonym **CYCLE** may also be used. Cycling may be interrupted by pressing the <SPACEBAR> or the <Esc> key. When in batch mode, the <SPACEBAR> will cause *FLAC3D* to stop cycling and skip to the next data line, whereas the <Esc> key will abort cycling and abort reading the data file.

For coupled processes, each step corresponds to one sub-step for each of the processes that are **on**. Timesteps are synchronized only when and for those processes turned **on** simultaneously.

STOP *FLAC3D* stops. Note that *all* information generated while in *FLAC3D* will be lost unless a **SAVE** command is issued prior to the **STOP** command. The synonym **QUIT** may also be used.

SYSTEM

The **SYSTEM** command spawns a DOS command session while keeping *FLAC3D* in the background. Any DOS command or utility may be used, and a completely different program may be run (memory permitting). There must be enough system memory available (approximately 600 KB) *after FLAC3D* has allocated its reserved memory for this command function. Otherwise, the DOS error message, "unable to execute COMMAND.COM," will result when the **SYSTEM** command is issued. Reload *FLAC3D* with a smaller amount of RAM to leave sufficient space for system memory. The DOS environment used will be the same as that which executed *FLAC3D*. Type <EXIT> to return to *FLAC3D*.

Note that the DOS command shell, "COMMAND.COM," must be available on the DOS path. The environment variable COMSPEC is not used.

An alternative form of the **SYSTEM** command is to issue the command followed by a DOS command on the same line. For example,

```
sys dir *.dat
```

will list the files with the extension "DAT." In this case, you will return to *FLAC3D* after the DOS command is completed. The following DOS commands may be given following the **SYSTEM** command:

CD	directory path
COPY	file1 file2
DEL	file
DIR	<file-spec>
REN	file1 file2
TYPE	file

CAUTION: Do not use these commands with files *FLAC3D* currently has open, such as the file to which plots are being sent. If you wish to copy such a file, first close it. If these commands attempt to access open files, the system may hang up and the files may be lost.

TABLE ***n*** <keyword> ***x1 y1*** <***x2 y2***> <***x3 y3***> . . .

This command sets up a table of x- and y-values for use by *FLAC3D*. Tables are used to define:

(1) the variation of friction, cohesion and dilation with accumulated plastic shear strain for the strain-softening model (see the **PROP ftable**, **PROP ctable** and **PROP dtable** commands) or the variation of tensile strength with accumulated plastic tensile strain (see the **PROP ttable** command); and

(2) histories of boundary conditions (see the **APPLY** command).

Multiple tables may be defined, each uniquely identified by a table number ***n***. The number of tables and x,y pairs are only limited by the amount of computer memory available. The command **PRINT table** can be used to verify the contents of a table. Individual tables can also be plotted using the **PLOT table *n*** command. Table numbers need not be sequential, but they must not be zero.

Tables can also be created and manipulated with *FISH* functions. See Section 2 in the ***FISH* volume** for a description of *FISH* table functions.

Histories may also be copied to tables (see the **HISTORY** command). This allows the user to perform post-processing on histories (by using *FISH*).

Optional keywords may be used to manipulate table contents. The available keywords are:

erase — erases all entries in table ***n***.

insert — One or more x,y 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 x-value of the new item. If an x-value for a new item is identical to that of an existing item, the existing item's y-value is updated (in **insert** mode).

name — 'string'

changes the name of table number ***n*** to 'string'. The table ID number is not changed.

position — ***i x y***

positions the next entry at the ***i***th position in the table. (This can be used to create dummy (0,0) entries if needed.)

read filename

reads file, filename (in the format described below), and places it in table ***n***.

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

Line 1 heading of up to 20 full words, four characters per word

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

Line 3 through Line ***np***+2 — ***np*** real values of the table y-value. The x-value is calculated as equally spaced values of time at intervals of ***tdel***. If ***tdel*** is specified as 0.0, then it is assumed that the timestep is not constant and the values for Line 3 through Line ***np***+2 are read as pairs.

sort This keyword performs the same function as **insert**, and causes all subsequent entries to be sorted in order of increasing x-value. This keyword also sorts all values previously entered in table ***n***.

TITLE <'string'>

The title for a *FLAC3D* model is printed on subsequent output plots and recorded in the save files.

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 `Title>` 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. Pressing <F3> will cause the old title to fill the command line for editing. To clear the title (causing the title window to disappear on output plots), give a null string (two single quotes with a space in between) with the **TITLE** command:

```
title ' '
```

TRACK x y z . . . <keyword>

The **TRACK** command places a particle in the grid at the location specified. The particle flows with the fluid and maintains a record of its path over a finite period of time. The grid must be configured for fluid flow (**CONFIG fluid**), and the zone must have a non-null fluid model in order for a particle to exist in a zone. The particle is not created (and no error message is output) if the location does not map into a zone with these specifications. The particles have no mass and do not influence the flow or mechanical behavior in any way; they are simply carried along by the fluid and record its motion. Several **TRACK** commands may be given, at different times, if desired. Location and keywords are given as follows:

The location of the particles can be specified as a single point, points on a **line**, or points on a **plane**.

track x0 y0 z0

Creates a single particle at ***x0 y0 z0***.

line ***x0 y0 z0 x1 y1 z1 n***

This keyword causes ***n*** particles to be introduced into the grid. The particles are evenly spaced along a line from ***(x0, y0, z0)*** to ***(x1, y1, z1)***.

plane ***x0 y0 z0 x1 y1 z1 x2 y2 z2 n0 n1***

This keyword causes ***n0*** $\times$ ***n1*** particles to be introduced into the grid. ***n0*** particles are evenly distributed on a line with endpoints ***(x0, y0, z0) (x1, y1, z1)***, and ***n1*** particles are evenly distributed on a line with endpoints ***(x0, y0, z0) (x2, y2, z2)***. Particle positions are placed on a regular grid formed by the two lines.

tortuosity ***t*** The tortuosity is specified as ***t***, which is defined as the ratio of the straight line path to the actual fluid path through a zone. This factor will be used to multiply track increments calculated from the flow velocity; it defaults to 1 if not given.

notail If **notail** is specified, then the particle does not store information about its flow path. Only its current location is stored.

list A summary of particles is printed: current position; zone into which the particle maps; and accumulated path length. The synonym **PRINT track** may also be used.

dump Coordinates of all points along all particle tracks are output to the screen. Note that a new coordinate pair is only saved if the particle moves more than a certain distance from its old position (one quarter of the length of the largest diagonal in a zone). Each line consists of four numbers: ***time***, x, y and z.

write is the same as **dump**, but the information is written to the file "FLAC3D.TRK." This ASCII file may then be imported to a graph-plotting program for manipulation and plotting.

The following commands turn particle tracking on and off.

SET track on
SET track off

Particle tracking is off by default and must be turned on for particles to move.

See related commands **PDELETE** and **PLOT track**.

WATER keyword ***value*** <keyword ***value***> . . . <**ra**nge . . . >

This command assigns fluid-flow conditions and properties for an effective stress calculation. *FLAC3D* uses effective stresses (i.e., total stresses plus pore pressure) in constitutive models. Pore pressures are defined at gridpoints, and zone pore pressures are then calculated as the average of the zone gridpoints. If **CONFIG fluid** mode is not set, pore pressures are not affected by zone volume changes, there is no flow of water, and the dry density must be specified for zones above the water table and the saturated material density for zones below.

The following keywords apply:

density ***value***

fluid density, ρ_w [SI units: kg/m^3]

table keyword ***value*** . . .

The **WATER table** command sets pore pressure for all gridpoints (and, hence, zones) below the water table. The pore-pressure gradient is given by the direction of the gravity vector, which can be arbitrary (see **SET gravity** command).

The water table plane can be defined in two forms: a single infinite plane; or an assembly of planar, convex polygons. For an infinite plane, the following keywords are used.

normal ***nx ny nz***

normal direction to the plane, defined by unit vector ***nx***, ***ny***, ***nz*** and pointing in the direction of increasing pore pressure.

origin ***x y z***

one point at coordinate location (***x, y, z***) on the plane

Alternatively, the water table can be defined by an assembly of planar, convex polygons. The following keyword phrase applies.

face ***x1, y1, z1*** . . . ***xn, yn, zn*** <**fa**ce . . . >

The **face** polygon is defined by nodes ***x1, y1, z1*** to ***xn, yn, zn***. The nodes *must* be coplanar and produce a convex polygon. Faces can have any number of nodes but are split into triangles for storage. Only gridpoints that project along the gravity direction "inside" faces are assigned pore pressure. No checking of face overlapping or intersection is performed.

The water table can be removed with the following command.

clear clears water table

ZONE {**i**d = *zid*} {**b**rick **w**edge **p**yramid **d**brick **t**etra} {**i**d = *gpid* or *x y z*}

This command creates a single new zone of the specified type using existing gridpoints. (For example, if gridpoints with IDs of 1 through 8 exist, then the command

```
zone id = 21 brick id 1 id 2 id 3 id 4 id 5 id 6 id 7 id 8
```

would create a new brick-type zone with an ID of 21.

If the zone ID (*zid*) parameter is missing, then the new zone is given the next available zone number. For example,

```
zone brick id 1 id 2 id 3 id 4 id 5 id 6 id 7 id 8
```

See Figure 1.1 for the zone geometry specifications. **brick** is the default zone type.

Instead of specifying the gridpoint ID, the *x, y, z* coordinates of the gridpoint can be specified. The gridpoint nearest to the specified *x, y, z* will be used. Any combination of coordinates and gridpoint IDs can be used with this command. For example,

```
zone id 72 tetra id 5 3.45 1.02 7.04 id 7 2.10 8.03 4.07
```

The above command specifies a **tetra** zone with two gridpoint IDs and two pairs of gridpoint *x y z* coordinates.

Also see the **GP** command.