APES$^+$

(Analysis Program for Earth Structures)

Computer Program Documentation

User Manual

Victor N. Kaliakin

February 6, 2023

Contents

Chapter 1

General Information

The APES$^+$ computer program[1] has been written to perform one-, two-, or three-dimensional static and quasi-static analyses of continua. For two-dimensional analyses, the solution domain can be idealized assuming conditions of plane strain, plane stress, or torsionless axisymmetry. Since the formulation used in the program takes into account the coupling between stress and flow, APES$^+$ is particularly well-suited for the analysis of earth structures. It is important to note, however, that the program is written in such a manner as to avoid unnecessary computations when this coupling is not desired in an analysis. In short, the efficiency of the program is *not* compromised in such a case.

With the exception of lexing[2] and parsing[3] software developed elsewhere, the APES$^+$ computer program is written in Fortran 95 [32], thus making use of the advances offered by the language. In keeping with modern programming practice, the design of the program is quite modular and facilitates the incorporation of additional solution algorithms, equation solvers, element types and of constitutive models.

This documentation assumes general familiarity with the finite element method; novices in this area are referred to standard texts on the subject e.g., [5; 44; 52; 117].

1.1 Historical Note

APES$^+$ grew out of the SAC-2 and SAC-3 computer programs. The original versions of these programs were written at the University of California Davis in 1983 [41; 42; 88]. A FORTRAN-77 version of the SAC-2 program, which improved the input and output options and which incorporated additional material models, followed [40]. This was followed by enhanced two- [50] and three-dimensional [51] versions of the program.

The present version of the APES$^+$ computer program has the following features:

[1] APES = Analysis Program for Earth Structures.

[2] Lexing, lexical analysis or tokenization is defined as the process of converting a sequence of characters into a sequence of tokens (strings with an assigned and thus identified meaning). A program that performs lexical analysis may be termed a lexer. A lexer is generally combined with a parser, which together analyze the syntax of programming languages, web pages, and so forth.

[3] Parsing, syntax analysis, or syntactic analysis is the process of analysing a string of symbols, either in natural language, computer languages or data structures, conforming to the rules of a formal grammar.

- The program is written in Fortran 95 [32], thus taking advantage of features of the language which lead to more efficient code.
- To facilitate program input and output, a simple problem-oriented language (POL) approach [58] is used in APES$^+$. The model generation phase of an APES$^+$ analysis is sufficiently flexible so as to easily accept data generated by stand-alone mesh pre-processors (e.g., [90]).
- An enlarged library of element types has been developed. This includes one-dimensional (bar and beam) elements, zero-thickness interface elements, and irreducible and mixed two- and three-dimensional continuum elements with both discontinuous and continuous pressure approximations.
- The number of material idealizations has been increased.
- The highly efficient SKYLDU skyline equation solver [94] for symmetric and/or non-symmetric systems has been implemented.
- Greater flexibility in displaying the results of analyses has also been incorporated into the program.

Chapter 2

Information Regarding Program Input

2.1 General Comments

Finite element analyses generally require large amounts of input data. The preparation of this data should be as easy as possible and should leave little room for error. In keeping with these concerns, the input associated with the APES$^+$ computer program has been designed for ease of use, flexibility, and consistency. Further details regarding the program input are given in the subsequent sections of this part of the documentation.

2.2 Developing Mathematical Models for APES$^+$

The greater availability of fast and relatively inexpensive computational resources has allowed engineers and physicists to analyze larger and more sophisticated problems. In the context of finite element analyses, this usually translates into the development of larger models with increasingly complex shapes and high levels of mesh refinement. As the models studied increase in complexity and size, more and more effort is spent on the development of the mathematical model prior to analysis; i.e., on the pre-processing phase. Likewise, a substantial amount of time is also spent in the post-processing phase viewing and interpreting the typically large body of results generated by the finite element analysis.

In appreciation of the need for powerful yet easy to use pre- and post-processing software, a modular approach to finite element analyses has developed over time. In particular, general "stand-alone" pre- and post-processors have been written to complement specialized analysis programs. Such a modular design is founded on the rationale that each phase of the analysis process should draw upon the expertise of a specific software development team with a specific design objective.

The pre-processor should be an efficient, powerful, yet flexible program that completely defines the geometry and topology (i.e., element connectivity) of the mathematical model, assigns material types to groups of elements, and possibly assigns nodal specifications. Ideally the pre-processor possesses robust graphical capabilities that can be displayed on all popular computing platforms (possessing reasonable computational resources, of course).

The end product of the pre-processor is a file(s), with known format, that contain all of the aforementioned data. Before leaving the subject of pre-processors, it is timely to note that the rise in popularity of computer-aided design (CAD) and computer-aided design and drafting (CADD) application software has led to their use in developing finite element meshes. Although this use of CAD and CADD software is still somewhat in its infancy, further expansion in this area is likely. It is important to note, however, that pre-processors typically cannot generate all the data required to describe a mathematical model for a given analysis. Consequently, the analyst must, typically via a text editor, modify the data generated by the pre-processor.

The purpose of the analysis program, or "computing engine," is to analyze a mathematical model. Ideally, the analysis program should not have to do any development of the mathematical model, for this should have all been done by the more efficient pre-processor. Likewise, the analysis program should not be assigned the task of graphically displaying results. This task is relegated to the more efficient standalone post-processor. Thus, the analysis program simply prompts for the name of a file(s) containing the description of a complete mathematical model, and for the name of a file(s) into which results should be written.

Finally, the purpose of the post-processor is to conveniently display the results of the analysis for scrutiny by the analyst. Typically this entails the use of interactive graphics. Similar to the pre-processor, the post-processor must be an efficient, powerful, yet flexible program. It must afford the user ample opportunity to ascertain the results of the finite element analysis. Examples of graphical capabilities include deformed mesh plots, contour plots, time-history plots, etc.

The only drawback of the above "modular" approach to finite element analysis is the disparity of formats associated with the input/output files created in each phase of the analysis. The remedy to this problem are file translators. A translator is a piece of software whose sole task is to read a data file, with known format, and to write (translate) this data into another format. Typically, a translator is written for a specific *pair* of programs; i.e., a program whose output will serve as input to the translator, and a program whose input will be the output of the translator. Ideally translation should be done automatically, with minimal input from the user. The role of translators in the "modular" analysis procedure is summarized in Figure 2.1.

The "pre-processor to analysis" translator reads the output from a specific pre-processor. It then proceeds to convert this data into the format recognized by the analysis program. In a similar fashion, the "analysis to post-processor" translator reads the output of the analysis program, and converts this data into a format recognized by a specific post-processor. In recognition of the fact that numerous stand-alone input pre-processors have, in recent years, been developed, the process of creating mathematical models for APES+ has been designed in such a manner as to easily accept data that has been suitably translated from stand-alone pre-processors.

In recognition of the fact that such pre-processors are not available to all analysts (if for no other reason than they tend to be rather expensive), rather simple mesh and data generation capabilities have been provided in APES+. Consequently, a complete mathematical model can be developed entirely within APES+, or another pre-processor can be used to generate portions of the model. The latter (and typically preferable) mode of operation is predicated

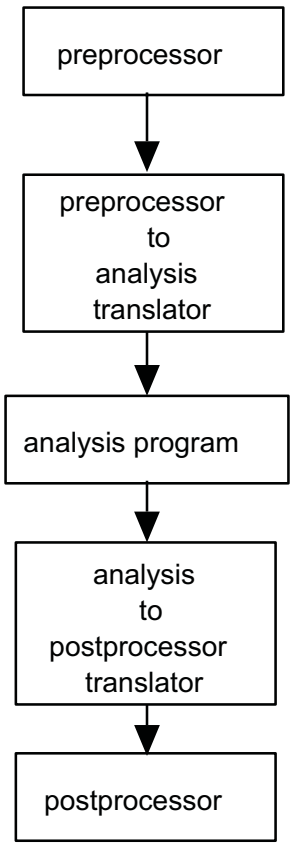

Figure 2.1: Schematic representation of "modular" finite element analyses.

upon the existence of suitable data translators. As input, such translators read files generated by pre-processors different from APES+. The output from such translators is data in a format read by APES+. The situation is illustrated schematically in Figure 2.1.

2.3 The APES+ Problem-Oriented Language

With the exception of the routines discussed below, the APES+ program is written in Fortran 95. Although FORTRAN provides free-form (list-directed) data entry, this input mechanism is typically inadequate. As a result, software engineers use other forms of user-machine dialogue that provide a better user interface than may be achieved with conventional FORTRAN input/output. One such proven approach is the problem-oriented language (POL).

A POL is an application-specific computer language. POLs typically consist of simple, easily remembered statements that are close to the user's natural language. Such input systems normally allow a convenient ordering of program input. POL command names, keywords, and data items are always read in free form. By freely mixing commands, keywords, data and comment lines, an input data file is transformed from a difficult to understand, rigidly formatted collection of numbers to a clear and natural description of the problem.

Data is input to the APES+ computer program using a POL. Each line of input is parsed;

i.e., is checked for valid syntax. This task is carried out by the powerful COMPRO command processor [95], which uses the INTERP system of subroutines for lexical analysis [48; 96]. Both COMPRO and INTERP are written in FORTRAN 77. The remainder of this section presents the underlying concepts of the APES+ POL.

2.3.1 Tokens

The APES+ POL considers each input data line as consisting of a collection of *tokens*. Strictly speaking, a token is defined to be a recognizable unit of information on a data line. The four token types relevant to this discussion are the text token, the numeric token, the string token, and the list token; examples of these four are: the command names and keywords, integer and real (floating point) numeric data, descriptive titles, and lists of integers, respectively. Additional details regarding these tokens are given below.

2.3.2 Integers

Integers are signed (+/-) or unsigned whole numbers without a decimal point and without an exponential multiplier. Some examples of properly formed integer numeric fields are given below:

125 -30 +142

In the command descriptions that follow, integers are indicated by the notation ##.

2.3.3 Real Numbers

Real numbers consist of a signed (+/-) or unsigned number containing at least one digit, and either an optional decimal point or an optional exponential multiplier. The latter consists of the letter "E" (or "e") followed by a signed (+/-) or unsigned integer. Some examples of properly formed real numeric fields are given below:

-12.25 +7.20E+02 -3.e-7 2e4

In the command descriptions that follow, real numbers are indicated by the notation #.#.

2.3.4 Strings

A string token is used to convey textual information. If the string contains other than the characters (A-Z, a-z, 0-9), or the underscore (_) character, it must be surrounded by a pair of double quotes ("). For example the commands

```
ANALYSIS  TITLE   run_1_iteration_3

ANALYSIS  TITLE  ‘‘run 1 iteration 3’’
```

both contain valid strings. Strings may not extend beyond data line boundaries. Delimiters (to be discussed below) and any other currently defined special characters may appear within a quoted string. In a string all special characters lose their significance and are simply treated as another character in the string. The entire string between the double quotes represents a single token. In the command descriptions that follow, strings are indicated by the notation "string".

2.3.5 Lists

Lists are used to group together a collection of integers. The syntax for lists can be better understood by studying the following simple examples. In particular, a list input is used in conjunction with the **OUTPUT NODES PRINT** command. The simplest list is a single integer. The following command requests the printing of primary dependent variables at node 4:

```
OUTPUT  NODES  PRINT  4
```

Multiple items can be separated by white space (tabs or spaces), or by commas (,). The following commands both request values to be printed at nodes 1, 5, and 9.

```
OUTPUT  NODES  PRINT  1  5  9 OUTPUT  NODES  PRINT  1, 5, 9
```

Multiple items following a simple pattern can be specified by supplying a start, a stop, and an increment value. The following command requests output at nodes 1, 6, 11, 16, 21, and 26.

```
OUTPUT  NODES  PRINT 1:26:5
```

The increment value is optional and *defaults* to one (1). The following command thus requests output at nodes 5 through 9.

```
OUTPUT  NODES  PRINT 5:9
```

The construction (*start* : *stop* : *increment*) is referred to as a *range*. A list can be composed of any number of ranges. Thus the commands

```
OUTPUT  NODES  PRINT 1:9:2  20:30:5  95

OUTPUT  NODES  PRINT 1:9:2, 20:30:5, 95
```

both request output for nodes 1, 3, 5, 7, 9, 20, 25, 30, and 95. Finally, the string all represents a special list. The following command prints values for all node points.

```
OUTPUT  NODES  PRINT  ALL
```

In the command descriptions that follow, lists are indicated by the notation # : # : #.

2.3.6 Delimiters

The most obvious method of setting apart tokens on a data line is to separate them with blanks. The blank is a member of a class of characters known as *delimiters*. Delimiters are characters used to set apart tokens on a data line; delimiters are otherwise ignored by the command lexer. The APES+ POL has four delimiters: the blank, the tab, the comma, and the equal sign (=). These may be used interchangeably to separate tokens on a data line. The comma is generally reserved for delineating items in a list, while the equal sign is generally used in the context of assigning a value to a keyword. For example, the following commands are correctly constructed and produce identical results:

```
dimension maximum nodes 540

dimension maximum nodes = 540
```

2.3.7 The Comment Character

The comment character allows the user to freely annotate an input data file, thus greatly increasing its readability. The exclamation point (!) is APES+'s comment character. This character may appear at any location on the data line. If it appears in the first column of a data line, the entire line will be ignored by APES+. If it appears at any location on the line other than the first column, that portion of the data line following the comment character will be ignored. This provides an on-line comment feature, allowing a short comment to follow valid data on the same line. For example,

```
ANALYSIS ACTION CHECK          ! only check the input data
```

2.3.8 Command Continuation

The continuation character provides a mechanism for line continuation within the APES+ POL. If a user is approaching the maximum allowable length (typically 80 characters) of the line being entered but still has more items that need to appear on this line, the user enters a continuation character, and data entry is continued on the next line. There is no limit to the number of times a line may be continued. The ampersand (&) is APES+'s continuation character. If the last non-comment character on a data line is an ampersand, long commands may be split across line boundaries. For example, the commands

```
ANALYSIS  DESCRIPTION  TERMINATING  NONLINEAR

ANALYSIS DESCRIPTION  &
         TERMINATING  NONLINEAR
```

are equivalent. If blank lines are used to increase the readability of a data file, they cannot appear between a line ending with a continuation character and the subsequent line.

2.4 The APES+ Command Syntax

The APES+ command syntax has been designed for ease of use, flexibility, and consistency. Where practical, the commands associated with the APES+ computer program follow the recommended format of the Standard Finite Element Language (SFEL) proposed by Nielan et al. [90].

2.4.1 Basic Structure

Commands consist of command names, of keywords and of numeric and/or alphanumeric data. These basic components of a command are separated with delimiters. As previously mentioned, the blank, tab, comma (,), and equals sign (=) are all used as delimiters. For example, the following commands are equivalent:

```
GRAVITY  ACCELERATION  VALUE   32.2

GRAVITY  ACCELERATION  VALUE = 32.2

GRAVITY  ACCELERATION  VALUE , 32.2
```

In the above commands, "GRAVITY" represents the command name; "ACCELERATION" represents a sub-command, "VALUE" is a keyword, and 32.2 is numeric data.

All commands and keywords are *case independent*. For example, the following commands are equivalent:

```
NONLINEAR ITERATION LIMIT  10

Nonlinear ITEration limit  10

nonlinear iteration limit  10
```

All keywords and their associated data (but not command or sub-command names) are *order independent*; i.e., within a given command, the keywords can be specified in any order.

2.4.2 Required and Optional Keywords

The APES+ command syntax contains two types of keywords: *required* and *optional* ones. Required keywords and their associated data must be present as part of the command. Omission of a required keyword and its data will result in an error condition. Optional keywords and their associated data may be omitted from the command and APES+ will either do nothing or, more likely, will provide a default value for the omitted data.

2.4.3 Command Name and Keyword Abbreviations

All command names and keywords associated with the APES+ POL may typically be abbreviated by specifying only their first *three* letters. For example, the following two commands are equivalent

```
GRAVITY  ANGLE  HISTORY  2

gra  ang  his  2
```

In the command descriptions provided in the user documentation, the *minimum* length of a command name and/or keyword are indicated by showing the required portion in *upper case* letters and the remaining optional letters in *lower case*.

2.4.4 Notation Used in Command Descriptions

To summarize the previous discussion, the following conventions are used in the subsequent user documentation:

- ## denotes an integer.
- #.# denotes a real number.
- # : # : # denotes a list of integers.
- "string" denotes a quoted string.

2.5 Units

The units used for various input quantities must be consistent, and determine the units of the output. In the subsequent description of the program input, the units associated with input quantities are described using the following conventions:

- "F" denotes units of force (e.g., Newtons, dynes, pounds, etc.).
- "L" denotes units of length (e.g., meters, centimeters, inches, etc.).
- "m" denotes units of mass (e.g., kilograms, grams, slugs, etc.).
- "t" denotes units of time (e.g., seconds, minutes, etc.).
- "T" denotes units of temperature (e.g., degrees Kelvin, Fahrenheit,etc.).

Chapter 3

Print Control Options in APES$^+$

Realizing that finite element analyses generate large amounts of data, several options have been provided in APES$^+$ to control this amount of data. These options are described below.

3.1 Print Options for a Given Element

The amount of data printed for a given element can be controlled by keywords entered in the element description. The following keywords are available:

DONT_PRINT_Results: Specification of this keyword omits output of secondary dependent variables (i.e., strains and stresses) for this element.

DONT_PRINT_STRAINS: Specification of this keyword omits output of element strains.

DONT_PRINT_STRESSES: Specification of this keyword omits the output of element stresses.

PRINT_PRIN_STRAINS: Specification of this keyword causes principal strains to be computed and printed.

PRINT_PRIN_STRESSES: Specification of this keyword causes principal stresses to be computed and printed.

PRINT_AVG_STRAINS: Specification of this keyword causes average strains to be computed (averaged over the secondary quadrature points) and printed. This option is only meaningful for elements possessing more than one quadrature point for secondary dependent variables.

PRINT_AVG_STRESSES: Specification of this keyword causes average stresses to be computed (averaged over the secondary quadrature points) and printed. This option is only meaningful for elements possessing more than one quadrature point for secondary dependent variables.

PRINT_VOLUMETRIC_STRAIN: Specification of this keyword causes the volumetric strain to be computed and printed for an element. In addition, the ratio of the absolute

value of the volumetric strain to the absolute value of the minimum non-zero normal strain in the element is printed. This ratio is instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit.

3.2 Print Options over all Elements

Extreme values (i.e., minima and maxima) for secondary dependent variables can be computed and printed for all elements in a mesh at a given solution increment. This is realized by specifying suitable **ECHO** commands *prior* to the **FINISHED SETTINGS** command in the input data file. The following extreme values can be printed:

Printing of maximum average strains: Maximum *average* (over all secondary quadrature points in the element) values of the appropriate infinitesimal strain components will be printed if the **ECHO ELEMENT_MAX_AVG_STRAIN ON** command is specified.

Printing of maximum average stresses: Maximum *average* (over all secondary quadrature points in the element) values of the appropriate stress components will be printed if the **ECHO ELEMENT_MAX_AVG_STRESS ON** command is specified.

Printing of maximum strains: Maximum values of the appropriate infinitesimal strain components will be printed if the **ECHO ELEMENT_MAX_STRAIN ON** command is specified.

Printing of maximum stresses: Maximum values of the appropriate stress components will be printed if the **ECHO ELEMENT_MAX_STRESS ON** command is specified.

Printing of maximum pore pressure: The maximum pore pressure will be printed if the **ECHO ELEMENT_MAX_PRESSURE ON** command is specified.

Printing of minimum average strains: Minimum *average* (over all secondary quadrature points in the element) values of the appropriate infinitesimal strain components will be printed if the **ECHO ELEMENT_MIN_AVG_STRAIN ON** command is specified.

Printing of minimum average stresses: Minimum *average* (over all secondary quadrature points in the element) values of the appropriate stress components will be printed if the **ECHO ELEMENT_MIN_AVG_STRESS ON** command is specified.

Printing of minimum strains: Minimum values of the appropriate infinitesimal strain components will be printed if the **ECHO ELEMENT_MIN_STRAIN ON** command is specified.

Printing of minimum stresses: Minimum values of the appropriate stress components will be printed if the **ECHO ELEMENT_MIN_STRESS ON** command is specified.

Printing of minimum pore pressures: The minimum pore pressure will be printed if the **ECHO ELEMENT_MIN_PRESSURE ON** command is specified.

3.3 Print Options over all Nodes

Extreme values (i.e., minima and maxima) for primary dependent (nodal) variables can be computed and printed for all nodes in a mesh at a given solution increment. This is realized by specifying suitable **ECHO** commands *prior* to the **FINISHED SETTINGS** command in the input data file. The following extreme values can be printed:

Maximum values of primary dependent variables: will be printed if the command **ECHO NODAL_MAXIMUMS ON** is specified.

Minimum values of primary dependent variables: will be printed if the command **ECHO NODAL_MINIMUMS ON** is specified.

Chapter 4

Overview of APES$^+$ Commands

The commands associated with the APES$^+$ computer program fall into one of two general categories, namely:

- Commands Defining Solution Control (see Section 4.1).
- Commands Describing the Mathematical Model (see Section 4.2).

Each of these categories is briefly described in the following two sections.

4.1 Commands Defining Solution Control

This category of APES$^+$ commands includes the specification of solution options and the dynamic allocation of memory for program execution. The associated commands are listed below; detailed descriptions of thee commands are provided in the chapters listed in parentheses following each general command type.

- General solution options: the **ANALYSIS** commands (see Chapter 5).
- Dynamic array dimensioning options: the **DIMENSION** commands (see Chapter 6).
- Input printing options: the **ECHO** commands (see Chapter 7).
- Delineation commands: the **FINISHED** commands (see Chapter 8).
- Gravitational acceleration information: the **GRAVITY** commands (see Chapter 9).
- Numerical integration options: the **INTEGRATION TIME** command (see Chapter 10).
- Nonlinear analysis options: the **NONLINEAR** commands (see Chapter 11).

4.2 Commands Describing the Mathematical Model

The second category of APES+ commands includes all commands necessary to define the mathematical model to be analyzed. The associated commands are listed below; detailed descriptions of thee commands are provided in the chapters listed in parentheses following each general command type.

NOTE: These commands can only be specified once the **FINISHED SETTINGS** command has been entered (see Section 8.3).

- Commands pertaining to the **ELEMENT** descriptions (see Chapter 18).
- Information pertaining to the **FUNCTION** commands (see Chapter 12).
- Generation of nodal coordinates on a surface and within a volume: the **GENERATE** commands (see Chapter 13).
- Information pertaining to the **HYDRAULIC CONDUCTIVITY** commands (see Chapter 14).
- Definition of the initial material state: the **INITIAL** commands (see Chapter 15).
- Information pertaining to the **MATERIAL** idealizations (see Chapter 30).
- Specification of the nodal coordinates: the **NODES** commands (see Chapter 35).
- The element and node output controls: the **OUTPUT** commands (see Chapter 37).
- Information pertaining to nodal specifications (boundary conditions): the **SPECIFICATION** commands (see Chapter 36).
- Specification of the solution time history: the **SOLUTION TIME** command (see Chapter 38).

Chapter 5

The ANALYSIS Commands

General solution options are specified using the following **ANALYSIS** commands:

- The **ANALYSIS ACTION** command (see Section 5.1).
- The **ANALYSIS DESCRIPTION** command (see Section 5.2).
- The **ANALYSIS IDEALIZATION** command (see Section 5.3).
- The **ANALYSIS MESH** commands (see Section 5.4).
- The **ANALYSIS SCALAR_TYPE** command (see Section 5.5).
- The **ANALYSIS STORAGE** command (see Section 5.6).
- The **ANALYSIS TEMPORAL** command (see Section 5.7).
- The **ANALYSIS TITLE** commands (see Section 5.8).
- The **ANALYSIS TYPE** command (see Section 5.9).
- The **ANALYSIS UPDATE** command (see Section 5.10).
- The **ANALYSIS OPTIONS FINISHED** command (see Section 5.11).

This chapter gives additional details pertaining to the above commands.

5.1 The ANALYSIS ACTION Command

Synopsis

The **ANALYSIS ACTION** command is used to select the extent of the solution that APES+ shall perform.

Syntax

The following syntax is associated with the **ANALYSIS ACTION** command:

ANAlysis ACTion [ANAlyze | CHEck]

Explanatory Notes

- If the **CHECK** keyword is specified, program execution will terminate after the mathematical model has been generated and pertinent values printed. In order to avoid wasting computer time analyzing an incorrect model, this option should be used prior to the initial analysis of all but the most trivial problem. For this reason, the **CHECK** option is the default condition set in APES+.
- If the **ANALYZE** keyword is specified, the mathematical model is created and an analysis is performed.

Example of Command Usage

To perform an analysis of a previously verified mathematical model, enter either of the following commands:

```
analysis action  analyze

ana act ana
```

5.2 The ANALYSIS DESCRIPTION Command

Synopsis

The **ANALYSIS DESCRIPTION** command is used to specify whether an analysis is **LINEAR** (the default condition) or is **NONLINEAR**.

Syntax

The following syntax is associated with the **ANALYSIS DESCRIPTION** command:

ANAlysis DEScription [LINear | TERminating (NONlinear) | NONterminating (NONlinear)]

Explanatory Notes

In the case of nonlinear analyses the **ANALYSIS DESCRIPTION** command also instructs APES+ whether or not to terminate the analysis when convergence has not been achieved within a given solution increment.

Example of Command Usage

To specify a nonlinear analysis in which execution is to be terminated if convergence is not achieved, enter either of the following commands:

```
analysis description terminating nonlinear

ana  des  ter  non
```

5.3 The ANALYSIS IDEALIZATION Command

Synopsis

The **ANALYSIS IDEALIZATION** command is used to specify the particular two, three or, in the case of generalized plane strain, pseudo three-dimensional idealization assumed for the analysis.

Syntax

The following syntax is associated with the **ANALYSIS IDEALIZATION** command:

ANAlysis IDEalization
[AXIsymmetric | GENERALIZED_PLANE_STRAin | PLANE_STREss
| PLANE_STRAin | THRee-dimensional]

Explanatory Notes

For torsionless axisymmetric analyses, realized by specifying the **AXISYMMETRIC** keyword, all reference in the subsequent sections to x_1 and x_2 should instead be interpreted as being to r (the radial coordinate) and to z (the axial coordinate), respectively. The default two-dimensional idealization for mechanical analyses is **PLANE_STRESS**.

Example of Command Usage

To specify an analysis in which a plane strain idealization is desired, enter either of the following commands:

```
analysis idealization  PLANE_STRAIN

ana ide  plane_stra
```

5.4 The ANALYSIS MESH Command

Synopsis

The **ANALYSIS MESH** commands are used to define options to be used in defining the mesh.

Syntax

The following syntax is associated with the **ANALYSIS MESH** command:

EXTernal | INTernal | ISOparametric | LAPlacian |

Explanatory Notes

Figure 5.1 schematically illustrates the meshing options available in the APES+ computer program.

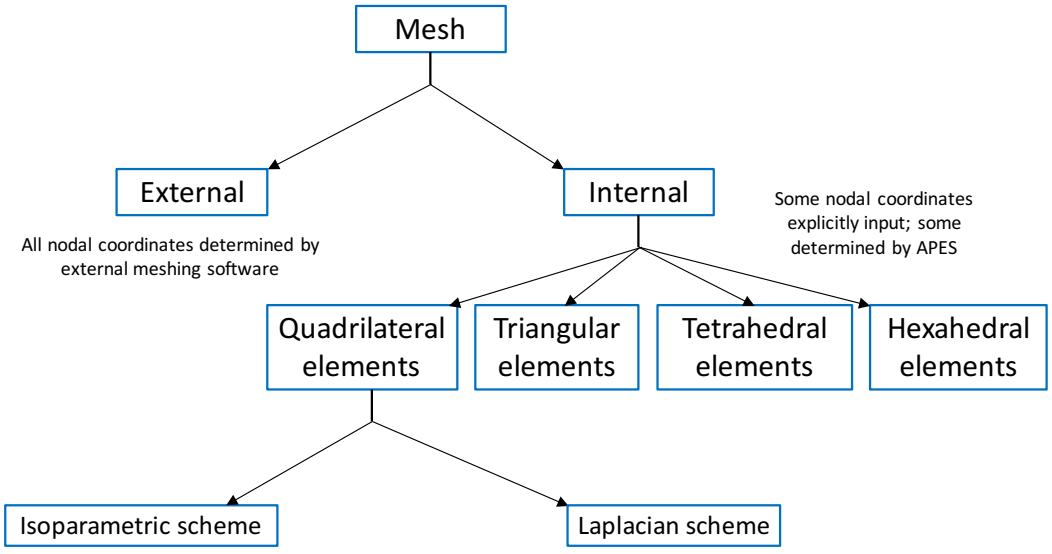

Figure 5.1: Schematic illustration of meshing options available in APES+.

The **ANALYSIS MESH EXTERNAL** command should be specified if the mesh has been generated using software *different* from APES+. In this case no mesh generation or coordinate location shall be performed in APES+. This is the default condition.

The **ANALYSIS MESH INTERNAL** command should be specified if the nodal coordinates shall be located using sundry interpolations within APES+. This pertains to elements that are not included in the automatic coordinate generation routines included in APES+ (see description of the remaining **ANALYSIS MESH** commands below), and involves sundry coordinate interpolations.

The **ANALYSIS MESH ISOPARAMETRIC** command instructs APES+ to use an "isoparametric" coordinate determination scheme [37; 49] to determine unknown nodal coordinates in the mesh. Presently this only applies to four-, eight- and nine-node quadrilateral elements (i.e., Q4##. Q8## and Q9## elements). This scheme is invoked by subsequently specifying the **GENERATE SURFACES** command (see Chapter 13).

The **ANALYSIS MESH LAPLACIAN** command instructs APES+ to use an "Laplacian" coordinate determination scheme [37; 49] to determine unknown nodal coordinates in the mesh. Presently this only applies to four-, eight- and none-node quadrilateral elements (i.e., Q4##. Q8## and Q9## elements). This scheme is invoked by subsequently specifying the **GENERATE SURFACES** command (see Chapter 13).

Example of Command Usage

To specify an analysis in which an isoparametric coordinate determination scheme is desired, enter either of the following commands:

```
analysis mesh  isoparametric

ana mes  iso
```

If all mesh data has been determined using post-processing software different from APES+, enter either of the following commands:

```
analysis mesh  external

ana mes  ext
```

5.5 The ANALYSIS SCALAR_TYPE Command

Synopsis

The **ANALYSIS SCALAR_TYPE** command is used to specify the type of scalar analysis to be performed. The command only has meaning if the **ANALYSIS TYPE SCALAR** command has been given (see Section 5.9).

Syntax

The following syntax is associated with the **ANALYSIS SCALAR_TYPE** command:

ANAlysis SCAlar_type [FLUid_flow | HEAT_flow | TORsion]

Explanatory Notes

The **ANALYSIS SCALAR_TYPE FLUID_FLOW** command should be specified if a problem involving the steady-state irrotational flow of fluid is to be analyzed.

The **ANALYSIS SCALAR_TYPE HEAT_FLOW** command should be specified if a problem involving the steady-state heat flow is to be analyzed. This is the default option.

The **ANALYSIS SCALAR_TYPE TORSION** command should be specified if a problem involving the torsion of prismatic bars with non-circular cross-sections is to be analyzed.

Example of Command Usage

To specify a scalar analysis for the torsion of a prismatic bar with non-circular cross-section, enter either of the following commands:

```
analysis scalar_type  torsion

analysis_sca tor
```

5.6 The ANALYSIS STORAGE Command

Synopsis

The **ANALYSIS STORAGE** command is used to specify the type of storage to be employed for the global ("system") equations for the finite element analysis.

Syntax

The following syntax is associated with the **ANALYSIS STORAGE** command:

ANAlysis STOrage [NONsymmetric | SYMmetric]

Explanatory Notes

The **NONSYMMETRIC** storage option is appropriate for material models employing non-associative flow rules. The default storage is **SYMMETRIC**.

Example of Command Usage

To specify a non-symmetric storage scheme, enter either of the following commands:

```
analysis storage nonsymmetric

ana sto  non
```

5.7 The ANALYSIS TEMPORAL Command

Synopsis

The **ANALYSIS TEMPORAL** command is used to specify the type of temporal idealization to be used in the analysis.

Syntax

The following syntax is associated with the **ANALYSIS TEMPORAL** command:

ANAlysis TEMporal [STAtic | TRAnsient | DYNamic]

Explanatory Notes

- A **TRANSIENT** analysis is associated with problems involving only first-order time derivatives - no inertial effects are included in the analysis. Examples of such analyses are flow through porous media under quasi-static conditions, quasi-static analyses of solids in which the loading is specified in several segments, etc.
- In **DYNAMIC** analyses, inertial effects are included in the analysis.
- Finally, **STATIC** analyses refer to those analyses, typically uncoupled and certainly linear, in which the load is applied very slowly in a single solution segment.
- The default is a **TRANSIENT** analysis.

Example of Command Usage

To specify a transient analysis, enter either of the following commands:

```
analysis  temporal  transient

ana  tem  tra
```

5.8 The ANALYSIS TITLE Command

Synopsis

The **ANALYSIS TITLE** command is used to supply alphanumeric information for the purpose of describing a given analysis.

Syntax

The following syntax is associated with the **ANALYSIS TITLE** command:

ANAlysis TITle ("string")

Explanatory Notes

The alphanumeric information supplied via the **ANALYSIS TITLE** command *must* be enclosed in double quotes (" "). If desired, all alphanumeric information is "echoed" in the program output.

Example of Command Usage

The following lines represent valid alphanumeric information that is supplied via the **ANALYSIS TITLE** command:

```
analysis title ''This example shows how to enter information''
ANALYSIS TITLE ''that is used to describe a particular''
anal title '' analysis. In this portion of the input, as''
ana tit   ''many lines as required may be entered''
```

5.9 The ANALYSIS TYPE Command

Synopsis

The **ANALYSIS TYPE** command is used to specify the type of analysis to be performed.

Syntax

The following syntax is associated with the **ANALYSIS TYPE** command:

ANAlysis TYPe [FLOW_poromechanical | FLOW_POROMECHANICAL_Thermal
| POROmechanical | PORO_Thermal
| SOLid | SOLID_Thermal
| SCAlar]

Explanatory Notes

- Specification of the **POROMECHANICAL** keyword indicates that a standard (uncoupled) quasi-static mechanical analysis of a porous material is to be performed. This is the default setting. In such analyses, the solution domain can be discretized using both *irreducible* elements (see Section 18.7) and *mixed* elements with *discontinuous* pressure approximations (see Chapter 27).
- If a quasi-static, semi-coupled thermoelastic analysis of a porous material is to be performed, the keyword **MECHANICAL_THERMAL** should be specified. In such analyses, the solution domain can be discretized using both *irreducible* elements (see Section 18.7) and *mixed* elements with *discontinuous* pressure approximations (see Chapter 27).
- To model saturated conditions where fluid flow occurs (or where a potential for fluid flow exists such as under ideal undrained conditions) in conjunction with (and coupled to) mechanical deformations, the keyword **FLOW_POROMECHANICAL** is specified. In such analyses, the solution domain must be discretized using *mixed* elements with *continuous* pressure approximations (see Chapter 28).
- If an analysis is to be performed in which mechanical deformations, flow of pore fluid, and thermal effects are coupled together, the keyword **FLOW_PORO_THERMAL** should be specified. In such analyses, the solution domain must be discretized using *mixed* elements with *continuous* pressure approximations and temperature unknowns.

- When this is the case, the mass density of the soil, specified using the **MATERIAL DENSITY SOLID** command, represents the dry mass density.[1] Since the stresses printed in the output are *effective*, the pore fluid pressure must be added to them in order to obtain *total* stresses.

- To model *unsaturated* conditions where the pore pressure is assumed to be zero and pore fluid is assumed not to flow, the **ANALYSIS TYPE** must be set to **POROMECHANICAL**. In this case the soil density must include the mass (or weight) of any water present in a partially saturated soil.

- Conditions where part of the soil mass is unsaturated and part is saturated can be modeled by specifying the **FLOW_POROMECHANICAL** keyword, along with a very small value of combined bulk modulus (see the **MATERIAL DENSITY FLUID** or **MATERIAL DENSITY SOLID** commands) for the unsaturated portions of the soil.

- Specification of the **SOLID** or **SOLID_THERMAL** command transforms APES+ into a non-linear analysis program in which the continuum is assumed to consist of a single phase; i.e., the solid. As such, APES+ can (without loss of efficiency) be used to analyze non-porous continua, or porous media where the flow of pore fluid is assumed to be of negligible importance. In the latter case, the material density must include the mass (or weight) of any water present in the soil. In such analyses, the solution domain can be discretized using both *irreducible* elements (see Section 18.7), *mixed* elements with *discontinuous* pressure approximations (see Chapter **??**), or *non-conforming* elements.

- Specification of the **SCALAR** keyword indicates that an analysis involving only scalar primary dependent variables (e.g., temperature, hydraulic head, Prandtl stress function for torsion analyses of non-circular cross-sections, etc.) is to be performed. In such analyses, the solution domain must be discretized using *scalar* elements.

Example of Command Usage

To specify a semi-coupled thermoelastic analysis, enter *either* of the following commands:

```
analysis type  mechanical_thermal

ana typ sca  mechanical_the
```

[1] If the acceleration of gravity, specified using the **GRAVITY ACCELERATION VALUE** command, is taken equal to unity, then the soil density represents the unit weight of the soil.

5.10 The ANALYSIS UPDATE Command

Synopsis

The **ANALYSIS UPDATE** command is used to specify whether the nodal coordinates will be updated during the course of an analysis.

Syntax

The following syntax is associated with the **ANALYSIS UPDATE** command:

ANAlysis UPDate (COOrdinates)

Explanatory Notes

Although infinitesimal measures of strain are still used, the process of updating coordinates can, in certain cases, improve the accuracy of the results. The default condition is no updating of nodal coordinates.

Example of Command Usage

To specify the updating of nodal coordinates, enter either of the following commands:

```
analysis  update coordinates

ana  upd  coor
```

5.11 The ANALYSIS OPTIONS FINISHED Command

Synopsis

The **ANALYSIS OPTIONS FINISHED** command is used to designate the end of the ANALYSIS commands.

Syntax

The following syntax is associated with the **ANALYSIS OPTIONS FINISHED** command:

ANAlysis OPTions (Finished)

Explanatory Notes

Specification of the **ANALYSIS OPTIONS FINISHED** command causes all specialized variables associated with the analysis to be initialized. The allocation of solution arrays takes place once the DIMENSION commands (see Chapter 6) are completed.

Example of Command Usage

To specify the updating of nodal coordinates, enter either of the following commands:

```
analysis  options finished

ana opt  fin

ana opt
```

Chapter 6

The DIMENSION Commands

All arrays associated with APES$^+$ whose dimensions are problem dependent are dimensioned dynamically. The following **DIMENSION MAXIMUM** Commands commands are used by the analyst to declare the maximum size of problem-dependent arrays:

- The **DIMENSION MAXIMUM** commands related to elements (see Section 6.1).
- The **DIMENSION MAXIMUM BULK** commands (see Section 6.2).
- The **DIMENSION MAXIMUM DENSITY** commands (see Section 6.3).
- The **DIMENSION MAXIMUM FUNCTION** commands (see Section 6.4).
- The **DIMENSION MAXIMUM HYDRAULIC** commands (see Section 6.5).
- The **DIMENSION MAXIMUM INITIAL** command (see Section 6.6).
- The **DIMENSION MAXIMUM MATERIAL** commands (see Section 6.7).
- The **DIMENSION MAXIMUM NODES** command (see Section 6.8).
- The **DIMENSION MAXIMUM SCALAR_PARAMETERS** commands (see Section 6.9).
- The **DIMENSION MAXIMUM SCALAR_SOURCE** commands (see Section 6.10).
- The **DIMENSION MAXIMUM SPECIFIC_HEAT** commands (see Section 6.11).
- The **DIMENSION MAXIMUM THERMAL_CONDUCTIVITY** commands (see Section 6.12).
- The **DIMENSION MAXIMUM THERMAL_EXPANSION** commands (see Section 6.13).

The subsequent sections give additional details pertaining to the above **DIMENSION** commands.

6.1 The DIMENSION MAXIMUM Element Commands

Synopsis

All arrays associated with APES+ whose dimensions are problem dependent are dimensioned dynamically. The **DIMENSION MAXIMUM** commands associated with element descriptions are used by the analyst to declare the maximum size of arrays used to store element information.

Syntax

The following syntax is associated with the **DIMENSION MAXIMUM** commands related to element data:

DIMension MAXimum L2P0 ##
DIMension MAXimum L3P0 ##
DIMension MAXimum L2FDP0 ##

DIMension MAXimum BE2_2D ##
DIMension MAXimum TB2_2D ##
DIMension MAXimum TB3_2D ##

DIMension MAXimum INFQ6 ##
DIMension MAXimum INFQ8 ##
DIMension MAXimum INFQ9 ##

DIMension MAXimum LK1 ##
DIMension MAXimum LRH ##

DIMension MAXimum T3P0 ##
DIMension MAXimum T6P0 ##
DIMension MAXimum T7P0 ##
DIMension MAXimum T10P0 ##
DIMension MAXimum T15P0 ##

DIMension MAXimum Q4P0 ##
DIMension MAXimum Q8P0 ##
DIMension MAXimum Q9P0 ##
DIMension MAXimum Q12P0 ##
DIMension MAXimum Q16P0 ##
DIMension MAXimum Q17P0 ##

```
DIMension MAXimum TET4P0  ##
DIMension MAXimum TET10P0  ##

DIMension MAXimum H8P0  ##
DIMension MAXimum H20P0  ##
DIMension MAXimum H27P0  ##

DIMension MAXimum T6P1d  ##
DIMension MAXimum T7P3d  ##
DIMension MAXimum T10P3d  ##

DIMension MAXimum Q4P1d  ##
DIMension MAXimum Q8P3d  ##
DIMension MAXimum Q9P3d  ##
DIMension MAXimum TET10P1d  ##

DIMension MAXimum H8P1d  ##
DIMension MAXimum H20P4d  ##
DIMension MAXimum H27P4d  ##

DIMension MAXimum T4P3c  ##
DIMension MAXimum T6P3c  ##
DIMension MAXimum T7P3c  ##
DIMension MAXimum T15P6c  ##
DIMension MAXimum Q5P4c  ##
DIMension MAXimum Q8P4c  ##
DIMension MAXimum Q9P4c  ##
DIMension MAXimum H9P8c  ##
DIMension MAXimum H20P8c  ##
DIMension MAXimum H27P8c  ##

DIMension MAXimum H8P8c  ##
DIMension MAXimum Q4P4c  ##

DIMension MAXimum QM6  ##

DIMension MAXimum L2S  ##
DIMension MAXimum T3S  ##
DIMension MAXimum Q4S  ##
DIMension MAXimum Q8S  ##
DIMension MAXimum H8S  ##
DIMension MAXimum TET4S  ##
```

Explanatory Notes

The keywords associated with the **DIMENSION MAXIMUM** commands related to element data have the following meaning:

L2P0: 2-node irreducible (linear displacement) line (bar) element for two- or three-dimensional mechanical analyses. Additional details pertaining to this element type are available in Section 19.2.

L3P0: 3-node (irreducible (quadratic displacement) line (bar) element for two- or three-dimensional mechanical analyses. Additional details pertaining to this element type are available in Section 19.3.

L2FDP0: 2-node irreducible (linear displacement) finite deformation "membrane" element for mechanical analyses. Additional details pertaining to this element type are available in Section 19.1.

BE2_2D: Cubic (Hermitian) 2-node Bernoulli-Euler (C^1 continuous) plane frame element. Additional details pertaining to this element type are available in Section 20.1.

TB2_2D: Linear 2-node "Timoshenko" (C^0 continuous) plane frame element. Additional details pertaining to this element type are available in Section 20.2.

TB3_2D: Quadratic 3-node "Timoshenko" (C^0 continuous) plane frame element. Additional details pertaining to this element type are available in Section 20.3.

INFQ6: 6-node irreducible "infinite" (linear/quadratic displacement) quadrilateral element. Additional details pertaining to this element type are available in Section 21.2.

INFQ8: 8-node irreducible "infinite" (bi-quadratic displacement) quadrilateral element. Additional details pertaining to this element type are available in Section 21.3.

INFQ9: 9-node irreducible "infinite" (bi-quadratic displacement) quadrilateral element. Additional details pertaining to this element type are available in Section 21.4.

LK1: 4-node (linear) kinematically consistent zero-thickness interface element developed by Li and Kaliakin [57]. Additional details pertaining to this element type are available in Section 22.1.

LRH: 4-node (linear) zero-thickness ("link") interface element originally developed by L. R. Herrmann in 1978 [38]. Additional details pertaining to this element type are available in Section 22.2.

T3P0: 3-node irreducible, *linear* displacement isoparametric triangular element. It possesses six (6) displacement degrees of freedom. Additional details pertaining to this element type are available in Section 23.1.

T6P0: 6-node irreducible, *quadratic* displacement isoparametric triangular element. It possesses twelve (12) displacement degrees of freedom. Additional details pertaining to this element type are available in Section 23.2.

T7P0: 7-node irreducible, *quadratic* displacement (enhanced by a cubic "bubble" mode), triangular element. It possesses fourteen (14) displacement degrees of freedom. Additional details pertaining to this element type are available in Section 23.3.

T10P0 : 10-node irreducible, *cubic* displacement isoparametric triangular element. It possesses twenty (20) displacement degrees of freedom. Additional details pertaining to this element type are available in Section 23.4.

T15P0 : 15-node irreducible, *quartic* displacement isoparametric triangular element.It possesses thirty (30) displacement degrees of freedom. Additional details pertaining to this element type are available in Section 23.5.

Q4P0: 4-node irreducible, bi-linear displacement isoparametric quadrilateral element. It possesses eight (8) displacement degrees of freedom. Additional details pertaining to this element type are available in Section 24.1.

Q8P0: 8-node irreducible, quadratic displacement isoparametric "serendipity" quadrilateral element. It possesses sixteen (16) displacement degrees of freedom. Additional details pertaining to this element type are available in Section 24.2.

Q9P0: 9-node irreducible, bi-quadratic displacement isoparametric Lagrangian quadrilateral element. It possesses eighteen (18) displacement degrees of freedom. Additional details pertaining to this element type are available in Section 24.3.

Q12P0: 12-node irreducible, cubic displacement isoparametric "serendipity" quadrilateral element. It possesses twenty-four (24) displacement degrees of freedom. Additional details pertaining to this element type are available in Section 24.4.

Q16P0: 16-node irreducible, cubic displacement isoparametric Lagrangian quadrilateral element. It possesses thirty-two (32) displacement degrees of freedom. Additional details pertaining to this element type are available in Section 24.5.

Q17P0: 17-node irreducible, quartic displacement isoparametric "serendipity" quadrilateral element. It possesses thirty-four (34) displacement degrees of freedom. Additional details pertaining to this element type are available in Section 24.6.

TET4P0: 4-node irreducible, linear displacement, tetrahedral element. It possesses twelve (12) displacement degrees of freedom. Additional details pertaining to this element type are available in Section 25.1.

TET10P0: 10-node irreducible, quadratic displacement, tetrahedral element. It possesses thirty (30) displacement degrees of freedom. Additional details pertaining to this element type are available in Section 25.2.

H8P0: 8-node irreducible, tri-linear displacement isoparametric hexahedral element. It possesses twenty-four (24) displacement degrees of freedom. Additional details pertaining to this element type are available in Section 26.1.

H20P0: 20-node irreducible, "serendipity" quadratic displacement isoparametric hexahedral element. It possesses sixty(60) displacement degrees of freedom. Additional details pertaining to this element type are available in Section 26.2.

H27P0: 27-node irreducible, Lagangian quadratic displacement isoparametric hexahedral element. It possesses eight-one (81) displacement degrees of freedom. Additional details pertaining to this element type are available in Section 26.3.

T6P1d: 6-node mixed-penalty triangle element. *Quadratic* in displacement, *constant* (discontinuous) pressure. The element possesses twelve (12) displacement degrees of freedom. The pressure degree of freedom is "condensed out" from the element equations prior to assembly. Additional details pertaining to this element type are available in Section 27.1.

T7P3d: 7-node mixed-penalty triangle element. A quadratic displacement approximation is enhanced by a cubic "bubble" mode. The pressure approximation is *linear* (discontinuous). The element possesses fourteen (14) displacement degrees of freedom. The pressure degrees of freedom are "condensed out" from the element equations prior to assembly. Additional details pertaining to this element type are available in Section 27.2.

T10P3d: 10-node mixed-penalty triangle element. *Cubic* in displacement, *linear* (discontinuous) pressure. The element possesses thirty (30) displacement degrees of freedom. The pressure degrees of freedom are "condensed out" from the element equations prior to assembly. Additional details pertaining to this element type are available in Section **??**.

Q4P1d: 4-node mixed-penalty quadrilateral element. *Bi-linear* in displacement, *constant* (discontinuous) pressure. The element possesses eight (8) displacement degrees of freedom. The pressure degree of freedom is "condensed out" from the element equations prior to assembly. Additional details pertaining to this element type are available in Section 27.3.

Q8P3d: 8-node mixed-penalty quadrilateral element. *Quadratic* in displacement, *linear* (discontinuous) pressure. The element possesses sixteen (16) displacement degrees of freedom. The three pressure degrees of freedom are "condensed out" from the element equations prior to assembly. Additional details pertaining to this element type are available in Section 27.4.

Q9P3d: 9-node mixed-penalty quadrilateral element. *Biquadratic* in displacement, *linear* (discontinuous) pressure. The element possesses eighteen (18) displacement degrees of freedom. The three pressure degrees of freedom are "condensed out" from the element equations prior to assembly. Additional details pertaining to this element type are available in Section 27.5.

H8P1d: 8-node mixed-penalty hexahedral element. *Trilinear* in displacement, *constant* (discontinuous) pressure. The element possesses twenty-four (24) displacement degrees of freedom. The pressure degree of freedom is "condensed out" from the element equations prior to assembly. Additional details pertaining to this element type are available in Section 27.6.

H20P4d: 20-node mixed-penalty hexahedral element. *Quadratic* in displacement, *linear* (discontinuous) pressure. The element possesses sixty (60) displacement degrees of freedom. The four pressure degrees of freedom are "condensed out" from the element equations prior to assembly. Additional details pertaining to this element type are available in Section **??**.

H27P4d: 27-node mixed-penalty hexahedral element. *Triquadratic* in displacement, *linear* (discontinuous) pressure. The element possesses eighty-one (81) displacement degrees of freedom. The four pressure degrees of freedom are "condensed out" from the element equations prior to assembly. Additional details pertaining to this element type are available in Section **??**.

T4P3c: 4-node mixed-penalty triangular element. *Linear* displacement approximation (enhanced with a cubic "bubble" mode) with *linear* pressure approximation. The element possesses eight (8) displacement degrees of freedom and three (3) pressure degrees of freedom. Additional details pertaining to this element type are available in Section 28.1.

T6P3c: 6-node mixed-penalty triangular element. *Quadratic* displacement approximation with *linear* pressure approximation. The element possesses twelve (12) displacement degrees of freedom and three (3) pressure degrees of freedom. Additional details pertaining to this element type are available in Section 28.2.

T7P3c: 7-node mixed-penalty triangular element. *Quadratic* displacement approximation (enhanced with a quadratic "bubble" mode) with *linear* pressure approximation. The element possesses fourteen (14) displacement degrees of freedom and three (3) pressure degrees of freedom. Additional details pertaining to this element type are available in Section 28.3.

T15P6c: 15-node mixed-penalty triangular element. *Quartic* displacement approximation with *quadratic* pressure approximation. The element possesses fifteen (15) displacement degrees of freedom and six (6) pressure degrees of freedom. Additional details pertaining to this element type are available in Section 28.4.

Q5P4c: 5-node mixed quadrilateral element. *Bilinear* displacement approximation (enhanced with a quadratic "bubble" mode) with *bilinear* pressure approximation. The element possesses ten (10) displacement degrees of freedom and four (4) pressure degrees of freedom. Additional details pertaining to this element type are available in Section 28.5.

Q8P4c: 8-node mixed-penalty quadrilateral element. *Quadratic* (serendipity) displacement approximation with *bilinear* pressure approximation. The element possesses sixteen (16) displacement degrees of freedom and four (4) pressure degrees of freedom. Additional details pertaining to this element type are available in Section 28.6.

Q9P4c: 9-node mixed-penalty quadrilateral element. *Biquadratic* (Lagrangian) displacement approximation with *bilinear* pressure approximation. The element possesses eighteen (18) displacement degrees of freedom and four (4) pressure degrees of freedom. Additional details pertaining to this element type are available in Section 28.7.

Q17P8c: 17-node mixed-penalty quadrilateral element. *Quartic* (serendipity) displacement approximation with *biquadratic* pressure approximation. The element possesses thirty-four (34) displacement degrees of freedom and eight (8) pressure degrees of freedom. Additional details pertaining to this element type are available in Section 28.8.

H9P8c: 9-node mixed hexahedral element with hybrid selective reduced integration. *Trilinear* displacement approximation (enhanced with a quadratic "bubble" mode) with *trilinear* pressure approximation. The element possesses twenty-seven (27) displacement degrees of freedom and eight (8) pressure degrees of freedom. Additional details pertaining to this element type are available in Section 28.9.

H20P8c : 20-node irreducible, *quadratic* displacement, "serendipity" hexahedral element. *Quadratic* displacement approximation with *trilinear* pressure approximation. The element possesses sixty (60) displacement degrees of freedom and eight (8) pressure degrees of freedom. Additional details pertaining to this element type are available in Section 28.10.

H27P8c : 27-node irreducible, *triquadratic* displacement, "Lagrangian" hexahedral element. *Triquadratic* displacement approximation with *trilinear* pressure approximation. The element possesses eighty-one (81) displacement degrees of freedom and eight (8) pressure degrees of freedom. Additional details pertaining to this element type are available in Section 28.11.

Q4P4c: 4-node mixed-penalty quadrilateral element with hybrid selective reduced integration. *Bilinear* displacement approximation with *bi-linear* pressure approximation. The element possesses eight (8) displacement degrees of freedom and four (4) pressure degrees of freedom. Additional details pertaining to this element type are available in Section 28.12.

H8P8c: 8-node mixed hexahedral element with selective reduced integration. Trilinear in displacement and pressure. Additional details pertaining to this element type are available in Section 28.13.

QM6: 4-node irreducible, *bi-linear* displacement, non-conforming quadrilateral developed by Taylor et al. [105]. The element possesses eight (8) displacement degrees of freedom. Additional details pertaining to this element type are available in Section **??**.

L2S: 2-node irreducible (linear scalar primary dependent variable) line (bar) element for two- or three-dimensional scalar analyses. It possesses two (2) scalar degrees of freedom. Additional details pertaining to this element type are available in Section **??**.

T3S: 3-node irreducible, *linear* scalar primary dependent variable isoparametric triangular element. It possesses three (3) scalar degrees of freedom. Additional details pertaining to this element type are available in Section 29.1.

Q4S: 4-node irreducible, bi-linear scalar primary dependent variable isoparametric quadrilateral element. It possesses four (4) scalar degrees of freedom. Additional details pertaining to this element type are available in Section 29.2.

Q8S: 8-node irreducible, quadratic scalar primary dependent variable isoparametric "serendipity" quadrilateral element. It possesses eight (8) scalar degrees of freedom. Additional details pertaining to this element type are available in Section 29.3.

TET4S: 4-node irreducible, linear scalar primary dependent variable, tetrahedral element. It possesses four (4) scalar degrees of freedom. Additional details pertaining to this element type are available in Section 29.4.

H8S: 8-node irreducible, tri-linear scalar primary dependent variable isoparametric hexahedral element. It possesses eight (8) scalar degrees of freedom. Additional details pertaining to this element type are available in Section 29.5.

Example of Command Usage

If the maximum number of irreducible eight-node quadrilateral elements is 220, enter either of the following commands:

```
dimension  maximum  q8p0  220

dim  max  q8p0  220
```

6.2 The DIMENSION MAXIMUM BULK Commands

Synopsis

The **DIMENSION MAXIMUM BULK** commands are used by the analyst to declare the maximum number of bulk modulus types for the solid and pore fluid phase to be used in the mathematical model.

Syntax

The following syntax is associated with the **DIMENSION MAXIMUM BULK_FLUID** commands:

DIMension MAXimum BULK_Fluid CONstant ##

Explanatory Notes

The keywords appearing above associated with the specification of bulk modulus data have the following meaning:

CONSTANT : equals the maximum number of constant bulk modulus types used to describe the pore fluid phase.

Syntax

The following syntax is associated with the **DIMENSION MAXIMUM BULK_SOLID** commands:

DIMension MAXimum BULK_Solid CONstant ##

Explanatory Notes

The keywords appearing above associated with the specification of bulk modulus data have the following meaning:

CONSTANT : equals the maximum number of constant bulk modulus types used to describe the solid phase.

Example of Command Usage

If three constant bulk modulus types are used to characterize the fluid phase, and two constant bulk modulus types are used for the solid phase, enter the following commands:

```
dim max bulk_fluid constant 3

dim max bulk_solid constant 2
```

6.3 The DIMENSION MAXIMUM DENSITY Commands

Synopsis

The **DIMENSION MAXIMUM DENSITY** commands are used by the analyst to declare the maximum number of mass density types for the solid and pore fluid phase to be used in the mathematical model.

Syntax

The**DIMENSION MAXIMUM DENSITY_FLUID** commands are specified using the following syntax:

DIMension MAXimum DENSITY_Fluid CONstant ##

Explanatory Notes

The keywords appearing above associated with the specification of mass density data have the following meaning:

CONSTANT : equals the maximum number of constant mass density types used to describe the pore fluid phase.

Syntax

The**DIMENSION MAXIMUM DENSITY_SOLID** commands are specified using the following syntax:

DIMension MAXimum DENSITY_Solid CONstant ##

Explanatory Notes

The keywords appearing above associated with the specification of mass density data have the following meaning:

CONSTANT : equals the maximum number of constant mass density types used to describe the solid phase.

Example of Command Usage

If one constant mass density is used to characterize the fluid phase, and two constant mass density types are used for the solid phase, enter the following commands:

```
dim max density_fluid constant 1

dim max density_solid constant 2
```

6.4 The DIMENSION MAXIMUM FUNCTION Commands

Synopsis

All arrays associated with APES+ whose dimensions are problem dependent are dimensioned dynamically. The **DIMENSION MAXIMUM FUNCTION** commands are used by the analyst to declare the maximum size of arrays used to store history function information.

Syntax

The following syntax is associated with the **DIMENSION MAXIMUM FUNCTION** commands:

DIMension MAXimum FUNction NUMbers ##

DIMension MAXimum FUNction PAIrs ##

Explanatory Notes

The keywords appearing above associated with the specification of history function information have the following meaning:

FUNCTION NUMBERS : equals the maximum number of history functions used in an analysis.

FUNCTION PAIRS : equals the largest number of time-function pairs (time, value) used in any history function.

Example of Command Usage

If the largest history function number is equal to 6 and the largest number of points in a given history is 15, enter the following commands:

```
dim max func  numbers  6

dim max func  pai  15
```

6.5 The DIMENSION MAXIMUM HYDRAULIC Commands

Synopsis

All arrays associated with APES+ whose dimensions are problem dependent are dimensioned dynamically. The **DIMENSION MAXIMUM HYDRAULIC** commands are used by the analyst to declare the maximum size of arrays used to store hydraulic conductivity (permeability) information.

Syntax

The following syntax is associated with the **DIMENSION MAXIMUM HYDRAULIC** commands:

DIMension MAXimum HYDraulic CONstant ##

DIMension MAXimum HYDraulic EXPonential ##

DIMension MAXimum HYDraulic POWer ##

DIMension MAXimum HYDraulic LOG-log ##

DIMension MAXimum HYDraulic SEMi-log ##

Explanatory Notes

Each **DIMENSION MAXIMUM HYDRAULIC** command is used by the analyst to tell APES+ the total number of hydraulic conductivity descriptions of a particular type that are to be used in a given mathematical model. The keywords associated with the **DIMENSION MAXIMUM HYDRAULIC** commands have the following meaning:

CONSTANT : refers to a hydraulic conductivity description with *constant* coefficients.

EXPONENTIAL : refers to a hydraulic conductivity description with coefficients that can vary *exponentially*.

POWER : refers to a hydraulic conductivity description with coefficients that can vary as a *power* of the void ratio.

LOG-LOG : refers to a hydraulic conductivity description with coefficients that can vary *logarithmically* with void ratio.

SEMI-LOG : refers to a hydraulic conductivity description with coefficients that can vary *semi-logarithmically* with void ratio.

Example of Command Usage

To specify three (3) constant hydraulic conductivity descriptions and one semi-logarithmic one, enter the following commands:

```
dimension max hydraulic  constant 3

dim max hyd semi-log  1
```

6.6 The DIMENSION MAXIMUM INITIAL Command

Synopsis

All arrays associated with APES+ whose dimensions are problem dependent are dimensioned dynamically. The **DIMENSION MAXIMUM INITIAL** commands are used by the analyst to declare the maximum size of arrays used to describe initial element material states.

Syntax

The following syntax is associated with the **DIMENSION MAXIMUM INITIAL** command:

DIMension MAXimum INItial STAtes ##

Explanatory Notes

The **DIMENSION MAXIMUM INITIAL** command is used to allocate sufficient memory to store the different initial material states.

Example of Command Usage

If six different initial material states are to be used in a mathematical model, enter the following commands:

```
dimension maximum initial  states 6

dim max ini sta  6
```

6.7 The DIMENSION MAXIMUM MATERIAL Commands

Synopsis

All arrays associated with APES+ whose dimensions are problem dependent are dimensioned dynamically. The **DIMENSION MAXIMUM MATERIAL** commands are used by the analyst to declare the maximum size of arrays used to store material information.

Syntax

The following syntax is associated with the **DIMENSION MAXIMUM MATERIAL** commands:

DIMension MAXimum MATerial ANIsotropic ELAstic ##

DIMension MAXimum MATerial BOUnding_SURface ISOtropic ##

DIMension MAXimum MATerial D_K GEOsynthetic ##

DIMension MAXimum MATerial FLExural ELAstic ##

DIMension MAXimum MATerial HERrmann_incompressible ELAstic ##

DIMension MAXimum MATerial HYPerbolic ELAstic ##

DIMension MAXimum MATerial INTerface STAndard ##

DIMension MAXimum MATerial ISOtropic ELAstic ##

DIMension MAXimum MATerial LING_tatsuoka GEOsynthetic ##

DIMension MAXimum MATerial ORThotropic ELAstic ##

DIMension MAXimum MATerial POROelastic ANIsotropic ##

DIMension MAXimum MATerial RAMberg-osgood LINe ##

DIMension MAXimum MATerial SPRing ##

DIMension MAXimum MATerial VARiable MODulus ##

DIMension MAXimum MATerial VIScoelastic LINe ##

Explanatory Notes

Each **DIMENSION MAXIMUM MATERIAL** command is used by the analyst to tell APES+ the total number of materials of a particular type that are to be used in a given mathematical model. The keywords associated with the **DIMENSION MAXIMUM MATERIAL** commands have the following meaning:

ANISOTROPIC ELASTIC : refers to an anisotropic elastic material idealization for solids.

BOUNDING_SURFACE ISOTROPIC : refers to the elastoplastic/viscoplastic constitutive model for isotropic cohesive soils.[53; 54; 55]

D_K GEOSYNTHETIC : refers to the the time-dependent material characterization of Dechasakulsom and Kaliakin [56] suitable for geosynthetic reinforcement.

FLEXURAL ELASTIC : refers to an isotropic elastic material idealization to be used in conjunction with frame elements.

HERRMANN_INCOMPRESSIBLE ELASTIC : refers to the isotropic material idealization developed by L. R. Herrmann [36] that is valid for both compressible and incompressible elasticity.

HYPERBOLIC ELASTIC : refers to the quasi-linear elastic model developed by Duncan and Chang [29], as well as to related variations. [73; 38; 28]

INTERFACE STANDARD : refers to a material idealization for zero-thickness interface elements.

ISOTROPIC ELASTIC : refers to an isotropic elastic material idealization for solids.

LING_TATSUOKA GEOSYNTHETIC : refers to the the time-independent material characterization of Ling and Tatsuoka [82] suitable for geosynthetic reinforcement.

ORTHOTROPIC ELASTIC : refers to an orthotropic elastic material idealization for solids.

POROELASTIC ANISOTROPIC : refers to an anisotropic linear poroelastic material idealization based on the theory of Biot [9; 10; 11].

RAMBERG-OSGOOD LINE : refers to a time-independent Ramberg-Osgood material idealization [101] suitable for use in conjunction with line (bar) elements.

VARIABLE MODULUS : refers to a quasi-elastic material idealization in which the elastic bulk modulus is a function of the mean stress. This idealization was developed to more realistically simulate the immediate (elastic) settlement of footings founded on sand. [100]

VISCOELASTIC LINE : refers to a linear, three-parameter viscoelastic material characterization suitable for use in conjunction with line (bar) elements.

Example of Command Usage

If a particular finite element model employs two orthotropic elastic, and one isotropic elastic material idealizations, enter the following commands:

```
dimension maximum material anisotropic elastic 2

dimension maximum material isotropic elastic 1
```

6.8 The DIMENSION MAXIMUM NODES Commands

Synopsis

All arrays associated with APES+ whose dimensions are problem dependent are dimensioned dynamically. The **DIMENSION MAXIMUM NODES** commands are used by the analyst to declare the maximum size of arrays used to store all nodal information.

Syntax

The following syntax is associated with the **DIMENSION MAXIMUM NODES** command:

DIMension MAXimum NODes ##

Explanatory Notes

The **DIMENSION MAXIMUM NODES** command is used to allocate sufficient memory to store all nodal information. The value "##" must equal to the *largest* node number present in the finite element model.

Example of Command Usage

If the largest node number in a mathematical model is 15,329, either of the following **DIMENSION MAXIMUM NODES** commands is appropriate:

```
dimension maximum nodes 15,329

dim max nod  15,329
```

6.9 The DIMENSION MAXIMUM SCALAR_PARAMETERS Commands

The **DIMENSION MAXIMUM SCALAR_PARAMETERS** commands are used by the analyst to declare the maximum size of arrays used to store scalar material parameter values.

Syntax

The following syntax is associated with the **DIMENSION MAXIMUM SCALAR_PARAMETERS** commands:

DIMension MAXimum SCALAR_PARAMETERS CONstant ##

Explanatory Notes

Each**DIMENSION MAXIMUM SCALAR_PARAMETERS** command is used by the analyst to tell APES+ the total number of scalar material parameter descriptions of a particular type that are to be used in a mathematical model. The keywords associated with the **DIMENSION MAXIMUM SCALAR_PARAMETERS** commands have the following meaning:

CONSTANT : refers to a scalar material parameter description with *constant* coefficients.

6.10 The DIMENSION MAXIMUM SCALAR_SOURCE Commands

The **DIMENSION MAXIMUM SCALAR_SOURCE** commands are used by the analyst to declare the maximum size of arrays used to store scalar source values.

Syntax

The following syntax is associated with the **DIMENSION MAXIMUM SCALAR_SOURCE** commands:

DIMension MAXimum SCALAR_SOURCE CONstant ##

Explanatory Notes

Each **DIMENSION MAXIMUM SCALAR_SOURCE** command is used by the analyst to tell APES+ the total number of scalar source descriptions of a particular type that are to be used in a mathematical model. The keywords associated with the **DIMENSION MAXIMUM SCALAR_SOURCE** commands have the following meaning:

CONSTANT : refers to a *constant* scalar source term.

6.11 The DIMENSION MAXIMUM SPECIFIC_HEAT Commands

Synopsis

The **DIMENSION MAXIMUM SPECIFIC_HEAT** commands are used by the analyst to declare the maximum number of specific heat types for the solid and pore fluid phase to be used in the mathematical model.

Syntax

The following syntax is associated with the **DIMENSION MAXIMUM SPECIFIC_HEAT_FLUID** commands:

DIMension MAXimum SPECIFIC_HEAT_Fluid CONstant ##

Explanatory Notes

The keywords appearing above associated with the specification of specific heat data have the following meaning:

CONSTANT : equals the maximum number of constant specific heat types used to describe the pore fluid phase.

Syntax

The following syntax is associated with the **DIMENSION MAXIMUM SPECIFIC_HEAT_SOLID** commands:

DIMension MAXimum SPECIFIC_HEAT_Solid CONstant ##

Explanatory Notes

The keywords appearing above associated with the specification of specific heat data have the following meaning:

CONSTANT : equals the maximum number of constant specific heat types used to describe the solid phase.

Example of Command Usage

If two constant specific heat descriptions are used to characterize the fluid phase, and three constant specific heat types are used for the solid phase, enter the following commands:

```
dim max specific_heat_fluid constant 2

dim max specific_heat_solid constant 3
```

6.12 The DIMENSION MAXIMUM THERMAL_CONDUCTIVITY Commands

Synopsis

The **DIMENSION MAXIMUM THERMAL_CONDUCTIVITY** commands are used by the analyst to declare the maximum size of arrays used to store thermal conductivity information.

Syntax

The **DIMENSION MAXIMUM THERMAL_CONDUCTIVITY** commands use the following syntax:

DIMension MAXimum THERMAL_Conductivity CONstant ##

Explanatory Notes

Each **DIMENSION MAXIMUM THERMAL_CONDUCTIVITY** command is used by the analyst to tell APES+ the total number of thermal conductivity descriptions of a particular type that are to be used in a given mathematical model. The keywords associated with the **DIMENSION MAXIMUM THERMAL_CONDUCTIVITY** commands have the following meaning:

CONSTANT : refers to a thermal conductivity description with *constant* coefficients.

6.13 The DIMENSION MAXIMUM THERMAL_EXPANSION Commands

Synopsis

The **DIMENSION MAXIMUM THERMAL_EXPANSION** commands are used by the analyst to declare the maximum number of coefficients of thermal expansion types for the solid and pore fluid phase to be used in the mathematical model.

Syntax

The following syntax is associated with the **DIMENSION MAXIMUM THERMAL_EXPANSION_FLUID** commands:

DIMension MAXimum THERMAL_EXPANSION_Fluid CONstant ##

Explanatory Notes

The keywords appearing above associated with the specification of coefficient of thermal expansion data have the following meaning:

CONSTANT : equals the maximum number of constant coefficients of thermal expansion used to describe the pore fluid phase.

Syntax

The following syntax is associated with the **DIMENSION MAXIMUM THERMAL_EXPANSION_SOLID** commands:

DIMension MAXimum THERMAL_EXPANSION_Solid CONstant ##

Explanatory Notes

The keywords appearing above associated with the specification of coefficient of thermal expansion data have the following meaning:

CONSTANT : equals the maximum number of constant coefficients of thermal expansion used to describe the solid phase.

Example of Command Usage

```
%dim max density_fluid constant 1

%dim max density_solid constant 2
```

Chapter 7

The ECHO Commands

Synopsis

The **ECHO** commands are used to control the verification or "echo" printing of various portions of the input data. These commands have no effect on the results of an analysis.

When a mathematical model has been verified and is being re-analyzed, use of these commands can save time for the analyst. In order to ensure the desired result, the **ECHO** commands should be the first commands entered in an input data file.

When the **ECHO ALL ON** or **ECHO ALL OFF** commands are specified, they supersede all previous **ECHO** commands that may have been issued.

Syntax

The following syntax is associated with the **ECHO** commands:

ECHo ALL [ON | OFF]

ECHo ANALYSIS_IDealization [ON | OFF]

ECHo ANALYSIS_Mesh_type [ON | OFF]

ECHo ANALYSIS_TEmporal [ON | OFF]

ECHo ANALYSIS_TYpe [ON | OFF]

ECHo BULk_modulus [ON | OFF]

ECHo DENsity [ON | OFF]

ECHo DIAgnostics [ON | OFF]

ECHo DIMensions [ON | OFF]

ECHo ELEMENT_DEScriptions [ON | OFF]

ECHo ELEments [ON | OFF]

ECHo ELEMENT_MAX_AVG_STRAin [ON | OFF]

ECHo ELEMENT_MAX_AVG_STREss [ON | OFF]

ECHo ELEMENT_MAX_AVG_VOL_strain [ON | OFF]

ECHo ELEMENT_MIN_AVG_STRAin [ON | OFF]

ECHo ELEMENT_MIN_AVG_STREss [ON | OFF]

ECHo ELEMENT_MIN_AVG_VOL_strain [ON | OFF]

ECHo ELEMENT_MAX_PREssure [ON | OFF]

ECHo ELEMENT_MAX_STRAin [ON | OFF]

ECHo ELEMENT_MAX_STREss [ON | OFF]

ECHo ELEMENT_MAX_VOL_strain [ON | OFF]

ECHo ELEMENT_TOT_VOL_strain [ON | OFF]

ECHo ELEMENT_MAX_SCAlar_flux [ON | OFF]

ECHo ELEMENT_MIN_PREssure [ON | OFF]

ECHo ELEMENT_MIN_STRAin [ON | OFF]

ECHo ELEMENT_MIN_STREss [ON | OFF]

ECHo ELEMENT_MIN_VOL_strain [ON | OFF]

ECHo ELEMENT_MIN_SCAlar_flux [ON | OFF]

ECHo FUNctions [ON | OFF]

ECHo GRAvity [ON | OFF]

ECHo HYDraulic_conductivity [ON | OFF]

ECHo INItial [ON | OFF]

ECHo MATerials [ON | OFF]

ECHo NODAL_MAximums [ON | OFF]

ECHo NODAL_MInimums [ON | OFF]

ECHo NODes [ON | OFF]

ECHo NONlinear [ON | OFF]

ECHo OPTions [ON | OFF]

ECHo QUAdrature [ON | OFF]

ECHo SCAlar [ON | OFF]

ECHo SCReen_print [ON | OFF]

ECHo SPECIFICAtions [ON | OFF]

ECHo SPECIFIC_heat [ON | OFF]

ECHo THErmal_conductivity [ON | OFF]

ECHo TITle [ON | OFF]

ECHo TRAnsformations [ON | OFF]

ECHo WARnings [ON | OFF]

Explanatory Notes

When a mathematical model has been verified and is being re-analyzed, use of these commands can save time for the analyst. In order to ensure the desired result, the **ECHO** commands should typically be the *first* commands entered in an input data file for APES. The keywords associated with the **ECHO** commands have the following meaning:

ECHO ALL ON: "echo" print all of the categories listed below. This command supersedes all previous **ECHO** commands that may have been issued.

ECHO ALL OFF: do not "echo" print any of the categories listed below. This command supersedes all previous **ECHO** commands that may have been issued. This is the default case.

ECHO ANALYSIS_IDEALIZATION: pertains to the printing of information related to the idealization (i.e., one-, two- or three-dimensional) used in performing an analysis. Details pertaining to the **ANALYSIS IDEALIZATION** command are given in Section 5.3. The default for this command is **ON**. If the **ECHO OPTIONS OFF** command has been specified, the value specified for the **ECHO ANALYSIS_IDEALIZATION** command is immaterial because the former command precludes the printing of information related to the idealization used in the analysis.

ECHO ANALYSIS_MESH_TYPE: pertains to the option used in providing information related to the mesh to be used in the analysis. Details pertaining to the **ANALYSIS MESH** commands command are given in Section 5.4. The default for this command is **ON**. If the **ECHO OPTIONS OFF** command has been specified, the value specified for the **ECHO ANALYSIS_MESH_TYPE** command is immaterial because the former command precludes the printing of information related to the mesh to be used in the analysis.

ECHO ANALYSIS_TEMPORAL: pertains to the printing of information related to the temporal integration used in the analysis. Details pertaining to the **ANALYSIS TEMPORAL** command are given in Section 5.7. The default for this command is **ON**. If the **ECHO OPTIONS OFF** command has been specified, the value specified for the **ECHO ANALYSIS_TEMPORAL** command is immaterial because the former command precludes the printing of information related to the temporal integration used in the analysis.

ECHO ANALYSIS_TYPE: pertains to the printing of information related to the type of analysis to be performed. Details pertaining to the **ANALYSIS TYPE** command are given in Section 5.9. The default for this command is **ON**. If the **ECHO OPTIONS OFF** command has been specified, the value specified for the **ECHO ANALYSIS_TYPE** command is immaterial because the former command precludes the printing of information related to the type of analysis to be performed.

BULK_MODULUS: pertains to the printing of bulk moduli values specified for the fluid and solid phase in conjunction with the analysis of solids and/or with a generalized Biot formulation for porous materials. The default for this command is **ON**.

DENSITY: pertains to the printing of mass densities specified for the fluid and solid phase. The default for this command is **ON**.

DIAGNOSTICS: pertains to diagnostic information associated with element attributes. The default for this command is **OFF**.

DIMENSIONS: pertains to the maximum dimensions specified by the analyst for dynamically dimensioned arrays. The default for this command is **ON**.

ELEMENT_DEScriptions: pertains to the printing of descriptive information associated with the description of elements. This information is supplied by the optional **DESCRIPTION** keyword. The default for this command is **ON**.

ELEMENTS: pertains to element data such as node numbers, material number, integration rule, etc. The default for this command is **ON**.

ELEMENT_MAX_AVG_STRAIN: pertains to the printing of *maximum* values of the *average* strain components (for a given element) over all elements present in the mesh in a particular solution step. This command is particularly useful for elements possessing more than one quadrature point for the computation of secondary dependent variables (e.g., Q8P0, Q9P0, Q9P3 elements, etc.). The default for this command is **OFF**.

ELEMENT_MAX_AVG_STRESS: pertains to the printing of *maximum* values of the *average* stress components (for a given element) over all elements present in the mesh in a particular solution step. This command is particularly useful for elements possessing more than one quadrature point for the computation of secondary dependent variables (e.g., Q8P0, Q9P0, Q9P3 elements, etc.). The default for this command is **OFF**.

ELEMENT_MAX_AVG_VOL_STRAIN: pertains to the printing of *maximum* value of the *average* infinitesimal volumetric strain (for a given element) over all elements present in the mesh in a particular solution step. This command is particularly useful for elements possessing more than one quadrature point for the computation of secondary dependent variables (e.g., Q8P0, Q9P0, Q9P3 elements, etc.).

In addition, the ratio of the absolute value of the *maximum average* volumetric strain to the absolute value of the minimum non-zero normal strain in the element is printed. That is,

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33}) \neq 0$$

This ratio is instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the default condition the volumetric strain and the aforementioned ratio are *not* computed and printed.

ELEMENT_MIN_AVG_STRAIN: pertains to the printing of *minimum* value of the *average* strain components (for a given element) over all elements present in the mesh in a particular solution step. This command is particularly useful for elements possessing more than one quadrature point for the computation of secondary dependent variables (e.g., Q8P0, Q9P0, Q9P3 elements, etc.). The default for this command is **OFF**.

ELEMENT_MIN_AVG_STRESS: pertains to the printing of *minimum* values of the *average* stress components (for a given element) over all elements present in the mesh in a particular solution step. This command is particularly useful for elements possessing

more than one quadrature point for the computation of secondary dependent variables (e.g., Q8P0, Q9P0, Q9P3 elements, etc.). The default for this command is **OFF**.

ELEMENT_MIN_AVG_VOL_STRAIN: pertains to the printing of *minimum* value of the *average* infinitesimal volumetric strain (for a given element) over all elements present in the mesh in a particular solution step. This command is particularly useful for elements possessing more than one quadrature point for the computation of secondary dependent variables (e.g., Q8P0, Q9P0, Q9P3 elements, etc.).

In addition, the ratio of the absolute value of the *minimum average* volumetric strain to the absolute value of the minimum non-zero normal strain in the element is printed. That is,

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33}) \neq 0$$

This ratio is instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the default condition the volumetric strain and the aforementioned ratio are *not* computed and printed.

ELEMENT_MAX_PRESSURE: pertains to the printing of *maximum* pore pressure over all elements present in the mesh in a particular solution step. For elements containing more than one pressure degree of freedom in the element (e.g., the Q9P3d and T7P3d elements), the *average* of these values is used in determining the maximum value over all the elements. The default for this command is **OFF**.

ELEMENT_MAX_STRAIN: pertains to the printing of *maximum* values of the strain components over all elements present in the mesh in a particular solution step. The default for this command is **OFF**.

ELEMENT_MAX_STRESS: pertains to the printing of *maximum* values of the stress components over all elements present in the mesh in a particular solution step. The default for this command is **OFF**.

ELEMENT_MAX_VOL_STRAIN: pertains to the printing of the *maximum* value of the infinitesimal volumetric strain over all elements present in the mesh in a particular solution step. The default for this command is **OFF**.

ELEMENT_TOT_VOL_STRAIN: pertains to the printing of the *total* average value of the infinitesimal volumetric strain (ε_{vol_tot}) over all elements present in the mesh in a particular solution step. This option is typically used for analyses in the incompressible limit, where the volumetric strain is going to zero within the solution domain. In addition, the following "domain volumetric strain ratio" (ε_{dvst}) is computed as follows:

$$\varepsilon_{dvst} = \frac{\varepsilon_{vol\_tot}}{\max\left(\sqrt{(\varepsilon_{11})^2 + (\varepsilon_{22})^2 + (\varepsilon_{33})^2}\right)}$$

The default for this command is **OFF**.

ELEMENT_MAX_SCAlar_FLUX: pertains to the printing of the *maximum* values of the components of the flux vector associated with the analysis of scalar field problems. The default for this command is **OFF**.

ELEMENT_MIN_PRESSURE: pertains to the printing of *minimum* pore pressure over all elements present in the mesh in a particular solution step. For elements containing more than one pressure degree of freedom in the element (e.g., the Q9P3d and T7P3d elements), the *average* of these values is used in determining the minimum value over all the elements. The default for this command is **OFF**.

ELEMENT_MIN_STRAIN: pertains to the printing of *minimum* values of the strain components over all elements present in the mesh in a particular solution step. The default for this command is **OFF**.

ELEMENT_MIN_STRESS: pertains to the printing of *minimum* values of the stress components over all elements present in the mesh in a particular solution step. The default for this command is **OFF**.

ELEMENT_MIN_VOL_STRAIN: pertains to the printing of the *minimum* value of the infinitesimal volumetric strain over all elements present in the mesh in a particular solution step. The default for this command is **OFF**.

ELEMENT_MIN_SCAlar_FLUX: pertains to the printing of the *minimum* values of the components of the flux vector associated with the analysis of scalar field problems. The default for this command is **OFF**.

FUNCTIONS: pertains to history function information. The default for this command is **ON**.

GRAVITY: pertains to information associated with the gravitational acceleration. The default for this command is **ON**.

HYDRAULIC_CONDUCTIVITY: pertains to the permeability information. The default for this command is **ON**.

INITIAL: pertains to the specification of the initial material state of elements. The default for this command is **ON**.

MATERIALS: pertains to material idealizations. The default for this command is **ON**.

NODAL_MAXIMUMS: pertains to the printing of maximum values of primary dependent (nodal) variables (e.g., displacements, temperatures, etc.). The default for this command is **OFF**.

NODAL_MINIMUMS: pertains to the printing of minimum values of primary dependent (nodal) variables (e.g., displacements, temperatures, etc.). The default for this command is **OFF**.

NODES: pertains to nodal coordinates. The default for this command is **ON**.

NONLINEAR: pertains to information related to nonlinear solutions. The default for this command is **ON**.

OPTIONS: pertains to general solution options (not including **DIMENSIONS**, **GRAVITY** or **NONLINEAR** information). The default for this command is **ON**.

QUADRATURE: pertains to the printing of numerical quadrature information in conjunction with the element information. If **ECHO ELEMENTS** is **OFF**, then no quadrature information will be printed. Consequently, in such cases the setting of **ECHO QUADRATURE** is immaterial. The default for this command is **ON**.

SCALAR: pertains to the printing of material parameter information associated with **SCALAR** elements. The default for this command is **ON**.

SCREEN_PRINT: pertains to the printing of the increment number and number of iterations used in a given solution step to the user's screen during the course of a solution. In very long analyses, time can be saved by setting this echo flag to **OFF**. The default for this command is **ON**.

SPECIFICATIONS: pertains to nodal specifications. The default for this command is **ON**.

SPECIFIC_HEAT: pertains to the specific heat information. The default for this command is **ON**.

THERMAL_CONDUCTIVITY: pertains to the thermal conductivity information. The default for this command is **ON**.

TITLE: pertains to the descriptive alphanumeric information associated with a solution. The default for this command is **ON**.

TRANSFORMATIONS: pertains to coordinate transformation information. The default for this command is **ON**.

WARNINGS: pertains to warning messages that are printed in the course of evaluating element attributes. The default for this command is **ON**.

Example of Command Usage

If an "echo" print of the gravity information and initial state information associated with element definitions is not desired, enter the commands:

```
echo  grav  off

echo  initial  off
```

Chapter 8

The FINISHED Commands

The **FINISHED** commands are used to denote the end of the description of the option **SETTINGS**, the input **DATA** and the **LOADING** histories. Thus, in order to begin the input of data describing the mathematical model, the **FINISHED SETTINGS** command must first be specified. Likewise, when all data describing the mathematical model has been input, the **FINISHED DATA** command must be specified. Finally, when the loading histories have all been described, the **FINISHED LOADING** command must be issued. This signals the end of the analysis.

The following **FINISHED** commands are used by the analyst to delineate various portions of the input data file:

- The **FINISHED DATA** command (see Section 8.1).
- The **FINISHED LOADING** command (see Section 8.2).
- The **FINISHED SETTINGS** command (see Section 8.3).

This section gives additional details pertaining to the above commands.

8.1 The FINISHED DATA command

Synopsis

The **FINISHED DATA** command must be specified in order to indicate that all of the information required to describe the mathematical model has been supplied. This command only applies to the pre-processor APES+.

Syntax

The following syntax is associated with the **FINISHED DATA** command:

FINished DATa

Explanatory Notes

Upon issuing the **FINISHED DATA** command, a check of the data used to describe the mathematical model is performed by APES+ and, if no errors have been detected in the input data, the model is generated. All subsequent information in a data file shall then be used to describe the nature of the solution to be performed.

Example of Command Usage

Once the model description information is complete, enter either of the following commands:

```
finished data

fin dat
```

The following snippet of an input data file shows the proper use of the **FINISHED DATA** command:

```
analysis action analyze
analysis type solid
analysis mesh internal
analysis idealization axisymmetric
analysis temp transient
analysis options finished
!
```

```
integration  time  parameter = 0.50
!
dim max material isotropic elastic 1
dim max nodes = 41
dim max t15P0 = 4
!
echo gravity off
!
finished settings
!
material elastic  isotropic  number 1  &
  descrip " linear elastic material idealization" modulus  30000.0  poissons  0.30
!
node  line  number  1
node  line  number  5  x1  1.00           incr  1
node  line  number 41  x1  1.00  x2  2.0  incr  9
node  line  number 37            x2  2.0  incr -1
node  line  number -1                     incr -9
!
node  line  number 21  x1  0.50  x2  1.0
!
element irreducible type "t15p0"  nodes  1 5 21 3 13 11 2 9 16 4 17 6 7 8 12 mat 1 &
            dont_print_stress dont_print_strain  print_avg_stress
!
element irreducible type "t15p0"  nodes  5 41 21 23 31 13 14 36 17 32 26 9 18 27 22 mat
            dont_print_stress dont_print_strain  print_avg_stress
!
element irreducible type "t15p0"  nodes  1 21 37 11 29 19 6 25 28 16 33 10 15 20 24  mat
            dont_print_stress dont_print_strain  print_avg_stress
!
element irreducible type "t15p0"  nodes  21 41 37 31 39 29 26 40 33 36 38 25 30 35 34 ma
            dont_print_stress dont_print_strain  print_avg_stress
!
specification conc mech nodes  41   1_dis 2_hist 0  2_force 2_value  -7.778
specification conc mech nodes  40         2_hist 0  2_force 2_value  -26.667
specification conc mech nodes  39         2_hist 0  2_force 2_value  -6.667
specification conc mech nodes  38         2_hist 0  2_force 2_value  -8.889
specification conc mech nodes  37   1_dis 2_hist 0  2_force 2_value   0.0
!
specification conc mech  nodes 10:28:9   1_dis
specification conc mech  nodes 14:32:9   1_dis
specification conc mech  nodes  1:5:1    1_dis 2_dis
!
finish data
!
```

8.2 The FINISHED LOADING command

Synopsis

The **FINISHED LOADING** command must be specified in order to indicate that all of the information required to describe the solution steps has been supplied in the data file.

Syntax

The following syntax is associated with the **FINISHED LOADING** command:

FINished LOAding

Explanatory Notes

The **FINISHED LOADING** command must be specified in order to indicate that all of the information required to describe the solution steps has been supplied. Upon issuing the command, the solution begins.

Example of Command Usage

Once the information describing the solutions history is complete, enter either of the following commands:

```
finished loading

fin load
```

The following snippet of an input data file shows the proper use of the **FINISHED LOADING** command (as well as the **FINISHED SETTINGS** and **FINISHED DATA** commands):

```
ana title  "patch test 'C' for quadrilateral mesh"
ana title  "after MacNeal & Harder (1985)"
!
!  sig_11 = 2250.0 ; sig_22 = -1750.0 are applied
!
!  under plane stress conditions this leads to the following:
!  eps_11 = 2.688e-03, eps_22 = -2.313e-03, gam_12 = 0.0, and
!  eps_33 = - nu*(sigma_11 + sigma_22)/E = -1.250e-04
```

```
!
echo  func off
echo  grav off
echo  ini  off
echo  warn off
!
analysis action analyze
analysis type solid
analysis idealization plane_stress
analysis temp transient
analysis options finished
!
integration time parameter 0.50
!
dim max material isotropic elastic 1
dim max nodes 25
dim max q8p0 = 5
!
finished settings
!
mat elastic isotropic number 1  desc " test 1 "  mod = 1.0e+06  poisson 0.25
!
nodes  line number 1  x1  0.04  x2  0.02
nodes  line number 2  x1  0.18  x2  0.03
nodes  line number 3  x1  0.16  x2  0.08
nodes  line number 4  x1  0.08  x2  0.08
nodes  line number 5  x1  0.110  x2  0.025
nodes  line number 6  x1  0.170  x2  0.055
nodes  line number 7  x1  0.120  x2  0.080
nodes  line number 8  x1  0.060  x2  0.050
nodes  line number 9  x1  0.115  x2  0.053
nodes  line number 10  x1  0.00  x2  0.00
nodes  line number 11  x1  0.24  x2  0.00
nodes  line number 12  x1  0.24  x2  0.12
nodes  line number 13  x1  0.00  x2  0.12
nodes  line number 14  x1  0.120  x2  0.000
nodes  line number 15  x1  0.240  x2  0.060
nodes  line number 16  x1  0.120  x2  0.120
nodes  line number 17  x1  0.000  x2  0.060
nodes  line number 18  x1  0.020  x2  0.010
nodes  line number 19  x1  0.210  x2  0.015
nodes  line number 20  x1  0.200  x2  0.100
nodes  line number 21  x1  0.040  x2  0.100
nodes  line number 22  x1  0.115  x2  0.013
nodes  line number 23  x1  0.205  x2  0.058
```

```
nodes  line number 24  x1  0.120  x2  0.100
nodes  line number 25  x1  0.030  x2  0.055
!
element irreducible type "q8p0" nodes 1  10  11  2  18  14  19  5   &
       mat  1 PRINT_AVG_STRAIN PRINT_AVG_STRESS
element irreducible type "q8p0" nodes 2  11  12  3  19  15  20  6   &
       mat  1 PRINT_AVG_STRAIN PRINT_AVG_STRESS
element irreducible type "q8p0" nodes 3  12  13  4  20  16  21  7   &
       mat  1 PRINT_AVG_STRAIN PRINT_AVG_STRESS
element irreducible type "q8p0" nodes 1   4  13  10  8  21  17  18  &
       mat  1 PRINT_AVG_STRAIN PRINT_AVG_STRESS
element irreducible type "q8p0" nodes 1   2   3   4  5   6   7   8   &
       mat  1 PRINT_AVG_STRAIN PRINT_AVG_STRESS
!
specification line quad mech node_b 11  node_end 12 1_incr 1 2_incr 4  1_hist 0  2_hist
    np_begin -2250.0  np_end  -2250.0
specification line quad mech node_b 12  node_end 13 1_incr 1 2_incr 4  1_hist 0  2_hist
    np_begin 1750.0  np_end  1750.0
!
specification conc mech nodes 10 1_dis   2_dis
specification conc mech nodes 11         2_dis     1_his 0
specification conc mech nodes 13 1_dis             2_his 0
specification conc mech nodes 14         2_dis
specification conc mech nodes 17 1_dis
!
finished data
!
solution  time  final  1.0  increments  1  output 1:10:1
!
finished load
!
```

8.3 The FINISHED SETTINGS command

Synopsis

The **FINISHED SETTINGS** command must be specified in order to indicate that the solution control information is complete. Once this is done, all arrays required in the analysis are dynamically allocated.

Syntax

The following syntax is associated with the **FINISHED SETTINGS** command:

FINished SETtings

Explanatory Notes

The **FINISHED SETTINGS** command must be specified in order to indicate that the solution control information is complete. Upon issuing the command, a check of this information is performed. All subsequent information in a data file shall then be used to describe the mathematical model under consideration, and the nature of the solution to be performed.

Example of Command Usage

Once the solution control information is complete, enter either of the following commands:

```
finished settings

fin set
```

The following snippet of an input data file shows the proper use of the **FINISHED SETTINGS** command:

```
echo  func off
echo  grav off
echo  ini  off
echo  warn off
!
analysis action analyze
analysis description linear
```

```
analysis type solid
analysis idealization plane_stress
analysis temp transient
analysis options finished
!
integration time parameter 0.50
!
dim max material isotropic elastic 1
dim max nodes 8
dim max q4p0 = 5
!
finished settings
!
```

Chapter 9

The GRAVITY Commands

The gravitational acceleration and its line of action are specified using the **GRAVITY ACCELERATION VALUE** and the **GRAVITY X1_ANGLE**, **GRAVITY X2_ANGLE**, **GRAVITY X3_ANGLE** commands, respectively. The associated history functions (refer also to the discussion of **FUNCTION** commands given in Chapter 12) are specified using the **GRAVITY ACCELERATION HISTORY** and **GRAVITY ANGLE HISTORY** commands.

The following **GRAVITY** commands are available for use by the analyst:

- The **GRAVITY ACCELERATION HISTORY** command (see Section 9.1).
- The **GRAVITY ACCELERATION VALUE** command (see Section 9.2).
- The **GRAVITY ANGLE** commands (see Section 9.3).
- The **GRAVITY ANGLE HISTORY** command (see Section 9.4).

This chapter gives additional details pertaining to the above commands.

9.1 The GRAVITY ACCELERATION HISTORY command

Synopsis

The **GRAVITY ACCELERATION HISTORY** command is used to specify the history function associated with the gravitational acceleration.

Syntax

The following syntax is associated with the **GRAVITY ACCELERATION HISTORY** command:

GRAvity ACCeleration HIStory #

Explanatory Notes

- The gravitational acceleration is used to compute body forces of the materials being analyzed.
- Using the **GRAVITY ACCELERATION HISTORY** command, a pre-existing gravity loading of a field deposit can be modeled by initializing the stresses and pore pressure to their proper values (refer to the **INITIAL** commands) and setting **GRAVITY ACCELERATION HISTORY** and **GRAVITY ANGLE HISTORY** equal to -3. The history of the effective gravity loading on a centrifuge model during "spin-up" can be modeled by describing, via the **FUNCTION** commands, a history function corresponding to the centrifuge velocity for the test; in the case of a fixed bucket, both the gravitational acceleration and its line of action would vary with time, while for a "swing-up" bucket only the acceleration would vary.
- The default gravitational acceleration history is equal to -2.

Example of Command Usage

To specify a gravitational acceleration history function number 3, enter the following commands:

```
grav accel his  3
```

9.2 The GRAVITY ACCELERATION VALUE command

Synopsis

The **GRAVITY ACCELERATION VALUE** command is used to specify the value of the gravitational acceleration.

Syntax

The following syntax is associated with the **GRAVITY ACCELERATION VALUE** command:

GRAvity ACCeleration VALue #.#

Explanatory Notes

- The gravitational acceleration is used to compute body forces of the materials being analyzed.
- The **GRAVITY ACCELERATION VALUE** can be input either in terms of units appropriate for the system of units selected for the problem (e.g., 32.2 ft/sec^2 for English units or 9.81 m/s^2 for metric units), or in terms of multiples of the acceleration of gravity at sea level (i.e., $g = 1.0$ for a field structure). In the first case the solid and fluid densities specified the **DENSITY SOLID** and **DENSITY FLUID** commands (see Chapter 17) would represent mass densities, while in the latter they would represent unit weights; in all cases, the products of the densities and the gravitational acceleration specified via a **GRAVITY ACCELERATION VALUE** command must have units of weight per unit volume.
- The default gravitational acceleration is equal to 0.0.

Example of Command Usage

To specify a gravitational acceleration equal to 32.2, enter the following commands:

```
grav accel value  32.2
```

9.3 The GRAVITY ANGLE commands

Synopsis

The **GRAVITY ANGLE** command is used to specify the angle that the gravitational acceleration vector makes with respect to each of the global coordinate axes.

Syntax

The following syntax is associated with the **GRAVITY ANGLE** command:

GRAvity X1_Angle VALue #.# (DEGrees)
GRAvity X2_Angle VALue #.# (DEGrees)
GRAvity X3_Angle VALue #.# (DEGrees)

Explanatory Notes

- The line of action, in space, of the gravitational acceleration vector is described by three angles. These angles are measured from the positive global x_1, x_2, and x_3-coordinate directions. The **GRAVITY ANGLE** commands are used to specify the magnitudes of these angles. The angles, which are measured from the respective positive coordinate directions, must be specified in *degrees*.

- Mathematically, the gravitational acceleration vector is given by

$$\mathbf{g} = g\left(\cos\theta_1 \mathbf{e_1} + \cos\theta_2 \mathbf{e_2} + \cos\theta_3 \mathbf{e_3}\right)$$

  where the $\mathbf{e_i}\,(i = 1, 2, 3)$ represent unit vectors along the three coordinate directions.

- For two-dimensional analyses only the **GRAVITY X1_ANGLE** and **GRAVITY X2_ANGLE** commands are applicable.

- The default values are: -90.0, 180.0, and -90.0 for the angles that the gravitational acceleration makes with the global x_1, x_2 and x_3-coordinate axes, respectively.

- It is the responsibility of the user to ensure that the angles specified are consistent; i.e., that

$$\left(\cos\theta_1\right)^2 + \left(\cos\theta_2\right)^2 + \left(\cos\theta_3\right)^2 = 1$$

Nonetheless, APES+ checks the above equation and, if the result differs from 1.0, prints a warning.

Example of Command Usage

In a particular two-dimensional analysis, the gravitational angle with respect to the x_1-axis is equal to 32.8 degrees. The appropriate commands for this case are

```
gravity x1_angle  32.8
gravity x2_angle  57.2
```

9.4 The GRAVITY ANGLE HISTORY command

Synopsis

The **GRAVITY ANGLE HISTORY** command is used to specify the history function associated with the gravitational acceleration.

Syntax

The following syntax is associated with the **GRAVITY ANGLE HISTORY** command:

GRAvity ANGle HIStory #

Explanatory Notes

The **GRAVITY ANGLE HISTORY** command provides the analyst with the ability to account for changing magnitudes of gravitational acceleration. The prime example of such changes is the "spin up" phase of a centrifugal experiment, in which the gravitational acceleration is continuously increased up to the final desired value.

Example of Command Usage

To specify a gravitational angle history function number 1, enter either of the following commands:

```
gravitational angle history  1
gra ang  his  1
```

Chapter 10

The INTEGRATION Commands

Synopsis

The **INTEGRATION** command determines the approximation used for the time derivatives in the governing equations.

Syntax

The following syntax is associated with the **INTEGRATION** command:

INTegration TIMe PARameter (##.#)

Explanatory Notes

- The optional numeric (real) value specified in conjunction with the **INTEGRATION** command determines the approximation used for the time derivatives in the governing equations [44]. This value must be in the range $0.0 \leq \text{value} \leq 1.0$.
- For values greater than or equal to 0.50, the temporal integration algorithm will be *unconditionally stable* [44].
- For a value of 0.50 (corresponding to the Crank-Nicholsen method) the algorithm will be *second-order* accurate; for all other values the algorithm will only be *first-order* accurate.
- The temporal integration algorithm will be *oscillation-free* for a value of 1.0 (Backward Euler scheme) [44].

- Values between 0.5 and 0.67 (corresponding to the so-called "Galerkin" method) are recommended.
- The default value is 0.67.

Example of Command Usage

To specify the Crank-Nicholsen method for numerical integration in time, enter either of the following commands:

```
integration  time  parameter  0.50

int  tim  par  0.50
```

Chapter 11

The NONLINEAR Commands

The following **NONLINEAR** commands are used to specify options related to nonlinear analyses:

- The **NONLINEAR ACCELERATION CONVERGENCE** command (see Section 11.1).
- The **NONLINEAR ACCELERATION FACTOR** command (see Section 11.2).
- The **NONLINEAR ERROR TOLERANCE** command (see Section 11.3).
- The **NONLINEAR ITERATION LIMIT** command (see Section 11.4).
- The **NONLINEAR REFORM STIFFNESS** command (see Section 11.5).
- The **NONLINEAR SCHEME** command (see Section 11.6).

This chapter gives additional details pertaining to the above commands.

11.1 The NONLINEAR ACCELERATION CONVERGENCE Command

Synopsis

The **NONLINEAR ACCELERATION CONVERGENCE** command controls the use of convergence acceleration factors applied to the components of the solution vector.

Syntax

The following syntax is associated with the **NONLINEAR ACCELERATION CONVERGENCE** command:

NONlinear ACCeleration CONvergence [ON | OFF]

Explanatory Notes

Additional details concerning convergence acceleration factors applied to the components of the solution vector can be found in Isaacson and Keller [45]. The default condition is no acceleration.

Example of Command Usage

To specify the acceleration option, enter either of the following commands:

```
nonlinear acceleration  convergence  on

non acc  con  on
```

11.2 The NONLINEAR ACCELERATION FACTOR Command

Synopsis

The **NONLINEAR ACCELERATION FACTOR** command specifies the parameter to be used in the convergence acceleration scheme.

Syntax

The following syntax is associated with the **NONLINEAR ACCELERATION FACTOR** command:

NONlinear ACCeleration FACtor #.#

Explanatory Notes

- If the analyst has specified **NONLINEAR ACCELERATION CONVERGENCE OFF**, then the factor specified via the **NONLINEAR ACCELERATION FACTOR** command is *immaterial*.
- The default value for the convergence acceleration factor is 0.30.

Example of Command Usage

To specify an acceleration factor of 0.45, enter either of the following commands:

```
nonlinear acceleration factor  0.45

non acc fac  0.45
```

11.3 The NONLINEAR ERROR TOLERANCE Command

Synopsis

The **NONLINEAR ERROR TOLERANCE** command specifies the tolerance that is to be used to ascertain when convergence has been realized in a given solution increment.

Syntax

The following syntax is associated with the **NONLINEAR ERROR TOLERANCE** command:

NONlinear ERRor TOLerance #.#

Explanatory Notes

- Convergence is measured by the relative change of the *L2* norm of the solution vector between successive iterations.
- For most problems, values of the error tolerance in the range 0.010 to 0.005 are adequate.
- The default value for the tolerance is 0.010.

Example of Command Usage

To specify an convergence error tolerance of 0.025, enter either of the following commands:

```
nonlinear error  tolerance   0.025

non  err tol   0.025
```

11.4 The NONLINEAR ITERATION LIMIT Command

Synopsis

The **NONLINEAR ITERATION LIMIT** command specifies the maximum number of iterations in which convergence can be realized in a given solution increment.

Syntax

The following syntax is associated with the **NONLINEAR ITERATION LIMIT** command:

NONlinear ITEration LIMit ##

Explanatory Notes

- The value of the iteration limit controls the maximum number of iterations permitted in any single solution increment.
- For initial analyses of a problem, a value in the range 5 to 10 is usually appropriate.
- The default value for the iteration limit is five (5).

Example of Command Usage

To specify a limit of 10 iterations per solution increment, enter either of the following commands:

```
nonlinear iteration  limit   10

non ite  lim   10
```

11.5 The NONLINEAR REFORM STIFFNESS Command

Synopsis

The **NONLINEAR REFORM STIFFNESS** command specifies the frequency of updating the incremental stiffness matrix in the course of an incremental solution.

Syntax

The following syntax is associated with the **NONLINEAR REFORM STIFFNESS** command:

NONlinear REForm (STIffness) EVEry ##

Explanatory Notes

- The frequency of updating the stiffness matrix during the iteration process is controlled by the value specified in the **NONLINEAR REFORM STIFFNESS** command.
- The default value for the stiffness updating is one (1). For initial analyses, this default value is appropriate.

Example of Command Usage

To reform the stiffness matrix every 3 iterations, enter either of the following commands:

```
nonlinear reform stiffness  every  3

non ref sti  eve  3
```

11.6 The NONLINEAR SCHEME Command

Synopsis

The **NONLINEAR SCHEME** command specifies the algorithm to be used in an incremental solution.

Syntax

The following syntax is associated with the **NONLINEAR SCHEME** command:

NONlinear SCHeme [TANgent (STIffness) | SUCcessive (APProximations) | PARameter #.#]

Explanatory Notes

- The value of the **NONLINEAR SCHEME PARAMETER** determines the approximation to be used for the Jacobian in the Newton-Raphson iteration for the nonlinear problem [93].
- The **TANGENT STIFFNESS** method corresponds to a parameter of 0.0; this is the default condition.
- For the method of **SUCCESSIVE APPROXIMATIONS**, the parameter should be set equal to 1.0.

Example of Command Usage

To specify the method of successive approximations, enter either of the following commands:

```
nonlinear scheme  suc  approx

non sch  parameter  1.0
```

Chapter 12

The FUNCTION Commands

The history (time) dependence of all input quantities (i.e., the gravitational acceleration, the angle defining its line of action, specified nodal displacements, loads, flows, pressures, temperatures, etc.) is specified by means of appropriate "history functions."

The following information pertains to the **FUNCTION** commands:

- General information pertaining to **FUNCTION** commands (see Section 12.1).
- The **FUNCTION EXPLICIT** command (see Section 12.2).
- The **FUNCTION FROM** command (see Section 12.3).

12.1 History FUNCTION commands: An Overview

Synopsis

The history (time) dependence of all input quantities (i.e., the gravitational acceleration, the angle defining its line of action, specified nodal displacements, loads, flows, pressures, temperatures, etc.) is specified by means of appropriate "history functions."

Four such functions are *built into* the APES+ program; as such, these require no specifications of the **FUNCTION** commands. These functions are assigned numbers -3, -2, -1 and 0. They are defined in the following manner:

Function number -3 : This specifies a unit value and a zero incremental value for all times (Figure 12.1).

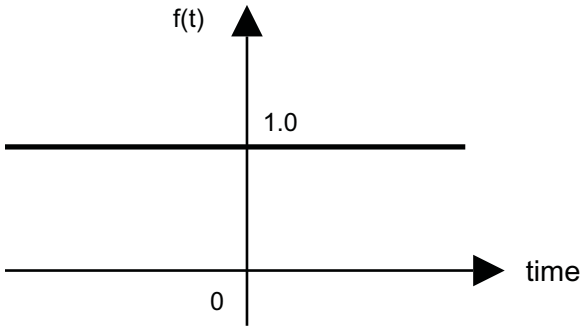

Figure 12.1: Pre-defined history function number -3.

Function number -2 : This specifies a zero value and a zero incremental value for all times (Figure 12.2).

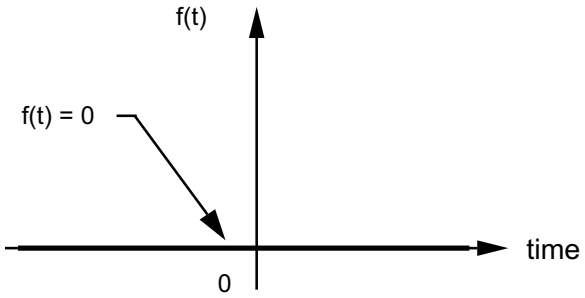

Figure 12.2: Pre-defined history function number -2.

Function number -1 : This specifies all incremental values equal to 1.0. The incremental values are taken to be *equal* regardless of the relative lengths of the time steps specified using the **SOLUTION TIME** command. The resulting history functions for the cases of equal and variable length time steps are illustrated in Figure 12.3.

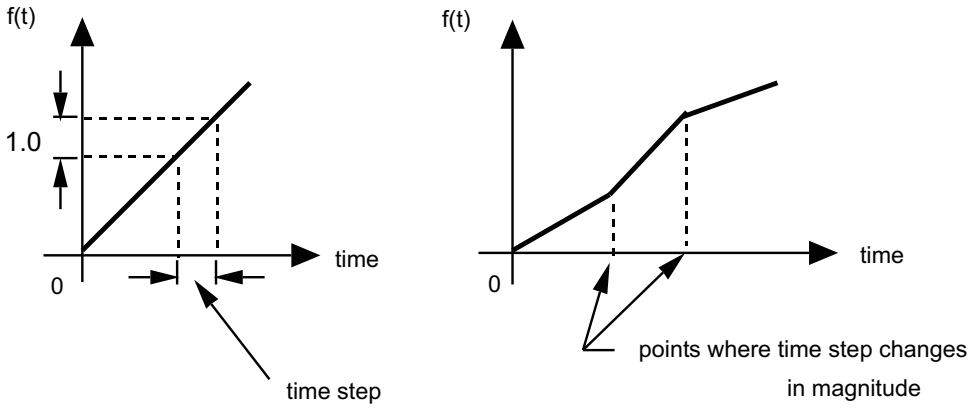

Figure 12.3: Pre-defined history function number -1.

Function number 0 : This specifies a step function at time $t = 0$; i.e., a quantity using this function is applied entirely during the first solution increment (Figure 12.4).

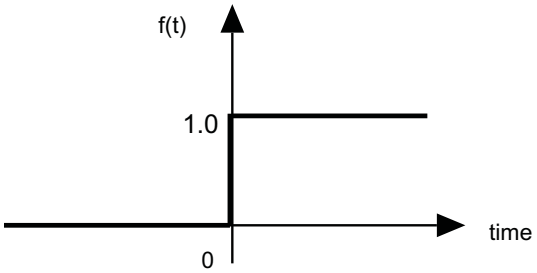

Figure 12.4: Pre-defined history function number 0.

In addition, the user may, by means of the **FUNCTION EXPLICIT** or **FUNCTION FROM** commands, describe as many more history functions (numbered 1,2, . . .) as needed.

For a particular history function, linear interpolation is used to identify the increment ΔF that corresponds to a given time increment Δt. For solution times beyond the last specified point (t_{numprs}), the final linear segment is extended indefinitely. If a magnitude V and a history function number *num* are specified for some given external quantity (i.e.; the gravitational acceleration, the angle defining its line of action, specified nodal displacements, loads, flows, pressures, etc.), then in the solution interval Δt an increment of the quantity equal to $V\Delta F$ is applied (see Figure 12.5). Here ΔF represents the increment in ordinate value corresponding to history function *num* and the time interval Δt.

If no history function specifications are required, the **FUNCTION** commands should be omitted entirely.

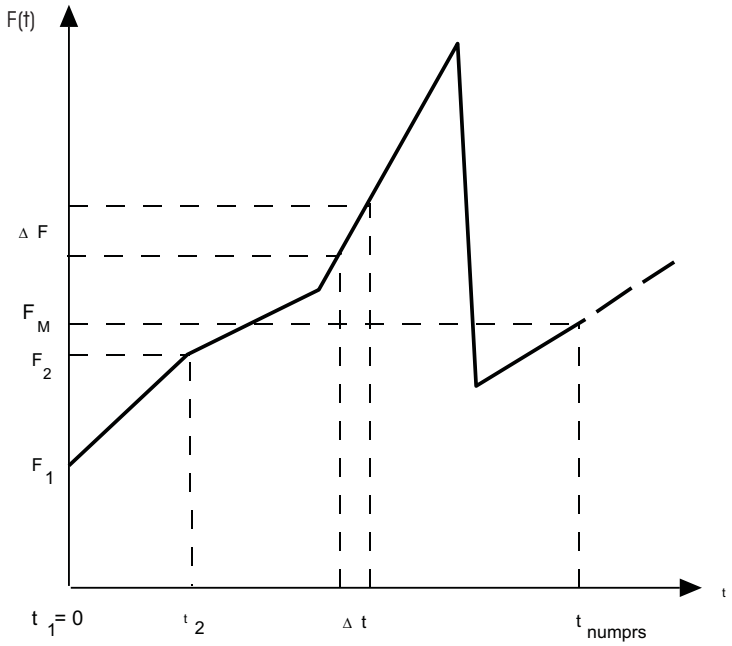

Figure 12.5: Hypothetical user-specified history function.

12.2 History FUNCTION EXPLICIT command: An Overview

Synopsis

The **FUNCTION EXPLICIT** command is used to *explicitly* define the history function to be used.

Syntax

The following syntax is associated with the **FUNCTION EXPLICIT** command:

FUNction EXPlicit NUMber ## PAIrs ##

Explanatory Notes

The **NUMBER** sub-command refers to the number assigned to the history function. This integer number must be greater than or equal to one.[1] The **PAIRS** sub-command denotes the number of abscissa-ordinate pairs that comprise this history function. Beginning on the record immediately following the **FUNCTION EXPLICIT** command, all of the abscissa-ordinate pairs that comprise this history function must be provided; these are read in a list-directed (free form) manner.

Example of Command Usage

If history function number 3 consists of 42 abscissa-ordinate pairs, enter the following command:

```
function explicit number 3 pairs 42
   0.0   0.0
  10.0   0.0
  11.0   0.156
  12.0   0.309
  13.0   0.454   14.0   0.588   15.0   0.707   16.0   0.809
  17.0   0.891   18.0   0.951   19.0   0.988   20.0   1.0
```

[1] As mentioned above, history functions numbered -3, -2, -1, and 0 are explicitly defined in the program and thus require no input by the analyst.

```
 21.0   0.988   22.0   0.951   23.0   0.891   24.0   0.809
 25.0   0.707   26.0   0.588   27.0   0.454   28.0   0.309
 29.0   0.156   30.0   0.0     31.0  -0.156   32.0  -0.309
 33.0  -0.454   34.0  -0.588   35.0  -0.707   36.0  -0.809
 37.0  -0.891   38.0  -0.951   39.0  -0.988   40.0  -1.0
 41.0  -0.988   42.0  -0.951   43.0  -0.891   44.0  -0.809
 45.0  -0.707   46.0  -0.588   47.0  -0.454   48.0  -0.309
 49.0  -0.156   50.0   0.0
```

The history function shown in Figure 12.6 is to be used in an analysis. It will be assigned the number *two* (2).

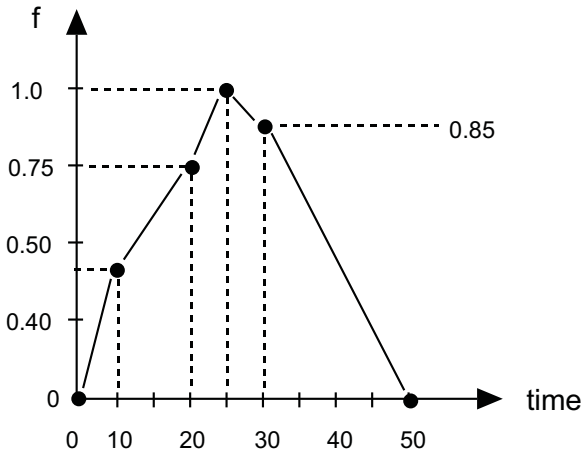

Figure 12.6: Sample time history function.

Using the **FUNCTION EXPLICIT** command, the appropriate history function options are specified in the following manner:

```
function explicit   number 2  pairs  6
!
   0.0 0.0  10.0 0.40  20.0  0.75  25.0  1.0
   30.0  0.85  50.0    0.0
```

This history function could have likewise been specified using the **FUNCTION FROM** command, with the above numerical values (including the history number and number of pairs) provided in an ASCII file.

12.3 History FUNCTION FROM command: An Overview

Synopsis

The **FUNCTION FROM** command instructs the APES+ program to read the history function to be used in an analysis from the ASCII file name "string". The history function contained in the file must be complete.The default **FILENAME** is *func.dat*.

Syntax

The following syntax is associated with the **FUNCTION FROM** command:

FUNction FROm FILename "string"

Explanatory Notes

The data in this file must contain the following values, which are input in a list directed (free-format) manner:

```
num   numprs

abscissa_1   ordinate_1    abscissa_2   ordinate_2
. . . . .
. . . . .
abscissa_numprs   ordinate_numprs
```

In the above data, *num* represents the number of the particular history function, *numprs* denotes the number of abscissa-ordinate pairs contained in the file. Following these parameters, and beginning on a new record, all of the abscissa-ordinate pairs that make up the history function are given.

Example of Command Usage

If data completely describing a history function is contained in the file `sine_42.dat`, enter either of the following commands:

```
function from filename ‘‘sine_42.dat’’ or

fun fro fil ‘‘sine_42.dat’’
```

Chapter 13

The GENERATE Commands

Synopsis

The **GENERATE SURFACES** command is used to determine the coordinates of nodes comprising two-dimensional surfaces consisting of quadrilateral "patches" of elements.

Syntax

The following syntax is associated with the **GENERATE SURFACES** command:

GENerate SURfaces

Explanatory Notes

For two-dimensional analyses, once the **GENERATE SURFACES** command has been issued, all unknown nodal coordinates in the mesh will be determined; as such, the mathematical model is complete with respect to the node and element determination. It follows that for two-dimensional analyses, the **GENERATE SURFACES** command can only be issued after all of the **ELEMENT** and **NODES** commands have been specified.

For three-dimensional analyses, the **GENERATE SURFACES** command must be followed by suitable **ELEMENT** commands (discussed above) that define/generate the coordinates of all nodes contained within the interior of the three-dimensional solution domain.

NOTE: If the coordinates of all nodes in a given two- or three-dimensional mesh have been explicitly specified (e.g., from a meshing pre-processor), there is no need to specify the **GENERATE SURFACES** command. Indeed, in the interest of computational efficiency this command should *not* be issued in such instances.

Without any prompting of the user the interior node point generation option automatically determines the location of all interior nodes whose coordinates have not been established through the options cited above (that is, all node points left undefined after **NODES CIRCLE** and/or **NODES LINE** commands have been processed; note that all boundary nodes must be directly or indirectly specified by the input). The locations of the undefined interior nodes are computed by means of the "Isoparametric" mesh generation scheme described by Herrmann [37] and subsequently generalized by Kaliakin [49].

Example of Command Usage

In general, to determine the unknown nodal coordinates in all two-dimensional surfaces that comprise a particular model, specify either of the following commands:

```
generate surfaces
```

```
gen sur
```

Two meshes prepared with the aid of the mesh generation schemes are shown below. Mesh number 1, shown in Figure 13.1, was developed using the straight line generation scheme to specify the exterior (boundary) nodes, and the interior generation scheme to determine the coordinates of the remaining nodes. Only the following records are required to construct the mesh (the nodes along each line are equally spaced):

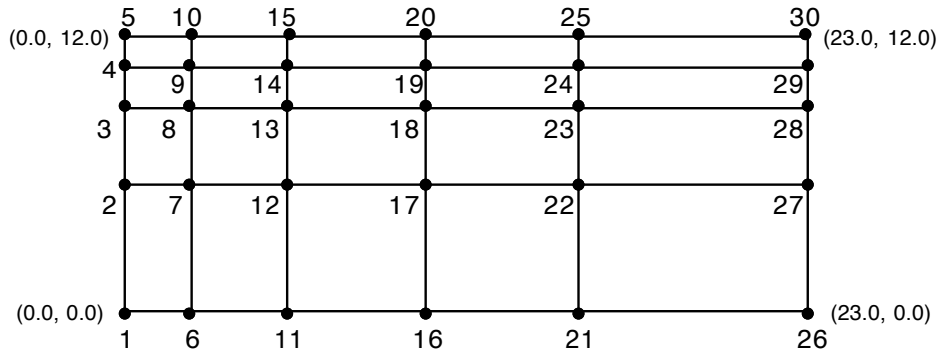

Figure 13.1: Hypothetical mesh number 1.

```
node line num  1
node line num  26   x1 23.0              inc  5  ratio  1.25
node line num  30   x1 23.0  x2  12.0   inc  1  ratio  0.50
node line num   5             x2  12.0  inc -5  ratio  0.80
node line num  -1                       inc -1  ratio  2.0
```

Mesh number 2, shown in Figure 13.2, was developed in a similar manner except that the straight line generation option was also used to define the nodes lying along the material

interface; i.e., along line from node 3 to 33. In all cases the nodes along a line are equally spaced. Only the following records are thus required to generate the mesh (#.# denotes suitable coordinates for the respective nodes):

```
node lin num  1  x1 #.#  x2  #.#
node lin num 31  x1 #.#  x2  #.#  incr  6
node lin num 33  x1 #.#  x2  #.#  incr  1
node lin num  3  x1 #.#  x2  #.#  incr -6
node lin num -1                   incr -1
node lin num 34  x1 #.#  x2  #.#
node lin num 28  x1 #.#  x2  #.#
node lin num 30  x1 #.#  x2  #.#
node lin num  6  x1 #.#  x2  #.#  incr -6
node lin num -3                   incr -1
```

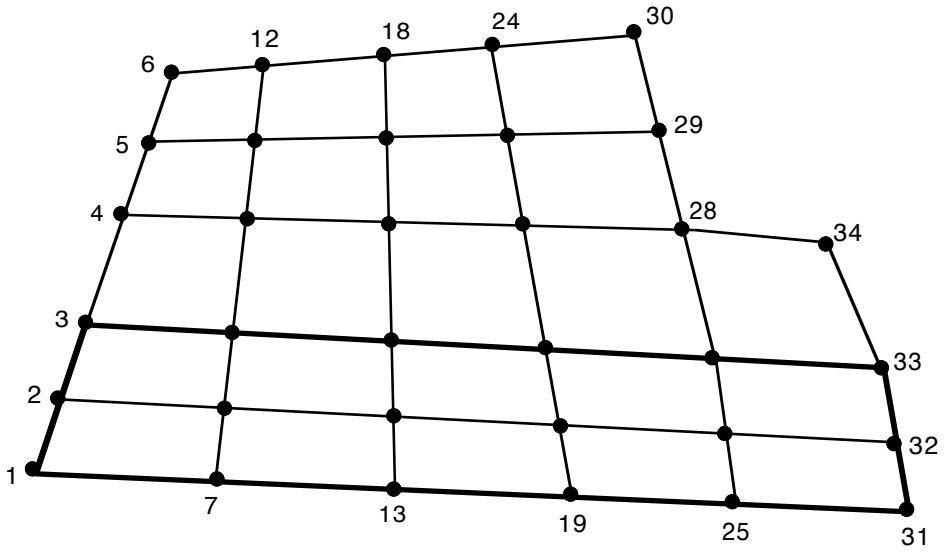

Figure 13.2: Hypothetical mesh number 2.

Although the above discussion regarding nodal generation has dealt with four-node quadrilateral elements, the schemes described are equally applicable to eight- and/or nine-node quadrilaterals, or to surfaces of hexahedral elements.

Chapter 14

The HYDRAULIC CONDUCTIVITY Commands

This chapter contains information associated with the specification of permeability coefficients for finite element analyses using the APES$^+$ computer program. It consists of the following sections:

- General information related to **HYDRAULIC CONDUCTIVITY** commands (see Section 14.1).
- The **HYDRAULIC CONDUCTIVITY CONSTANT** command (see Section 14.2).
- The **HYDRAULIC CONDUCTIVITY EXPONENTIAL** command (see Section 14.3).
- The **HYDRAULIC CONDUCTIVITY LOG-LOG** command (see Section 14.4).
- The **HYDRAULIC CONDUCTIVITY POWER** command (see Section 14.5).
- The **HYDRAULIC CONDUCTIVITY SEMI-LOG** command (see Section 14.6).

14.1 HYDRAULIC CONDUCTIVITY commands: An Overview

Synopsis

The **HYDRAULIC CONDUCTIVITY** commands are used to describe the idealizations used to characterize the flow of pore fluid in **FLOW_POROMECHANICAL** and in **FLOW_PORO_THERMAL** analyses. The following points apply to all **HYDRAULIC CONDUCTIVITY** commands:

- The value associated with the **NUMBER** keyword is independent of the numbers given to specific material idealizations; this value must, however, be different for each permeability description.
- Associated with the **DESCRIPTION** keyword is a string that is used to describe the permeability idealization. This is included solely for the benefit of the analyst; the associated string is printed in the "echo" of the **HYDRAULIC CONDUCTIVITY** data.
- The permeability coefficients are assigned to specific elements through the **HYDRAULIC CONDUCTIVITY** keywords associated with **ELEMENT MIXED_FLOW** commands.
- The **HYDRAULIC CONDUCTIVITY** commands are separated from the **MATERIAL** commands in order that different permeability models can more easily be incorporated into the APES+ computer program.
- **UNIT_WEIGHT** refers to the unit weight (units of FL^{-3}) of the fluid flowing through the porous material. The default **UNIT_WEIGHT** is equal to one (1.0).
- The matrix of permeability coefficients is assumed to be symmetric. Internally within APES+, the values for the permeability coefficients **K11**, **K12**, **K13**, **K22**, **K23**, **K33** that apply to a given analysis are divided by the unit weight of the pore fluid to thus give "effective" permeability coefficients. Thus, provided that the respective quotients are correct, the values used for the permeability coefficients and the fluid unit weight need not correspond to actual ones associated with the respective material.

Table 14.1 gives a classification of soil based on the value of the coefficient of permeability k. A similar, though somewhat more compact, classification is given in Table 14.2. Table 14.3 lists hydraulic permeabilities for different soils and rocks.

Table 14.1: Classification of Soils According to Their Coefficients of Permeability

Relative Permeability	Typical Soil	Value of k (cm/s)
High	Coarse gravel	$> 10^{-1}$
Medium	Sand, fine sand	10^{-1} to 10^{-3}
Low	Silty sand, dirty sand	10^{-3} to 10^{-5}
Very low	Silt, fine sandstone	10^{-5} to 10^{-7}
Practically impermeable	Clay	$< 10^{-7}$

From Terzaghi and Peck [106].

Table 14.2: Classification of Soils According to Their Coefficients of Permeability

Relative Permeability	Typical Soil	Value of k (cm/s)
Pervious	Clean gravel and sand	10^{0} to 10^{-3}
Slightly pervious	Fine sand, sandy silt, silt	10^{-3} to 10^{-7}
Practically impervious	Homogeneous clay	10^{-7} to 10^{-9}

From Kèzdi [64].

Table 14.3: Typical Values of Hydraulic Conductivity

Material Classification	Soil Type	Value of K (cm/s)
Unconsolidated sand & gravel	Well sorted gravel	10^{2} to 10^{0}
	Well sorted sand or sand & gravel	10^{0} to 10^{-3}
	Very fine sand, silt, loess, loam	10^{-3} to 10^{-7}
Unconsolidated clay & organic	Peat	10^{-2} to 10^{-4}
	Stratified clay	10^{-4} to 10^{-7}
	Fat/unweathered clay	10^{-7} to 10^{-11}
Consolidated rocks	Highly fractured rocks	10^{2} to 10^{-2}
	Oil rocks	10^{-2} to 10^{-5}
	Fresh sandstone	10^{-5} to 10^{-7}
	Good limestone, dolomite	10^{-7} to 10^{-9}
	Breccia, granite	10^{-9} to 10^{-11}

From Bear [7].

14.2 The HYDRAULIC CONDUCTIVITY CONSTANT Command

Synopsis

The **HYDRAULIC CONDUCTIVITY CONSTANT** command is used to specify values for *constant* permeability coefficients.

Syntax

The following syntax is associated with the **HYDRAULIC CONDUCTIVITY CONSTANT** command:

HYDraulic CONductivity CONstant NUMber ## DEScription "string"
K11 #.# K12 #.# K13 #.# K22 #.# K23 #.# K33 #.#
UNIt_weight #.#

Explanatory Notes

The keywords associated with the **HYDRAULIC CONDUCTIVITY CONSTANT** command have the following meaning:

NUMBER: The integer associated with this keyword is used to specify the (global) number of the constant permeability idealization. The default permeability number is one (1).

DESCRIPTION: The alphanumeric string associated with this keyword is included to describe the permeability specification. It has no effect on the values used in the specification. The alphanumeric string *must* be enclosed in double quotes (" ").

K11: The real number associated with this keyword represents the value of the coefficient of permeability k_{11} (units of Lt^{-1}). The default value of **K11** is equal to one (1.0).

K12: The real number associated with this keyword represents the value of the coefficient of permeability k_{12} (units of Lt^{-1}). The default value of **K12** is equal to zero (0.0).

K13: The real number associated with this keyword represents the value of the coefficient of permeability k_{13} (units of Lt^{-1}). The **K13** keyword is only applicable to three-dimensional analyses. The default value of **K13** is equal to zero (0.0).

K22: The real number associated with this keyword represents the value of the coefficient of permeability k_{22} (units of Lt^{-1}). The default value of **K22** is equal to one (1.0).

K23: The real number associated with this keyword represents the value of the coefficient of permeability k_{23} (units of Lt^{-1}). The **K23** keyword is only applicable to three-dimensional analyses. The default value of **K23** is equal to zero (0.0).

K33: The real number associated with this keyword represents the value of the coefficient of permeability k_{23} (units of Lt^{-1}). The **K33** keyword is only applicable to three-dimensional analyses. The default value of **K33** is equal to one (1.0).

UNIT_WEIGHT: The real number associated with this keyword represents the unit weight ($\gamma^f = \rho^f g$) of the pore fluid (units of FL^{-3}). The default value of γ^f is 1.0.

Remark: The permeability coefficients have the units of Lt^{-1}. As such, they represent the permeability coefficients commonly specified by civil engineers. ◀

Example of Command Usage

The following command is used to specify constant permeability coefficients associated with a two-dimensional spatial idealization:

```
hydraulic conductivity  constant  number 1 &
      descr  ‘‘sample permeability coefficients’’  &
          k11  3.611e-06  k22 3.611e-06  unit  0.03611
```

14.3 The HYDRAULIC CONDUCTIVITY EXPONENTIAL Command

Synopsis

The **HYDRAULIC CONDUCTIVITY EXPONENTIAL** command is used to specify values for permeability coefficients that vary *exponentially*.

Syntax

The following syntax is associated with the **HYDRAULIC CONDUCTIVITY EXPONENTIAL** command:

HYDraulic CONductivity EXPonential NUMber ## DESCription "string"
K11 #.# K12 #.# K13 #.# K22 #.# K23 #.# K33 #.#
UNIt_weight #.# N_Parameter #.#

Explanatory Notes

During an analysis, the permeability coefficients can be made a function of the current effective stress state (more precisely, of the mean normal pressure) in the following manner (recall that *tensile* stresses are assumed *positive*):

$$k = k_{in} \exp\left[n \frac{(\sigma'_{11} + \sigma'_{22} + \sigma'_{33})}{3}\right]$$

where k_{in} is an initial value of k (perhaps the in situ permeability).

The keywords associated with the **HYDRAULIC CONDUCTIVITY EXPONENTIAL** command have the following meaning:

NUMBER: The integer associated with this keyword is used to specify the (global) number of the semi-logarithmic permeability idealization. The default permeability number is one (1).

DESCRIPTION: The alphanumeric string associated with this keyword is included to describe the permeability specification. It has no effect on the values used in the specification. The alphanumeric string *must* be enclosed in double quotes (" ").

K11: The real number associated with this keyword represents the in situ value (k_{in}) of the coefficient of permeability k_{11} (units of Lt^{-1}). The default value of **K11** is equal to one (1.0).

K12: The real number associated with this keyword represents the in situ value (k_{in}) of the coefficient of permeability k_{12} (units of Lt^{-1}). The default value of **K12** is equal to zero (0.0).

K13: The real number associated with this keyword represents the in situ value (k_{in}) of the coefficient of permeability k_{13} (units of Lt^{-1}). The **K13** keyword is only applicable to three-dimensional analyses. The default value of **K13** is equal to zero (0.0).

K22: The real number associated with this keyword represents the in situ value (k_{in}) of the coefficient of permeability k_{22} (units of Lt^{-1}). The default value of **K22** is equal to one (1.0).

K23: The real number associated with this keyword represents the in situ value (k_{in}) of the coefficient of permeability k_{23} (units of Lt^{-1}). The **K23** keyword is only applicable to three-dimensional analyses. The default value of **K23** is equal to zero (0.0).

K33: The real number associated with this keyword represents the in situ value (k_{in}) of the coefficient of permeability k_{23} (units of Lt^{-1}). The **K33** keyword is only applicable to three-dimensional analyses. The default value of **K33** is equal to one (1.0).

UNIT_WEIGHT: The real number associated with this keyword represents the unit weight ($\gamma^f = \rho^f g$) of the pore fluid (units of FL^{-3}). The default value of γ^f is 1.0.

N_PARAMETER: The real number associated with this keyword represents the "permeability change index" C_k. The default value of C_k is zero (0.0). In this case the **EXPONENTIAL** idealization reduces to the **CONSTANT** one.

Remark: The permeability coefficients have the units of Lt^{-1}. As such, they represent the permeability coefficients commonly specified by civil engineers. ◀

Example of Command Usage

The following command is used to specify an exponential permeability idealization associated with a two-dimensional (spatially) analysis:

```
hydraulic  cond  exponential  number 1 &
      descr ‘‘sample permeability coefficients’’  &
          k11  1.25e-04  k22 0.938e-04  unit_weight  1.0  &
          n_param  0.85
```

14.4 The HYDRAULIC CONDUCTIVITY LOG-LOG Command

Synopsis

The **HYDRAULIC CONDUCTIVITY LOG-LOG** command is used to specify values for permeability coefficients that vary *logarithmically*. Such a an empirical relationship between the logarithm of the coefficient of permeability and logarithm of the void ratio was proposed by Mesri and Olson [87]. It is particularly applicable to *clay* soils.

Syntax

The following syntax is associated with the **HYDRAULIC CONDUCTIVITY LOG-LOG** command:

HYDraulic CONductivity LOG-log NUMber ## DEScription "string"
K11 #.# K12 #.# K13 #.# K22 #.# K23 #.# K33 #.#
UNIt_weight #.# A_Parameter #.# B_Parameter #.#

Explanatory Notes

Mesri and Olson [87] found that, if considered over a very wide range of void ratios, Taylor's formula [104] for clays; viz.,

$$\log_{10} k = \log_{10} k_{in} - \frac{(e_{in} - e)}{C_k} \tag{14.1}$$

did not hold in all cases. Consequently, they proposed the following two-parameter *linear* relationship between the logarithms of the coefficient of permeability k and void ratio e:

$$\log_{10} k = A \log_{10} e + B \tag{14.2}$$

where A and B are constants. Equation (14.2) is typically re-written in the following form:

$$k = e^A 10^B \tag{14.3}$$

The keywords associated with the **HYDRAULIC CONDUCTIVITY LOG-LOG** command have the following meaning:

NUMBER: The integer associated with this keyword is used to specify the (global) number of the semi-logarithmic permeability idealization. The default permeability number is one (1).

DESCRIPTION: The alphanumeric string associated with this keyword is included to describe the permeability specification. It has no effect on the values used in the specification. The alphanumeric string *must* be enclosed in double quotes (" ").

K11: The real number associated with this keyword represents the value of the coefficient of permeability k_{11} (units of Lt^{-1}). The default value of **K11** is equal to one (1.0).

K12: The real number associated with this keyword represents the value of the coefficient of permeability k_{12} (units of Lt^{-1}). The default value of **K12** is equal to zero (0.0).

K13: The real number associated with this keyword represents the value of the coefficient of permeability k_{13} (units of Lt^{-1}). The **K13** keyword is only applicable to three-dimensional analyses. The default value of **K13** is equal to zero (0.0).

K22: The real number associated with this keyword represents the value of the coefficient of permeability k_{22} (units of Lt^{-1}). The default value of **K22** is equal to one (1.0).

K23: The real number associated with this keyword represents the value of the coefficient of permeability k_{23} (units of Lt^{-1}). The **K23** keyword is only applicable to three-dimensional analyses. The default value of **K23** is equal to zero (0.0).

K33: The real number associated with this keyword represents the value of the coefficient of permeability k_{23} (units of Lt^{-1}). The **K33** keyword is only applicable to three-dimensional analyses. The default value of **K33** is equal to one (1.0).

UNIT_WEIGHT: The real number associated with this keyword represents the unit weight ($\gamma^f = \rho^f g$) of the pore fluid (units of FL^{-3}). The default value of γ^f is 1.0.

A_PARAMETER: The real number associated with this keyword represents the constant A in Equation (14.2). The default value for this parameter is one (1.0).

B_PARAMETER: The real number associated with this keyword represents the constant B in Equation (14.2). The default value for this parameter is one (1.0).

Remark: The permeability coefficients have the units of Lt^{-1}. As such, they represent the permeability coefficients commonly specified by civil engineers. ◀

Example of Command Usage

The following command is used to specify a log-log idealization of the permeability associated with a two-dimensional spatial idealization:

```
hydraulic conductivity  log-log  number 1 &
     descr ‘‘Mesri-Olson (1971) permeability idealization’’  &
         k11  1.25e-04  k22 0.938e-04  unit_weight  1.0  &
         a_param 2.5  b_param  0.85
```

14.5 The HYDRAULIC CONDUCTIVITY POWER Command

Synopsis

The **HYDRAULIC CONDUCTIVITY POWER** command is used to specify values for permeability coefficients that vary in an exponential (*power*) manner. Samarasinghe et al. [102] proposed such an empirical relationship between the coefficient of permeability and the void ratio. It is particularly applicable to *clay* soils.

Syntax

The following syntax is associated with the **HYDRAULIC CONDUCTIVITY POWER** command:

HYDraulic CONductivity POWer NUMber ## DEScription "string"
K11 #.# K12 #.# K13 #.# K22 #.# K23 #.# K33 #.#
UNIt_weight #.# CONstant #.# B_Parameter #.#

Explanatory Notes

Samarasinghe et al. [102] suggested that a slightly modified version of the Koženy-Carman equation would be generally applicable to normally consolidated clays. In particular, the proposed the following relationship between the coefficient of permeability (k) and the current void ratio (e):

$$k = C\,\frac{e^b}{1+e} \tag{14.4}$$

where C is a reference permeability that characterizes the material and b is an exponent (real number) on the order of 4 to 5. If $b = 3$, equation (14.4) becomes similar to Taylor's formula for sands; viz.,

$$k = D_s^2\left(\frac{\rho^f g}{\eta}\right)\frac{e^3}{1+e}C \tag{14.5}$$

where D_s is some effective particle diameter (units of L), and C is a dimensionless shape constant, sometimes referred to as the "composite shape factor" [64]. Equation (14.5) can be considered to be a simplification of the Koženy-Carman equation [71; 13] viz.,

$$k = d_e^2 \left(\frac{\rho^f g}{\eta} \right) \frac{Ce^3}{1+e} \tag{14.6}$$

Here d_e is the diameter of the spherical grain that has the same ratio of volume to surface area as holds collectively for an assemblage of particles in a given soil; this parameter is sometimes referred to as the "effective grain size" [64] (units of L).

In the present implementation, the original relationship of Samarasinghe et al. [102] (Eq. 14.4) is modified to the following form:

$$k = k_{in} C \left(\frac{e^b}{1+e} \right) \tag{14.7}$$

where k_{in} is the in situ permeability.

The keywords associated with the **HYDRAULIC CONDUCTIVITY POWER** command have the following meaning:

NUMBER: The integer associated with this keyword is used to specify the (global) number of the semi-logarithmic permeability idealization. The default permeability number is one (1).

DESCRIPTION: The alphanumeric string associated with this keyword is included to describe the permeability specification. It has no effect on the values used in the specification. The alphanumeric string *must* be enclosed in double quotes (" ").

K11: The real number associated with this keyword represents the in situ value (k_{in}) of the coefficient of permeability k_{11} (units of Lt^{-1}). The default value of **K11** is equal to one (1.0).

K12: The real number associated with this keyword represents the in situ value (k_{in}) of the coefficient of permeability k_{12} (units of Lt^{-1}). The default value of **K12** is equal to zero (0.0).

K13: The real number associated with this keyword represents the in situ value (k_{in}) of the coefficient of permeability k_{13} (units of Lt^{-1}). The **K13** keyword is only applicable to three-dimensional analyses. The default value of **K13** is equal to zero (0.0).

K22: The real number associated with this keyword represents the in situ value (k_{in}) of the coefficient of permeability k_{22} (units of Lt^{-1}). The default value of **K22** is equal to one (1.0).

K23: The real number associated with this keyword represents the in situ value (k_{in}) of the coefficient of permeability k_{23} (units of Lt^{-1}). The **K23** keyword is only applicable to three-dimensional analyses. The default value of **K23** is equal to zero (0.0).

K33: The real number associated with this keyword represents the in situ value (k_{in}) of the coefficient of permeability k_{23} (units of Lt^{-1}). The **K33** keyword is only applicable to three-dimensional analyses. The default value of **K33** is equal to one (1.0).

UNIt_weight: The real number associated with this keyword represents the unit weight ($\gamma^f = \rho^f g$) of the pore fluid (units of FL^{-3}). The default value of γ^f is 1.0.

CONSTANT: The real number associated with this keyword represents the constant C. The default value for this parameter is one (1.0).

B_PARAMETER: The real number associated with this keyword represents the exponent b. The default value for this parameter is three (3.0).

Remark: The permeability coefficients have the units of Lt^{-1}. As such, they represent the permeability coefficients commonly specified by civil engineers. ◀

Example of Command Usage

The following command is used to specify a power idealization of the permeability associated with a two-dimensional (spatially) analysis:

```
hyd cond  power  number 1 &
      descr  ‘‘permeability coefficient is a function of void ratio’’ &
          k11  1.25e-04  k22 0.938e-04  unit_weight  1.0  &
          constant 2.5  b_param  0.85
```

14.6 The HYDRAULIC CONDUCTIVITY SEMI-LOG Command

Synopsis

The **HYDRAULIC CONDUCTIVITY SEMI-LOG** command is used to specify values for permeability coefficients that vary semi-logarithmically. Such a an empirical relationship between the coefficient of permeability and the void ratio was proposed by Taylor [104]. It is particularly applicable to *clay* soils.

Syntax

The following syntax is associated with the **HYDRAULIC CONDUCTIVITY SEMI-LOG** command:

HYDraulic CONductivity SEMi-log NUMber ## DEScription "string"
K11 #.# K12 #.# K13 #.# K22 #.# K23 #.# K33 #.#
UNIt_weight #.# INItial_void #.# CONstant #.#

Explanatory Notes

Noting that the expression of Koženy-Carman was applicable only to sands, Taylor [104] proposed the following empirical relationship between the logarithm of the coefficient of permeability k and the void ratio:

$$\log_{10} k = \log_{10} k_{in} - \frac{(e_{in} - e)}{C_k} \tag{14.8}$$

where C_k is the "permeability change index"; i.e., the slope of a straight line in $e - \log_{10} k$ space, k_{in} is the in situ permeability, and e_{in} is the in situ void ratio. Equation (14.8) is typically re-written in the following form:

$$k = k_{in} 10^{[(e-e_{in})/C_k]} \tag{14.9}$$

The keywords associated with the **HYDRAULIC CONDUCTIVITY SEMI-LOG** command have the following meaning:

NUMBER: The integer associated with this keyword is used to specify the (global) number of the semi-logarithmic permeability idealization. The default permeability number is one (1).

DESCRIPTION: The alphanumeric string associated with this keyword is included to describe the permeability specification. It has no effect on the values used in the specification. The alphanumeric string *must* be enclosed in double quotes (" ").

K11: The real number associated with this keyword represents the in situ value (k_{in}) of the coefficient of permeability k_{11} (units of Lt^{-1}). The default value of **K11** is equal to one (1.0).

K12: The real number associated with this keyword represents the in situ value (k_{in}) of the coefficient of permeability k_{12} (units of Lt^{-1}). The default value of **K12** is equal to zero (0.0).

K13: The real number associated with this keyword represents the in situ value (k_{in}) of the coefficient of permeability k_{13} (units of Lt^{-1}). The **K13** keyword is only applicable to three-dimensional analyses. The default value of **K13** is equal to zero (0.0).

K22: The real number associated with this keyword represents the in situ value (k_{in}) of the coefficient of permeability k_{22} (units of Lt^{-1}). The default value of **K22** is equal to one (1.0).

K23: The real number associated with this keyword represents the in situ value (k_{in}) of the coefficient of permeability k_{23} (units of Lt^{-1}). The **K23** keyword is only applicable to three-dimensional analyses. The default value of **K23** is equal to zero (0.0).

K33: The real number associated with this keyword represents the in situ value (k_{in}) of the coefficient of permeability k_{23} (units of Lt^{-1}). The **K33** keyword is only applicable to three-dimensional analyses. The default value of **K33** is equal to one (1.0).

UNIt_weight: The real number associated with this keyword represents the unit weight ($\gamma^f = \rho^f g$) of the pore fluid (units of FL^{-3}). The default value of γ^f is 1.0.

INITIAL_VOID: The real number associated with this keyword represents the value of the in situ void ratio e_{in}. The default value of e_{in} is one (1.0).

CONSTANT: The real number associated with this keyword represents the "permeability change index" C_k. The default value of C_k is one (1.0).

Remark: The permeability coefficients have the units of Lt^{-1}. As such, they represent the permeability coefficients commonly specified by civil engineers. ◀

Example of Command Usage

The following command is used to specify a semi-logarithmic idealization of the hydraulic conductivity associated with a two-dimensional spatial idealization:

```
hydraulic conductivity  semi-log  number 2 &
      descr ‘‘semi_log representation after Taylor (1948)’’  &
          k11  1.25e-04  k22 0.938e-04  unit_weight  1.0  &
          initial_void 0.692  constant  1.02
```

Chapter 15

The INITIAL Commands

The information in this chapter is used to establish the initial state of the material (e.g., soil) for all portions of the solution domain. The initial states are described by means of simple linear equations that are functions of the x_1, x_2 and, for three-dimensional analyses, x_3-coordinates; viz.,

$$\begin{aligned}\text{initial state parameter} = \text{constant\_value} &+ (\text{1\_value})(x_1) \\ &+ (\text{2\_value})(x_2) + (\text{3\_value})(x_3)\end{aligned}$$

where the quantities "constant_value", "1_value", "2_value", and "3_value" are specified in conjunction with the keywords **CONSTANT**, **1_COEFFICIENT**, **2_COEFFICIENT**, and possibly **3_COEFFICIENT**. These values are thus dependent upon the location selected for the origin of the coordinate system.

The initial state for the following quantities can be prescribed:

- The normal total stresses σ_{11}, σ_{22} and σ_{33}.
- The shear stresses σ_{12}, σ_{13} and σ_{23}.
- The pore pressure (ϑ).
- The initial void ratio (e_{in}).
- The initial size of the bounding surface (I_0).

The last two quantities are needed only needed if the bounding surface constitutive model is used in idealizing a cohesive soil (refer to the **MATERIAL BOUNDING_SURFACE** commands).

The normal stresses σ_{11}, σ_{22} and σ_{33} represent *effective* stresses (i.e., total stress minus pore pressure), and are positive when *tensile*. For linear elasticity problems the initial stresses may be set equal to zero - the solution stresses then represent additions to the initial state (i.e., superposition is valid).

The following information pertains to the **INITIAL** commands:

- The **INITIAL BOUNDING** command (see Section 15.1).
- The **INITIAL PRECON_1** command (see Section 15.2).
- The **INITIAL PRECON_2** command (see Section 15.3).
- The **INITIAL PRESSURE** command (see Section 15.4).
- The **INITIAL STRESS_11** command (see Section 15.5).
- The **INITIAL STRESS_22** command (see Section 15.6).
- The **INITIAL STRESS_33** command (see Section 15.7).
- The **INITIAL STRESS_12** command (see Section 15.8).
- The **INITIAL STRESS_13** command (see Section 15.9).
- The **INITIAL STRESS_23** command (see Section 15.10).
- The **INITIAL VOID** command (see Section 15.11).

An example that illustrates the use of the **INITIAL** commands is also available (see Section 15.12).

15.1 The INITIAL BOUNDING Command

Synopsis

The **INITIAL BOUNDING** command is used to specify the manner in which the bounding surface in stress space shall be defined for initial state description **NUMBER**.

Syntax

The following syntax is associated with the **INITIAL BOUNDING** command:

INItial BOUnding NUMber ## OPTion ##

Explanatory Notes

This command is associated with use of the bounding surface constitutive relation for isotropic cohesive soils; if such a relation is not being utilyzed in the mathematical model under consideration, the **INITIAL BOUNDING** command (and the **INITIAL PRECON_1** and **INITIAL PRECON_2** commands to be discussed below) does not have to be specified. The keyword **NUMBER** refers to the number of the initial state specification. Associated with the **OPTION** keyword are *three* legal values, namely:

- If **OPTION** = 0 , the size of bounding surface is specified directly.
- If **OPTION** = 1, the size of bounding surface is specified in terms of the current state.
- If **OPTION** = 2, the size of bounding surface specified in terms of past state.

Additional details pertaining to these options are given below. For a soil characterized by all forms of the bounding surface model for cohesive soils [53; 54; 59], the initial size of the bounding surface must be prescribed. This size is controlled by the value of its intersect (I_0) with the hydrostatic (I) axis. There are three options available in the program for prescribing the initial value of I_0. These options are specified by the value specified in conjunction with the **INITIAL BOUNDING** command. Depending on the value of **OPTION** in this command, the values associated with the **PRECON_1** and **PRECON_2** commands take on different meanings, namely:

1. For **OPTION** = 0: The value $I_0/3$ is input directly be means of the quantity **PRECON_1**; i.e.,

$$\frac{1}{3}I_0 = CONSTANT + 1\_COEFFICIENT(x_1) + 2\_COEFFICIENT(x_2) + 3\_COEFFICIENT(x_3))$$

The division by 3 is introduced so that the APES$^+$ input remains compatible with previous versions of APES [50; 51], SAC-2 [42] and SAC-3 [88] computer programs. In this case *no entry is required* for **PRECON_2**.

2. For **OPTION** = 1: The value of I_0 is indirectly specified in terms of the initial effective stress state (I, J) and the value of the dimensionless similarity ratio b $(1 \leq b \leq \infty)$. The stress invariants I and J are related to the initial values of the horizontal (σ'_h) and vertical (σ'_v) effective stress as follows:

$$I = \sigma'_1 + 2\sigma'_3 \quad ; \quad J = \frac{1}{\sqrt{3}}(\sigma'_1 - \sigma'_3)$$

where

$$\sigma'_1 = \max(\sigma'_v, \sigma'_h) \quad ; \quad \sigma'_3 = \min(\sigma'_v, \sigma'_h)$$

The value of the dimensionless similarity ratio b is input by means of the quantity **PRECON_1**; viz.,

$$b = CONSTANT + 1\_COEFFICIENT(x_1) + 2\_COEFFICIENT(x_2) + 3\_COEFFICIENT(x_3))$$

In this case, no entry is required for **PRECON_2**.

For purposes of describing the size of the bounding surface, the projection center [24] is assumed to be located at the origin in stress invariant space. The value of b is obviously related to the overconsolidation ratio (OCR) associated with the material state of a soil.

3. For **OPTION** = 2: The value of I_0 is indirectly specified in terms of some past effective stress state (I^*, J^*), which is assumed to lie on the bounding surface. The stress invariants I^* and J^* are related to the horizontal (σ^*_h) and vertical (σ^*_v) effective stress as follows:

$$I^* = \sigma'_1 + 2\sigma'_3 \quad ; \quad J^* = \frac{1}{\sqrt{3}}(\sigma'_1 - \sigma'_3)$$

where

$$\sigma'_1 = \max(\sigma^*_v, \sigma^*_h) \quad ; \quad \sigma'_3 = \min(\sigma^*_v, \sigma^*_h)$$

The values of σ^*_v and σ^*_h are specified by means of the **PRECON_1** and **PRECON_2** commands, respectively.

Regardless of which of the above input procedures is selected, if b is less than 1.0, a warning message shall be printed.

Example of Command Usage

To indicate that the size of the bounding surface associated with initial state number 3 shall be defined in terms of the current state, specify either of the following commands:

```
INITIAL   BOUNDING  NUMBER  3  OPTION  2

ini bou num 3 opt 2
```

15.2 The INITIAL PRECON_1 Command

Synopsis

The **INITIAL PRECON_1** command is used to specify the initial preconsolidation pressure in a soil. As such, it is instrumental in determining the initial size of the bounding surface associated with bounding surface models for cohesive soils. Specification of the **INITIAL PRECON_1** command is thus only required when such constitutive models are being used to characterize the soil(s) in a mathematical model.

As in the case of the majority of the other **INITIAL** commands, this initial preconsolidation pressure is described by means of simple linear equations that are functions of the x_1, x_2, and possibly x_3 coordinates.

Syntax

The following syntax is associated with the **INITIAL PRECON_1** command:

INItial PRECON_1 NUMber ## CONSTANT #.# 1_Coefficient #.# 2_Coefficient #.# 3_Coefficient #.#

Explanatory Notes

The keyword **NUMBER** refers to the number of the initial state specification.

This command is used to describe the initial preconsolidation pressure by means of the following simple linear equation:

$$\begin{aligned}\text{preconsolidation parameter 1} = CONSTANT &+ 1\_COEFFICIENT(x_1) \\ &+ 2\_COEFFICIENT(x_2) \\ &+ 3\_COEFFICIENT(x_3)\end{aligned}$$

Compressive preconsolidation pressures are assumed to be *positive*.

15.3 The INITIAL PRECON_2 Command

Synopsis

The **INITIAL PRECON_2** command is used to specify the initial preconsolidation pressure in a soil. As such, it is instrumental in determining the initial size of the bounding surface associated with bounding surface models for cohesive soils. Specification of the **INITIAL PRECON_2** command is thus only required when such constitutive models are being used to characterize the soil(s) in a mathematical model.

As in the case of the majority of the other **INITIAL** commands, this initial preconsolidation pressure is described by means of simple linear equations that are functions of the x_1, x_2, and possibly x_3 coordinates.

Syntax

The following syntax is associated with the **INITIAL PRECON_2** command:

INItial PRECON_2 NUMber ## CONSTANT #.# 1_Coefficient #.#
2_Coefficient #.# 3_Coefficient #.#

Explanatory Notes

The keyword **NUMBER** refers to the number of the initial state specification.

This command is used to describe the initial preconsolidation pressure by means of the following simple linear equation:

$$\begin{aligned}\text{preconsolidation parameter } 2 = CONSTANT &+ 1\_COEFFICIENT(x_1) \\ &+ 2\_COEFFICIENT(x_2) \\ &+ 3\_COEFFICIENT(x_3)\end{aligned}$$

Compressive preconsolidation pressures are assumed to be *positive*.

15.4 The INITIAL PRESSURE Command

Synopsis

The **INITIAL PRESSURE** command is used to specify the initial distribution of pore pressure (ϑ) in a soil mass associated with initial state description **NUMBER**. *Compressive* pore pressures are assumed to be *positive*.

As in the case of the majority of the other **INITIAL** commands, this initial pore pressure is described by means of simple linear equations that are functions of the x_1, x_2, and possibly x_3 coordinates.

Syntax

The following syntax is associated with the **INITIAL PRESSURE** command:

INItial PREssure NUMber ## CONSTANT #.# 1_Coefficient #.#
2_Coefficient #.# 3_Coefficient #.#

Explanatory Notes

The units of pore pressure ϑ are those of stress. The values of ϑ, specified by means of the **INITIAL PRESSURE** command, are used directly to initialize the pore pressure in the elements, and indirectly to initialize the pore pressures at the nodes. The initial pore pressure distribution must be consistent with the manner in which the flow problem is being analyzed (also refer to the discussion of the **ANALYSIS TYPE** command given in Section 5.9). That is, if the solution is stated in terms of *total* pore pressure, the initial *total* pore pressure distribution must be specified in this section. If, on the other hand, the solution is specified in terms of *excess* pore pressure, the initial *excess* pore pressure distribution must be specified.

15.5 The INITIAL STRESS_11 Command

Synopsis

The **INITIAL STRESS_11** command is used to specify the initial distribution of normal stress parallel to the global x_1-coordinate direction (i.e., σ_{11}) in a soil mass associated with initial state description **NUMBER**. *Tensile* stresses are assumed to be *positive*.

As in the case of the majority of the other **INITIAL** commands, this initial pore pressure is described by means of simple linear equations that are functions of the x_1, x_2, and possibly x_3 coordinates.

Syntax

The following syntax is associated with the **INITIAL STRESS_11** command:

INItial STRESS_11 NUMber ## CONSTANT #.# 1_Coefficient #.#
2_Coefficient #.# 3_Coefficient #.#

Explanatory Notes

For analyses involving geomaterials, σ_{11} is an *effective* stress (i.e., total stress minus pore pressure). For linear elasticity problems the initial stresses may be set equal to zero - the stresses determined as part of the solution then represent additions to the initial state (i.e., the principle of superposition is valid).

15.6 The INITIAL STRESS_22 Command

Synopsis

The **INITIAL STRESS_22** command is used to specify the initial distribution of normal stress parallel to the global x_2-coordinate direction (i.e., σ_{22}) in a soil mass associated with initial state description **NUMBER**. For analyses involving geomaterials, this stress is an *effective* one. *Tensile* stresses are assumed to be *positive*.

As in the case of the majority of the other **INITIAL** commands, this initial pore pressure is described by means of simple linear equations that are functions of the x_1, x_2, and possibly x_3 coordinates.

Syntax

The following syntax is associated with the **INITIAL STRESS_22** command:

INItial STRESS_22 NUMber ## CONSTANT #.# 1_Coefficient #.#
2_Coefficient #.# 3_Coefficient #.#

Explanatory Notes

For analyses involving geomaterials, σ_{22} is an *effective* stress (i.e., total stress minus pore pressure). For linear elasticity problems the initial stresses may be set equal to zero - the stresses determined as part of the solution then represent additions to the initial state (i.e., the principle of superposition is valid).

15.7 The INITIAL STRESS_33 Command

Synopsis

The **INITIAL STRESS_33** command is used to specify the initial distribution of normal stress parallel to the global x_3-coordinate direction (i.e., σ_{33}) in a soil mass associated with initial state description **NUMBER**. For analyses involving geomaterials, this stress is an *effective* one. *Tensile* stresses are assumed to be *positive*.

As in the case of the majority of the other **INITIAL** commands, this initial pore pressure is described by means of simple linear equations that are functions of the x_1, x_2, and possibly x_3 coordinates.

Syntax

The following syntax is associated with the **INITIAL STRESS_33** command:

INItial STRESS_33 NUMber ## CONSTANT #.# 1_Coefficient #.#
2_Coefficient #.# 3_Coefficient #.#

Explanatory Notes

For analyses involving geomaterials, σ_{33} is an *effective* stress (i.e., total stress minus pore pressure). For linear elasticity problems the initial stresses may be set equal to zero - the stresses determined as part of the solution then represent additions to the initial state (i.e., the principle of superposition is valid).

15.8 The INITIAL STRESS_12 Command

Synopsis

The **INITIAL STRESS_12** command is used to specify the initial distribution of shear stress in the x_1 - x_2 plane in a soil mass associated with initial state description **NUMBER**.

As in the case of the majority of the other **INITIAL** commands, this initial pore pressure is described by means of simple linear equations that are functions of the x_1, x_2, and possibly x_3 coordinates.

Syntax

The following syntax is associated with the **INITIAL STRESS_12** command:

INItial STRESS_12 NUMber ## CONSTANT #.# 1_Coefficient #.# 2_Coefficient #.# 3_Coefficient #.#

Explanatory Notes

For linear elasticity problems the initial stresses may be set equal to zero - the stresses determined as part of the solution then represent additions to the initial state (i.e., the principle of superposition is valid).

15.9 The INITIAL STRESS_13 Command

Synopsis

The **INITIAL STRESS_13** command is used to specify the initial distribution of shear stress in the x_1 - x_3 plane in a soil mass associated with initial state description **NUMBER**.

As in the case of the majority of the other **INITIAL** commands, this initial pore pressure is described by means of simple linear equations that are functions of the x_1, x_2, and possibly x_3 coordinates.

Syntax

The following syntax is associated with the **INITIAL STRESS_13** command:

INItial STRESS_13 NUMber ## CONSTANT #.# 1_Coefficient #.# 2_Coefficient #.# 3_Coefficient #.#

Explanatory Notes

For linear elasticity problems the initial stresses may be set equal to zero - the stresses determined as part of the solution then represent additions to the initial state (i.e., the principle of superposition is valid).

15.10 The INITIAL STRESS_23 Command

Synopsis

The **INITIAL STRESS_23** command is used to specify the initial distribution of shear stress in the x_2 - x_3 plane in a soil mass associated with initial state description **NUMBER**.

As in the case of the majority of the other **INITIAL** commands, this initial pore pressure is described by means of simple linear equations that are functions of the x_1, x_2, and possibly x_3 coordinates.

Syntax

The following syntax is associated with the **INITIAL STRESS_23** command:

INItial STRESS_23 NUMber ## CONSTANT #.# 1_Coefficient #.# 2_Coefficient #.# 3_Coefficient #.#

Explanatory Notes

For linear elasticity problems the initial stresses may be set equal to zero - the stresses determined as part of the solution then represent additions to the initial state (i.e., the principle of superposition is valid).

15.11 The INITIAL VOID Command

Synopsis

The **INITIAL VOID** command is used to specify the initial distribution of void ratio in a soil mass associated with initial state description **NUMBER**.

As in the case of the majority of the other **INITIAL** commands, this initial pore pressure is described by means of simple linear equations that are functions of the x_1, x_2, and possibly x_3 coordinates.

Syntax

The following syntax is associated with the **INITIAL VOID** command:

INItial VOID NUMber ## CONSTANT #.# 1_Coefficient #.#
2_Coefficient #.# 3_Coefficient #.#

Explanatory Notes

The initial void ratio distribution is needed only needed if the bounding surface constitutive model is used in idealizing a cohesive soil (refer to the **MATERIAL BOUNDING_SURFACE** commands).

15.12 Example Illustrating use of the INITIAL Commands

Figure 15.1 shows a two-dimensional field example. The location of the coordinate origin is shown in the aforementioned figure. Units of pounds and inches are used in the analysis. The initial state is assumed to vary only in the global y (x_2) coordinate direction.

A value of 1.0 for the gravitational acceleration is prescribed using the **GRAVITY ACCELERATION VALUE** command (see Section 9.2). The quantities input for the solid and pore fluid densities thus have units of force per unit volume. These values are input using the **DENSITY SOLID CONSTANT** command (see Section 17.2) and the **DENSITY FLUID CONSTANT** command (see Section 17.1), respectively. The analysis will thus be performed in terms of *total* (as opposed to excess) pore pressure.

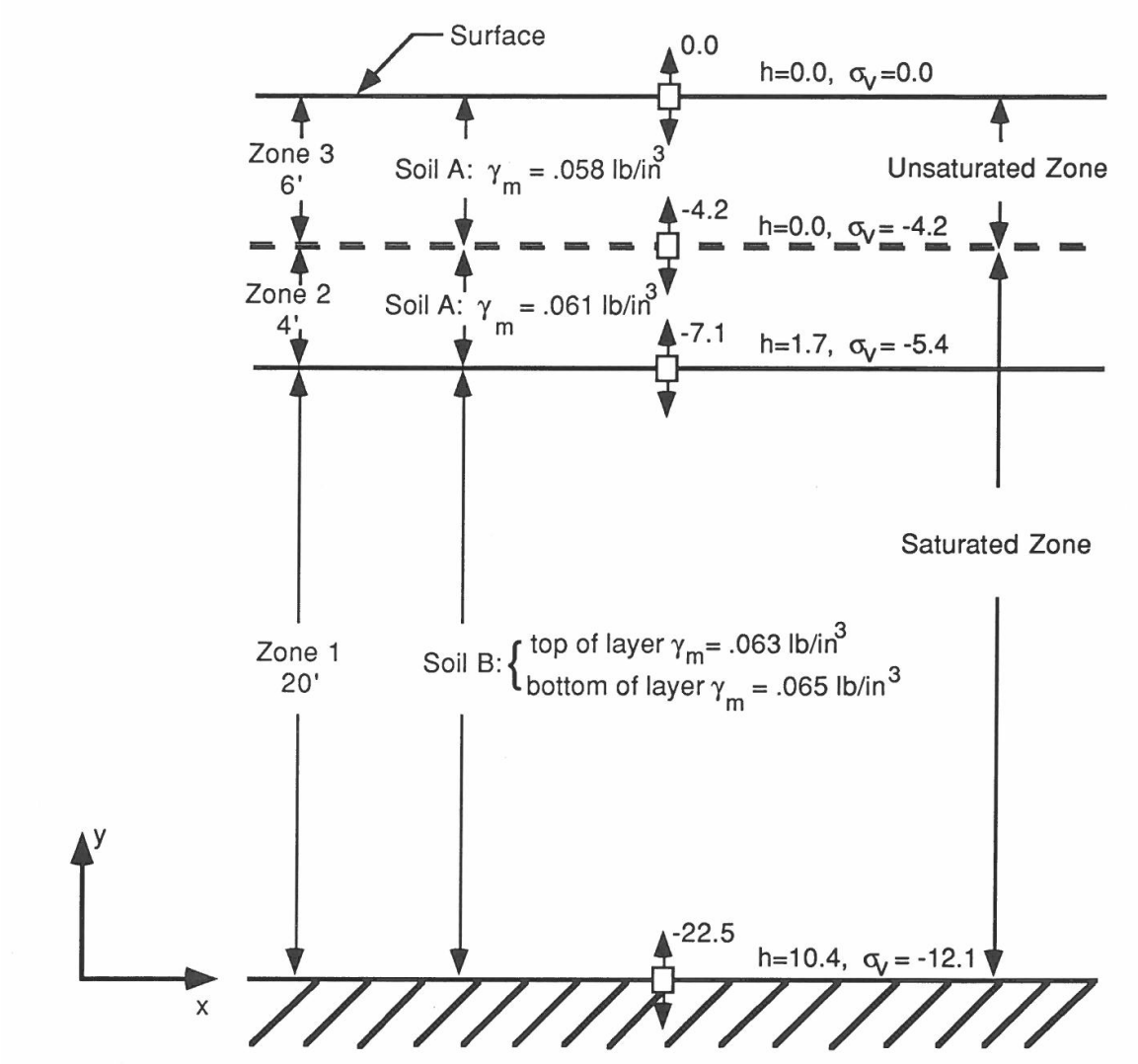

Figure 15.1: Hypothetical soil profile.

The vertical and horizontal soil stresses are computed from the unit weights of the respective soils and an assumed coefficient of lateral earth pressure at rest (K_0) equal to 0.45. The initial shear stress throughout the soil deposit is assumed to be zero.

For Zone 1, the initial state description is given the number 1. For this description, the following commands are used to define the initial material state:

```
initial  stress_11  number 1  constant -5.430  2_coefficient  0.0126
initial  stress_22  number 1  constant -12.060  2_coefficient  0.028
initial pressure  number 1  constant  0.0  2_coefficient  0.0
initial void  number 1  constant  10.40  2_coefficient  -0.0361
```

For Zone 2, the initial state description is given the number 2. For this description, the following commands are used to define the initial material state:

```
initial  stress_11  number 2  constant -5.106  2_coefficient  0.0112
initial  stress_22  number 2  constant -11.346  2_coefficient  0.0249
initial pressure  number 2  constant  0.0  2_coefficient  0.0
initial void  number 2  constant  10.40  2_coefficient  -0.0361
```

For Zone 3, the initial state description is given the number 3. For this description, the following commands are used to define the initial material state:

```
initial  stress_11  number 3  constant -9.396  2_coefficient  0.0261
initial  stress_22  number 3  constant -20.880  2_coefficient  0.058
initial pressure  number 3  constant  0.0  2_coefficient  0.0
initial void  number 3  constant  0.0  2_coefficient  0.0
```

Chapter 16

The BULK Commands

The **BULK** commands are used to specify the bulk modulus (units of FL^{-2}) of the solid and, if applicable, the pore fluid phase. The solid bulk modulus applies to the following types of analyses:

- **ANALYSIS TYPE FLOW_POROMECHANICAL**
- **ANALYSIS TYPE FLOW_PORO_THERMAL**
- **ANALYSIS TYPE POROMECHANICAL**
- **ANALYSIS TYPE PORO_THERMAL**
- **ANALYSIS TYPE SOLID**
- **ANALYSIS TYPE SOLID_THERMAL**

The pore fluid bulk modulus is only required for the following types of analyses:

- **ANALYSIS TYPE FLOW_POROMECHANICAL**
- **ANALYSIS TYPE FLOW_PORO_THERMAL**

The following **BULK** commands are available for use by the analyst:

- The **BULK FLUID CONSTANT** command (see Section 16.1).
- The **BULK SOLID CONSTANT** command (see Section 16.2).

This chapter gives additional details pertaining to the above commands.

16.1 The BULK_MODULUS FLUID CONSTANT command

Synopsis

The **BULK_MODULUS FLUID CONSTANT** command is used to specify a constant bulk modulus of the pore fluid phase.

Before using the **BULK_MODULUS FLUID CONSTANT** command, the **DIMENSION MAXIMUM BULK_FLUID CONSTANT** command must be given, along with the number of different constant bulk modulus idealizations that are to be used in the analysis. Additional details pertaining to the latter command are given in Section 6.2.

Syntax

The following syntax is associated with the **BULK_MODULUS FLUID CONSTANT** command:

BULk_modulus FLUid CONStant NUMber # (DEScription "string") VALue #.#

Explanatory Notes

- The **NUMBER** keyword is used to specify the number of the constant bulk modulus description. The value associated with this description is assigned to an element through the **FLUID_BULK** keyword that is part of an element command.
- The **STRING** keyword is used to enter an optional text string that is used to identify the particular **BULK_MODULUS FLUID CONSTANT** specification. This feature is provided solely for the benefit of the analyst. The text string is stored and, provided the **ECHO BULK_MODULUS OFF** command has not been given, is printed with the **FLUID_BULK** information.
- The default **VALUE** of the solid bulk modulus is zero (0.0).

Example of Command Usage

To specify a constant pore fluid bulk modulus equal to 2.10 GPa for description number 3, enter either of the following commands:

```
bulk_modulus fluid constant number 3 description ‘‘bulk modulus of water’’  &
   value 2.10

bulk_mod flu con num 3 des ‘‘bulk modulus of water’’  val  2.10
```

16.2 The BULK_MODULUS SOLID CONSTANT command

Synopsis

The **BULK_MODULUS SOLID CONSTANT** command is used to specify a constant bulk modulus of the solid phase.

Before using the **BULK_MODULUS SOLID CONSTANT** command, the **DIMENSION MAXIMUM BULK_SOLID CONSTANT** command must be given, along with the number of different constant bulk modulus idealizations that are to be used in the analysis. Additional details pertaining to the latter command are given in Section 6.2.

Syntax

The following syntax is associated with the **BULK_MODULUS SOLID CONSTANT** command:

BULk_modulus SOLid CONstant NUMber # (DEScription "string") VALue #.#

Explanatory Notes

- The **NUMBER** keyword is used to specify the number of the constant bulk modulus description. The value associated with this description is assigned to an element through the **SOLID_BULK** keyword that is part of an element command.
- The **STRING** keyword is used to enter an optional text string that is used to identify the particular **BULK_MODULUS SOLID CONSTANT** specification. This feature is provided solely for the benefit of the analyst. The text string is stored and, provided the **ECHO BULK_MODULUS OFF** command has not been given, is printed with the **SOLID_BULK** information.
- The default **VALUE** of the solid bulk modulus is zero (0.0).

Example of Command Usage

To specify a constant solid bulk modulus equal to 1.80×10^8 for description number 2, enter either of the following commands:

```
bulk_modulus solid constant number 2  &
          description ‘‘bulk modulus for coarse sand’’  value 1.80e+08

bulk_mod sol con num 2 des ‘‘bulk modulus for coarse sand’’ val 1.80e+08
```

Chapter 17

The DENSITY Commands

The **DENSITY** commands are used to specify the mass density (units of ML^{-3} or $FL^{-4}t^2$) of the solid and, if applicable, the pore fluid phase. The solid density applies to the following types of analyses:

- **ANALYSIS TYPE FLOW_POROMECHANICAL**
- **ANALYSIS TYPE FLOW_PORO_THERMAL**
- **ANALYSIS TYPE POROMECHANICAL**
- **ANALYSIS TYPE PORO_THERMAL**
- **ANALYSIS TYPE SOLID**
- **ANALYSIS TYPE SOLID_THERMAL**

The pore fluid density is only required for the following types of analyses:

- **ANALYSIS TYPE FLOW_POROMECHANICAL**
- **ANALYSIS TYPE FLOW_PORO_THERMAL**

As noted in the description of the **GRAVITY ACCELERATION VALUE** command (see Section 9.2), the units of the solid and pore fluid material **DENSITY** must be compatible with the units selected for the gravitational acceleration g. In all instances, the products g and these densities must have units of weight per unit volume (i.e., FL^{-3}). The solid **DENSITY** denotes the dry mass density (or unit weight) of the solid phase of the soil, i.e., the mass (or weight) of the solids divided by the total volume.

Problems in which flow of pore fluid is taken into account can be stated in terms of either *total* or *excess* pore pressure. In the first case, the fluid **DENSITY** must be set equal to the mass density (or the unit weight - see previous paragraph) of the fluid; in the second case fluid **DENSITY** should be set equal to zero (0.0).

The unit weights of the solid and fluid phases are then equal to the product of the density and the gravitational acceleration (see Section 9.2). Denoting the density of the solid and pore fluid phases by ρ_s and ρ_w, respectively, the corresponding unit weights (units of FL^{-3}) are then

$$\gamma_s = \rho_s\, g \quad \text{and} \quad \gamma_w = \rho_w\, g$$

where the value of g and the associated history are specified using the The **GRAVITY ACCELERATION VALUE** (see Section 9.2) and **GRAVITY ACCELERATION HISTORY** (see Section 9.1) commands, respectively.

In **FLOW_POROMECHANICAL** and **FLOW_PORO_THERMAL** analyses, the saturated unit weight is computed as follows:

$$\rho_{sat} = \frac{\rho_w(G_s + e)}{1 + e}$$

where G_s is the specific gravity of solids, and e is the void ratio. Since $G_s = \rho_s/\rho_w$, the above expressions is re-written as

$$\rho_{sat} = \frac{\rho_w(\rho_s/\rho_w + e)}{1 + e} = \frac{\rho_s + \rho_w e}{1 + e}$$

The saturated unit weight is then $\gamma_{sat} = \rho_{sat}\, g$.

The following **DENSITY** commands are available for use by the analyst:

- The **DENSITY FLUID CONSTANT** command (see Section 17.1).
- The **DENSITY SOLID CONSTANT** command (see Section 17.2).

This chapter gives additional details pertaining to the above commands.

17.1 The DENSITY FLUID CONSTANT command

Synopsis

The **DENSITY FLUID CONSTANT** command is used to specify a constant mass density of the pore fluid phase.

Before using the **DENSITY SOLID CONSTANT** command, the **DIMENSION MAXIMUM DENSITY_SOLID** command must be given, along with the number of different constant density idealizations that are to be used in the analysis. Additional details pertaining to the latter command are given in Section 6.3.

Syntax

The following syntax is associated with the **DENSITY FLUID CONSTANT** command:

DENsity FLUid CONstant NUMber # (DEScription "string") VALue #.#

Explanatory Notes

- The **NUMBER** keyword is used to specify the number of the constant density description. The value associated with this description is assigned to an element through the **FLUID_DENSITY** keyword that is part of an element command.
- The **STRING** keyword is used to enter an optional text string that is used to identify the particular **DENSITY SOLID CONSTANT** specification. This feature is provided solely for the benefit of the analyst. The text string is stored and, provided the **ECHO DENSITY OFF** command has not been given, is printed with the **DENSITY FLUID** information.
- The default **VALUE** of the solid density is zero (0.0).

Example of Command Usage

To specify a constant pore fluid density equal to 62.4 for description number 2, enter either of the following commands:

```
density fluid constant number 2 description ‘‘density of water’’  &
   value 62.4

den flu con num 2 des ‘‘density of water’’  val 62.4
```

17.2 The DENSITY SOLID CONSTANT command

Synopsis

The **DENSITY SOLID CONSTANT** command is used to specify a constant mass density of the solid phase.

Before using the **DENSITY SOLID CONSTANT** command, the **DIMENSION MAXIMUM DENSITY_SOLID** command must be given, along with the number of different constant density idealizations that are to be used in the analysis. Additional details pertaining to the latter command are given in Section 6.3.

Syntax

The following syntax is associated with the **DENSITY SOLID CONSTANT** command:

DENsity SOLid CONstant NUMber # (DEScription "string") VALue #.#

Explanatory Notes

- The **NUMBER** keyword is used to specify the number of the constant density description. The value associated with this description is assigned to an element through the **SOLID_DENSITY** keyword that is part of an element command.
- The **STRING** keyword is used to enter an optional text string that is used to identify the particular **DENSITY SOLID CONSTANT** specification. This feature is provided solely for the benefit of the analyst. The text string is stored and, provided the **ECHO DENSITY OFF** command has not been given, is printed with the **DENSITY SOLID** information.
- The default **VALUE** of the solid density is zero (0.0).

Example of Command Usage

To specify a constant solid density equal to 1.92 for description number 3, enter either of the following commands:

```
density solid constant number 3 description ‘‘density for coarse sand’’  &
   value 1.92

den sol con num 3 des ‘‘density for coarse sand’’  val 1.92
```

Chapter 18

The ELEMENT Commands

This chapter contains information associated with the description of element data for finite element analyses using the APES$^+$ computer program.

18.1 General ELEMENT Information

Synopsis

The **ELEMENT** commands are used to describe all the elements found in the mathematical model being created. Elements are described in *groups*; each group contains elements of a specific *kind* and *type*. The order of the element records need bear no relation to the actual locations of the elements within the body being modeled. The order will, however, determine the assigned element numbers that appear in the printed output file.

In using APES$^+$ it is possible to model the process of "incremental construction" of earth structures [19]. The values associated with the **CONSTRUCTION** keyword specify the solution increment in which the material in the particular element is added to the model. Material in existence at the beginning of the analysis period must be given a value equal to zero (0). In the current version of APES$^+$ the hydraulic boundary condition, if applicable (e.g., refer to the description of **MIXED_FLOW elements**), at the temporary top of a soil mass that is being constructed is zero flow; i.e., an impervious surface. The conditions on top of the final surface can, however, be specified through the use of the **SPECIFICATIONS CONCENTRATED FLOW** command.

It is likewise possible to model the process of "incremental excavation"; i.e., the removal of material from a mathematical model. The values associated with the **EXCAVATION** keyword specify the solution increment in which the material in the particular element is removed from the model.

18.2 KINDS of Elements Available in APES+

Ten *kinds* of elements are available in the current version of APES+, namely:

- **BAR_MECHANICAL** elements (see Section 18.3).
- **BAR_SCALAR** elements.
- **FLEXURAL** elements (see Section 18.4).
- **INFINITE** elements (see Section 18.5).
- **INTERFACE** elements (see Section 18.6).
- **IRREDUCIBLE** elements for the analysis of solids and/or poromechanical materials.
  - Currently available **TRIANGULAR** irreducible elements (see Section 18.7.1).
  - Currently available **QUADRILATERAL** irreducible elements (see Section 18.7.2).
  - Currently available **TETRAHEDRAL** irreducible elements (see Section 18.7.3).
  - Currently available **HEXAHEDRAL** irreducible elements (see Section 18.7.4).
- **MIXED** elements for solids and/or geomaterials (see Section 18.8).
- **MIXED_FLOW** elements for saturated (two-phase) geomaterials (see Section 18.9).
- **NON-CONFORMING** irreducible elements.
- **SCALAR** irreducible elements (see Section 18.10).

Each kind contains at least one *type* of element. Lists of the element types available for each kind of element are available via the above links.

18.3 Available BAR_MECHANICAL Elements

The types of line (bar) elements that are suitable for mechanical analyses are listed below. These elements are entered using the **ELEMENT BAR_MECHANICAL** commands.

- General information regarding these element types is given in Chapter 19.

18.3.1 Type L2FDP0 Bar_Mechanical Elements

The **L2FDP0** is a 2-node, irreducible (linear displacement) finite deformation "membrane" element for one-, or two-dimensional mechanical analyses.

- The command syntax for this element type is available in Section 19.1.

18.3.2 Type L2P0 Bar_Mechanical Elements

The **L2P0** is a 2-node, irreducible (linear displacement) line (bar) element for one-, two- or three-dimensional mechanical analyses.

- The command syntax is available in Section 19.2.

18.3.3 Type L3P0 Bar_Mechanical Elements

The **L3P0** is a 3-node, irreducible (quadratic displacement) line (bar) element for one-, two- or three-dimensional mechanical analyses.

- The command syntax is available in Section 19.3.

18.4 Available FLEXURAL Elements

The types of frame (beam) elements are listed below. These elements are entered using the **ELEMENT FLEXURAL** commands.

- General information regarding these element types is given in Chapter 20.

18.4.1 Type BE2_2D Flexural Elements

The **BE2_2D** is a 2-node cubic Bernoulli-Euler (C^1 continuous) plane frame element.

- The command syntax is available in Section 20.1.

18.4.2 Type TB2_2D Flexural Elements

The **TB2_2D** is a 2-node linear "Timoshenko" (C^0 continuous) plane frame element.

- The command syntax is available in Section 20.2.

18.4.3 Type TB3_2D Flexural Elements

The **TB3_2D** is a 3-node quadratic "Timoshenko" (C^0 continuous) plane frame element.

- The command syntax is available in Section 20.3.

18.5 Available INFINITE Elements

The available *infinite* continuum elements for mechanical analyses are listed below. These elements are entered using the ELEMENT INFINITE commands.

- General information regarding these element types is given in Chapter 21.

18.5.1 Type INFQ6 Infinite Elements

The **INFQ6** is a 6-node irreducible "infinite" (linear/quadratic displacement) quadrilateral element.

- The command syntax is available in Section 21.2.

18.5.2 Type INFQ8 Infinite Elements

The **INFQ8** is an 8-node irreducible "infinite" (biquadratic displacement) quadrilateral element.

- The command syntax is available in Section 21.3.

18.5.3 Type INFQ9 Infinite Elements

The **INFQ9** is a 9-node irreducible "infinite" (biquadratic displacement) quadrilateral element.

- The command syntax is available in Section 21.4.

18.6 Available INTERFACE Elements

The available zero-thickness interface elements for mechanical analyses are listed below. These elements are entered using the ELEMENT INTERFACE commands.

- General information regarding these element types is given in Chapter 22.

18.6.1 Type LK1 Interface Elements

The **LK1** is a 4-node (linear) kinematically consistent *zero-thickness* interface element developed by Kaliakin and Li [57].

- The command syntax is available in Section 22.1.

18.6.2 Type LRH Interface Elements

The **LRH** is a 4-node (linear) *zero-thickness* ("link") interface element originally developed by Herrmann [39].

- The command syntax is available in Section 22.2.

18.7 Available IRREDUCIBLE Elements

The available *irreducible* continuum elements for mechanical analyses are listed below. These elements are entered using the ELEMENT IRREDUCIBLE commands.

18.7.1 Type TRIANGULAR Irreducible Elements

The irreducible *triangular* elements for mechanical analyses currently available in APES+ are listed below.

- General information regarding these element types is given in Chapter 23.

18.7.1.1 Type T3P0 Irreducible Elements

The **T3P0** is an irreducible, 3-node *linear* displacement isoparametric triangular element for mechanical analyses.

- The command syntax is available in Section 23.1.

18.7.1.2 Type T6P0 Irreducible Elements

The **T6P0** is an irreducible, 6-node *quadratic* displacement, parametrically mapped triangular element for mechanical analyses.

- The command syntax is available in Section 23.2.

18.7.1.3 Type T7P0 Irreducible Elements

The **T7P0** is an irreducible, 7-node "enhanced" *quadratic* displacement, parametrically mapped triangular element for mechanical analyses. This element adds a cubic "bubble" mode to the quadratic T6P0 element. **NOTE**: this element is meant for research purposes and it is *not* recommended for general use.

- The command syntax is available in Section 23.3.

18.7.1.4 Type T10P0 Irreducible Elements

The **T10P0** is an irreducible, 10-node *cubic* displacement, parametrically mapped triangular element for mechanical analyses.

- The command syntax is available in Section 23.4.

18.7.1.5 Type T15P0 Irreducible Elements

The **T15P0** is a 15-node irreducible, *quartic* displacement, parametrically mapped triangular element for mechanical analyses.

- The command syntax is available in Section 23.5.

18.7.2 Type QUADRILATERAL Irreducible Elements

The *quadrilateral* irreducible elements for mechanical analyses currently available in APES+ are listed below.

- General information regarding these element types is given in Chapter 24.

18.7.2.1 Type Q4P0 Irreducible Elements

The **Q4P0** is a 4-node irreducible, *bilinear* displacement, isoparametric Lagrangian quadrilateral element for mechanical analyses.

- The command syntax is available in Section 24.1.

18.7.2.2 Type Q8P0 Irreducible Elements

The **Q8P0** is an 8-node irreducible, *quadratic* displacement, parametrically mapped "serendipity" quadrilateral element for mechanical analyses.

- The command syntax is available in Section 24.2.

18.7.2.3 Type Q9P0 Irreducible Elements

The **Q9P0** is a 9-node irreducible, *quadratic* displacement, parametrically mapped Lagrangian quadrilateral element for mechanical analyses.

- The command syntax is available in Section 24.3.

18.7.2.4 Type Q12P0 Irreducible Elements

The **Q12P0** is a 12-node irreducible, *cubic* displacement, parametrically mapped "serendipity" quadrilateral element for mechanical analyses.

- The command syntax is available in Section 24.4.

18.7.2.5 Type Q16P0 Irreducible Elements

The **Q16P0** is a 16-node irreducible, *cubic* displacement, parametrically mapped Lagrangian quadrilateral element for mechanical analyses.

- The command syntax is available in Section 24.5.

18.7.3 Type TETRAHEDRAL Irreducible Elements

The *tetrahedral* irreducible elements for mechanical analyses currently available in APES+ are listed below.

- General information regarding these element types is given in Chapter 25.

18.7.3.1 Type TET4P0 Irreducible Elements

The **TET4P0** is a 4-node irreducible, *linear* displacement isoparametric tetrahedral element for mechanical analyses.

- The command syntax is available in Section 25.1.

18.7.3.2 Type TET10P0 Irreducible Elements

The **TET10P0** is a 10-node irreducible, *quadratic* displacement, parametrically mapped tetrahedral element for mechanical analyses.

- The command syntax is available in Section 25.2.

18.7.4 Type HEXAHEDRAL Irreducible Elements

The *hexahedral* irreducible elements for mechanical analyses currently available in APES+ are listed below.

- General information regarding these element types is given in Chapter 26.

18.7.4.1 Type H8P0 Irreducible Elements

The **H8P0** is an 8-node irreducible, *trilinear* displacement, isoparametric Lagrangian hexahedral element for mechanical analyses.

- The command syntax is available in Section 26.1.

18.7.4.2 Type H20P0 Irreducible Elements

The **H20P0** is a 20-node irreducible, *quadratic* displacement, parametrically mapped "serendipity" hexahedral element for mechanical analyses.

- The command syntax is available in Section 26.2.

18.7.4.3 Type H27P0 Irreducible Elements

The **H27P0** is a 27-node irreducible, *triquadratic* displacement, parametrically mapped Lagrangian hexahedral element for mechanical analyses.

- The command syntax is available in Section 26.3.

18.8 Available MIXED Elements

The available *mixed* continuum elements that account for the presence of pressure mechanical analyses are listed below. In these elements, the pressure approximation is *discontinuous*. Such elements are entered using the ELEMENT MIXED commands.

- General information regarding these element types is given in Chapter 27.

18.8.1 Type T6P1d Mixed Elements

The **T6P1d** is a 6-node mixed or mixed-penalty, parametrically mapped triangular element. The approximation for displacement is quadratic; a constant (discontinuous) pressure is assumed.

- The command syntax is available in Section 27.1.

18.8.2 Type T7P3d Mixed Elements

The **T7P3d** is a 7-node mixed or mixed-penalty, parametrically mapped triangular element. The approximation for displacement is quadratic with an additional "bubble" node. A constant (discontinuous) pressure approximation is assumed.

- The command syntax is available in Section 27.2.

18.8.3 Type Q4P1d Mixed Elements

The **Q4P1d** is a 4-node mixed or mixed-penalty, isoparametric quadrilateral element. The approximation for displacement is bilinear in displacement; a constant (discontinuous) pressure is assumed.

- The command syntax is available in Section 27.3.

18.8.4 Type Q8P3d Mixed Elements

The **Q8P3d** is an 8-node mixed or mixed-penalty, parametrically mapped quadrilateral element. The approximation for displacement is quadratic, of "serendipity" type; a linear (discontinuous) pressure is assumed.

- The command syntax is available in Section 27.4.

18.8.5 Type Q9P3d Mixed Elements

The **Q9P3d** is a 9-node mixed or mixed-penalty, parametrically mapped quadrilateral element. The approximation for displacement is quadratic, of Lagrangian type; a linear (dis-

continuous) pressure is assumed.

- The command syntax is available in Section 27.5.

18.8.6 Type H8P1d Mixed Elements

The **H8P1d** is an 8-node mixed or mixed-penalty, isoparametric hexahedral element. The approximation for displacement is trilinear in displacement; a constant (discontinuous) pressure is assumed.

- The command syntax is available in Section 27.6.

18.9 Available MIXED_FLOW Elements

The available mixed continuum elements with *continuous* pressure approximations for analyses that include the flow of pore fluid are listed below. These elements are entered using the ELEMENT MIXED_FLOW commands.

- General information regarding these element types is given in Chapter 28.

18.9.1 "Taylor-Hood" Elements

This element sub-class contains two- and three-dimensional mixed finite elements with *continuous* linear pressure approximations and a displacement interpolation that is one order *higher* than the pressure interpolation. Such elements are sometimes referred to as "Taylor-Hood" elements, in recognition of the pioneering use of this element class by C. Taylor and P. Hood [103; 43] for solving problems in computational fluid dynamics (CFD). Since such elements are known to generally yield stable results, they have been used rather extensively in the past.

18.9.1.1 Type T6P3c Mixed_Flow Elements

The **T6P3c** is a 6-node mixed, parametrically mapped triangular element. Uses a *quadratic* displacement interpolation and a continuous *linear* pressure interpolation assumed.

- The command syntax is available in Section 28.2.

18.9.1.2 Type Q8P4c Mixed_Flow Elements

The **Q8P4c** is an 8-node mixed, parametrically mapped quadrilateral element. It uses a *quadratic* displacement interpolation and a continuous *bilinear* pressure interpolation.

- The command syntax is available in Section 28.6.

18.9.1.3 Type Q9P4c Mixed_Flow Elements

The **Q9P4c** is a 9-node mixed, parametrically mapped quadrilateral element. It uses a *biquadratic* displacement interpolation and a continuous *bilinear* pressure interpolation.

- The command syntax is available in Section 28.7.

18.9.1.4 Type H20P8c Mixed_Flow Elements

The **H20P8c** is a 20-node mixed, parametrically mapped hexahedral element. It uses a *quadratic* displacement interpolation and a continuous *trilinear* pressure interpolation.

- The command syntax is available in Section 28.10.

18.9.1.5 Type H27P8c Mixed_Flow Elements

The **H27P8c** is a 27-node mixed, parametrically mapped hexahedral element. It uses a *triquadratic* displacement interpolation and a continuous *trilinear* pressure interpolation.

- The command syntax is available in Section 28.11.

18.9.2 "Bubble Function" Elements

This element sub-class includes two- and three-dimensional mixed finite elements with continuous pressure approximations that employ a so-called displacement "bubble" function. "Bubble" functions are additional displacement degrees of freedom that are included in an element formulation so as to provide the missing spatial distributions that will relax shear locking and allow for bending response to be properly simulated [85]. In the case of mixed elements possessing continuous pressure approximations, bubble functions are included in the hope that they improve the element response in the incompressible limit; i.e., under undrained conditions.

It is timely to note that "bubble" function elements are *conforming*. This is in contrast to nonconforming (or incompatible) elements that also have interior degrees of freedom, but whose remaining interpolation functions are not modified to account for the presence of these additional degrees of freedom.

18.9.2.1 Type T4P3c Mixed_Flow Elements

The **T4P3c** is a 4-node mixed, isoparametric triangular element. It uses an "enhanced" *linear* displacement interpolation and a continuous *linear* pressure interpolation.

- The command syntax is available in Section 28.1.

18.9.2.2 Type T7P3c Mixed_Flow Elements

The **T7P3c** is a 7-node mixed, isoparametric triangular element. It uses an "enhanced" *quadratic* displacement interpolation and a continuous *linear* pressure interpolation.

- The command syntax is available in Section 28.3.

18.9.2.3 Type Q5P4c Mixed_Flow Elements

The **Q5P4c** is a 5-node mixed, isoparametric quadrilateral element. It uses an "enhanced" *bilinear* displacement interpolation and a continuous *bilinear* pressure interpolation.

- The command syntax is available in Section 28.5.

18.9.2.4 Type H9P8c Mixed_Flow Elements

The **H9P8c** is a 9-node mixed, isoparametric hexahedral element. It uses an "enhanced" *trilinear* displacement interpolation and a continuous *trilinear* pressure interpolation.

- The command syntax is available in Section 28.9.

18.9.3 Equal-Order of Interpolation (EOI) Elements

This element sub-class includes two- and three-dimensional linear mixed finite elements with *continuous* pressure approximations that employ an equal-order interpolation (EOI) along with a *hybrid* reduced integration scheme.

18.9.3.1 Type Q4P4c Mixed_Flow Elements

The **Q4P4c** is a 4-node mixed, conforming isoparametric quadrilateral element with *hybrid* selective reduced integration. It uses a *bilinear* displacement interpolation and a continuous *bilinear* pressure interpolation.

- The command syntax is available in Section 28.12.

18.9.3.2 Type H8P8c Mixed_Flow Elements

The **H8P8c** is an 8-node mixed, isoparametric hexahedral element with *hybrid* selective reduced integration. It uses a *trilinear* displacement interpolation and a continuous *trilinear* pressure interpolation.

- The command syntax is available in Section 28.13.

18.9.4 Higher-Order Elements

This element sub-class includes mixed finite elements with *continuous* quadratic pressure approximations.

18.9.4.1 Type T15P6c Mixed_Flow Elements

The **T15P6c** is a 15-node mixed, isoparametric triangular element. It uses a *quartic* displacement interpolation and a *quadratic* pressure interpolation.

- The command syntax is available in Section 28.4.

18.9.4.2 Type Q17P8c Mixed_Flow Elements

The **Q17P8c** is a 17-node mixed, isoparametric quadrilateral element. It uses a *quartic* displacement interpolation and a *quadratic* pressure interpolation.

- The command syntax is available in Section 28.8.

18.10 Available SCALAR Elements

The available continuum elements suitable for scalar field analyses are listed below. Such elements are entered using the ELEMENT SCALAR commands.

- General information regarding these element types is given in Chapter 29.

18.10.1 Type T3S Scalar Elements

The **T3S** is an irreducible, 3-node linear isoparametric triangular continuum element for use in the analysis of scalar field problems.

- The command syntax is available in Section 29.1.

18.10.2 Type Q4S Scalar Elements

The **Q4S** is an irreducible, 4-node bi-linear isoparametric quadrilateral continuum element for use in the analysis of scalar field problems.

- The command syntax is available in Section 29.2.

18.10.3 Type Q8S Scalar Elements

The **Q8S** is an irreducible, 8-node bi-quadratic isoparametric quadrilateral continuum element for use in the analysis of scalar field problems.

- The command syntax is available in Section 29.3.

18.10.4 Type TET4S Scalar Elements

The **TET4S** is an irreducible, 4-node trilinear isoparametric tetrahedral continuum element for use in the analysis of scalar field problems.

- The command syntax is available in Section 29.4.

18.10.5 Type H8S Scalar Elements

The **H8S** is an irreducible, 8-node trilinear isoparametric hexahedral continuum element for use in the analysis of scalar field problems.

- The command syntax is available in Section 29.5.

Chapter 19

ELEMENT BAR_MECHANICAL Commands

The available line (bar) elements for mechanical analyses are listed below. These elements are entered using the **ELEMENT BAR_MECHANICAL** commands.

- Type **L2FDP0**: a 2-node irreducible (linear displacement) finite deformation "membrane" element for mechanical analyses. The command syntax for this element type is available in Section 19.1.
- Type **L2P0**: a 2-node irreducible (linear displacement) line (bar) element for two- or three-dimensional mechanical analyses. The command syntax for this element type is available in Section 19.2.
- Type **L3P0**: a 3-node (irreducible (quadratic displacement) line (bar) element for two- or three-dimensional mechanical analyses. The command syntax for this element type is available in Section 19.3.

19.1 The ELEMENT BAR_MECHANICAL TYPE L2FDP0 Command

Synopsis

The **ELEMENT BAR_MECHANICAL L2FDP0** command is used to describe all irreducible 2-node finite deformation "membrane" elements that are to be used in mechanical analyses.

Syntax

The following syntax is used to describe a typical **L2FDP0** irreducible bar element:

ELEment BAR_Mechanical TYPe L2FDP0 (DEScription "string")
NODes #:#:# (MATerial #)
(CONstruction #) (EXCavation #) (AREa #.#)
(FORce_initial #.#) (STRain_initial #.#)
(1_Additional #) (1_Increment #)
(DONT_PRINT_Results)

Explanatory Notes

- The **L2FDP0** is an irreducible, linear, finite deformation "membrane" element. The element
  - Contains two (2) nodes.
  - Has two (2) displacement degrees of freedom at each node.
  - Possesses a total of four (4) displacement degrees of freedom.

  This element, in conjunction with a linear viscoelastic material idealization, has been used to characterize the time-dependent behavior of polymeric inclusions used to reinforce soil embankment [98].
- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENTS OFF** or the **ECHO ELEMENT_DESCRIPTIONS OFF** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.

- The numbering order of **NODES** associated with the **L2FDP0** element is shown in Figure 19.1.

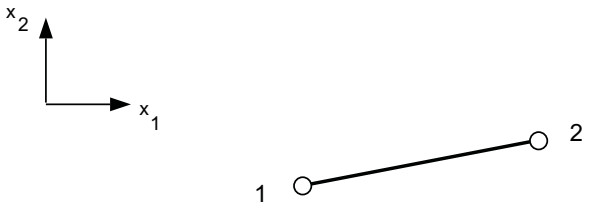

Figure 19.1: Node numbering associated with typical 2-node linear finite deformation (L2FDP0) "membrane" element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The *default* values for the **MATERIAL** number is one (1).

  The following material idealizations are currently available for use with **L2FDP0** elements:

  - Isotropic linear elastic (**MATERIAL ELASTIC ISOTROPIC**)
  - Time-independent, quasilinear idealization for geosynthetics (**MATERIAL GEOSYNTHETIC LING_TATSUOKA**)
  - Time-dependent idealization for geosynthetics (**MATERIAL GEOSYNTHETIC D_K**)
  - Viscoelastic material idealization (**MATERIAL VISCOELASTIC**)

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to, or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the *default* condition, no input is required in such a case. The condition of no excavation is likewise the default.

- The **AREA** keyword is used to specify the element's cross-sectional area. The *default* **AREA** value is equal to 1.0. In plane strain idealizations this constitutes the area per unit width. The cross-sectional area is assumed to be *constant* over a given element.

- The **FORCE_INITIAL** and **STRAIN_INITIAL** commands are used to specify *initial* forces and strains, respectively, acting in the element. The *default* **FORCE_INITIAL** and **STRAIN_INITIAL** values are zero (0.0).

- Specification of the keyword **DONT_PRINT_Results** indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strain, stress and axial force) for this element. If *generation* is performed using this **ELEMENT BAR_MECHANICAL** command, then all the elements generated will be affected in a like manner by the above print control commands.

19.2 The ELEMENT BAR_MECHANICAL TYPE L2P0 Command

Synopsis

The **ELEMENT BAR_MECHANICAL L2P0** command is used to describe all irreducible 2-node bar (line) elements that are to be used in mechanical analyses.

Syntax

The following syntax is used to describe a typical **L2P0** irreducible bar element:

ELEment BAR_Mechanical TYPe L2P0 (DEScription "string")
NODes #:#:# (MATerial #)
(CONstruction #) (EXCavation #) (AREa #.#)
(FORce_initial #.#) (STRain_initial #.#)
(1_Additional #) (1_Increment #)
(DONT_PRINT_Results)

Explanatory Notes

- The **L2P0** is an irreducible, linear, isoparametric Lagrangian element [52].
- The **L2P0** can be used for two-dimensional or three-dimensional analyses. The number of displacement degrees of freedom present at each node in such an element thus depends on the spatial dimension of the model; i.e., two or three. For two-dimensional analyses it possesses four (4) displacement degrees of freedom; for three-dimensional analyses it possesses six (6) displacement degrees of freedom.
- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENTS OFF** or the **ECHO ELEMENT_DESCRIPTIONS OFF** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with the **L2P0** element is shown in Figure 19.2.

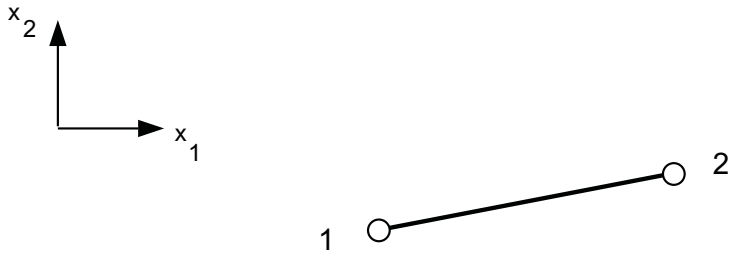

Figure 19.2: Node numbering associated with typical 2-node linear (L2P0) bar element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The *default* values for the **MATERIAL** number is one (1).

  The following material idealizations are currently available for use with **L2P0** elements:

  - Isotropic linear elastic (**MATERIAL ELASTIC ISOTROPIC**)
  - Time-independent, quasilinear idealization for geosynthetics (**MATERIAL GEOSYNTHETIC LING_TATSUOKA**)
  - Time-dependent idealization for geosynthetics (**MATERIAL GEOSYNTHETIC D_K**)
  - Ramberg-Osgood material idealization (**MATERIAL RAMBERG-OSGOOD**)
  - Viscoelastic material idealization (**MATERIAL VISCOELASTIC**)

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to, or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the *default* condition, no input is required in such a case. The condition of no excavation is likewise the default.

- The **AREA** keyword is used to specify the element's cross-sectional area. The *default* **AREA** value is equal to 1.0. In plane strain idealizations this constitutes the area per unit width. The cross-sectional area is assumed to be *constant* over a given element.

  It is possible to *exactly* account for the *linear* variation in cross-sectional area shown in Figure 19.3a. In this case the value specified in conjunction with the **AREA** keyword should be the average of the two values at the nodes; viz.,

$$A^{(e)} = \frac{A_1^{(e)} + A_2^{(e)}}{2}$$

  It is also possible to *exactly* account for the *parabolic* variation in cross-sectional area shown in Figure 19.3b. In this case the following value should be specified in conjunction with the **AREA** keyword:

$$A^{(e)} = \frac{A_1^{(e)} + A_2^{(e)} + 4A_c^{(e)}}{6}$$

The *default* is a unit (1.0) cross-sectional **AREA**.

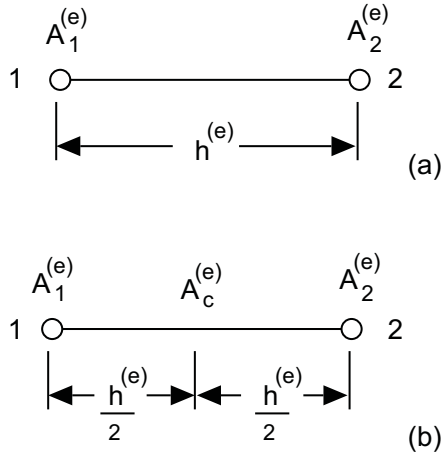

Figure 19.3: Possible definition of cross-sectional area variation associated with typical 2-node linear bar element.

- The **FORCE_INITIAL** and **STRAIN_INITIAL** commands are used to specify *initial* forces and strains, respectively, acting in the element. The *default* **FORCE_INITIAL** and **STRAIN_INITIAL** values are zero (0.0).
- If the body being analyzed can be divided into a layer of elements, and if the characteristics of the frame element (i.e., the **MATERIAL**, the incremental **CONSTRUCTION** and **EXCAVATION** numbers, the **AREA**, **FORCE_INITIAL** and **STRAIN_INITIAL** are the *same* for several elements along a line, and if the nodes are numbered in a *consistent* fashion, then an element data generation option can be employed. To generate a sequence of frame elements along a line, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**.

Example of Command Usage: Generating a Line of L2P0 Elements

The sequence of two-node bar (**L2P0**) elements shown in Figure 19.4 shall be generated. In this figure the circled numbers are element numbers; these are automatically assigned by APES+ once the elements are generated. Each of the elements: (1) Has a cross-sectional area equal to 0.10; (2) Is described by material number 1; and (3) Is "placed" in solution increment 3. The following data is used to describe the line of bar (axial) elements shown in the figure below.

```
element irreducible type "l2p0" nodes 1 2  mat 1  area 0.10  &
 construc 3  1_add  4  1_incr  1
```

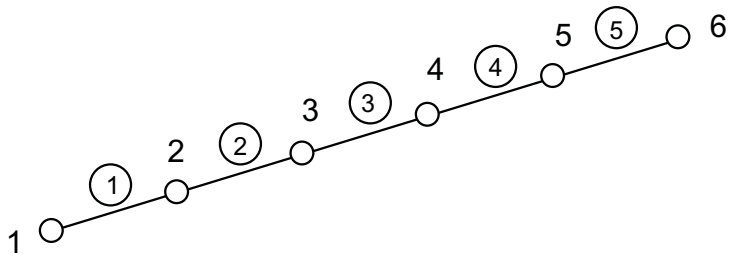

Figure 19.4: Five-element mesh consisting of L2P0 elements.

19.3 The ELEMENT BAR_MECHANICAL TYPE L3P0 Command

Synopsis

The **ELEMENT BAR_MECHANICAL L3P0** command is used to describe all irreducible 3-node bar (line) elements that are to be used in mechanical analyses.

Syntax

The following syntax is used to describe a typical **L3P0** irreducible bar element:

ELEment BAR_Mechanical TYPe L3P0 (DEScription "string")
NODes #:#:# (MATerial #)
(CONstruction #) (EXCavation #) (AREa #.#)
(FORce_initial #.#) (STRain_initial #.#)
(1_Additional #) (1_Increment #)
(DONT_PRINT_Results)

Explanatory Notes

- The **L3P0** is an irreducible, quadratic, isoparametric Lagrangian element [52]. The **L3P0** can be used for two-dimensional or three-dimensional analyses. The number of displacement degrees of freedom present at each node in such an element thus depends on the spatial dimension of the model; i.e., two or three. For two-dimensional analyses it possesses six (6) displacement degrees of freedom; for three-dimensional analyses it possesses nine (9) displacement degrees of freedom.
- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENTS OFF** or the **ECHO ELEMENT_DESCRIPTIONS OFF** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with the **L3P0** element is shown in Figure 19.5.
- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The *default* values for the **MATERIAL** number is one (1).

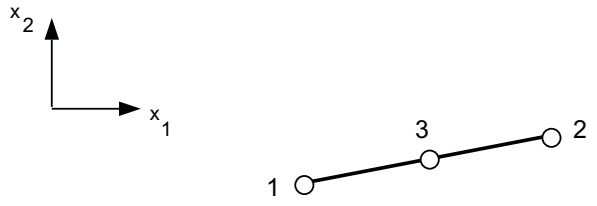

Figure 19.5: Node numbering associated with typical 3-node quadratic (L3P0) bar element.

The following material idealizations are currently available for use with **L3P0** elements:

- Isotropic linear elastic (**MATERIAL ELASTIC ISOTROPIC**)
- Viscoelastic material idealization (**MATERIAL VISCOELASTIC**)

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to, or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the *default* condition, no input is required in such a case. The condition of no excavation is likewise the default.

- The **AREA** keyword is used to specify the element's cross-sectional area. The *default* **AREA** value is equal to 1.0. In plane strain idealizations this constitutes the area per unit width. The cross-sectional area is assumed to be *constant* over a given element.

- The **FORCE_INITIAL** and **STRAIN_INITIAL** commands are used to specify *initial* forces and strains, respectively, acting in the element. The *default* **FORCE_INITIAL** and **STRAIN_INITIAL** values are zero (0.0).

- If the body being analyzed can be divided into a layer of elements, and if the characteristics of the frame element (i.e., the **MATERIAL**, the incremental **CONSTRUCTION** and **EXCAVATION** numbers, the **AREA**, **FORCE_INITIAL** and **STRAIN_INITIAL** are the *same* for several elements along a line, and if the nodes are numbered in a *consistent* fashion, then an element data generation option can be employed. To generate a sequence of frame elements along a line, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**.

- Specification of the keyword **DONT_PRINT_Results** indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strain, stress and axial force) for this element. If *generation* is performed using this **ELEMENT BAR_MECHANICAL** command, then all the elements generated will be affected in a like manner by the above print control commands.

Chapter 20

ELEMENT FLEXURAL Commands

The available *frame* (beam) elements for mechanical analyses are listed below. These elements are entered using the **ELEMENT FLEXURAL** commands.

- Type **BE2_2D**: a 2-node cubic Bernoulli-Euler (C^1 continuous) plane frame element. The command syntax for this element type is available in Section 20.1.
- Type **TB2_2D**: a 2-node linear "Timoshenko" (C^0 continuous) plane frame element. The command syntax for this element type is available in Section 20.2.
- Type **TB3_2D**: a 3-node quadratic "Timoshenko" (C^0 continuous) plane frame element. The command syntax for this element type is available in Section 20.3.

20.1 The ELEMENT FLEXURAL TYPE BE2_2D Command

Synopsis

The ELEMENT FLEXURAL TYPE BE2_2D command is used to describe all 2-node cubic Bernoulli-Euler (C^1 continuous) frame elements that are to be used in planar mechanical analyses.

Syntax

The following syntax is used to describe a typical **BE2_2D** frame element:

ELEment FLExural TYPe BE2_2D (DEScription "string")
NODes #:#:# (MATerial #) (CONstruction #) (EXCavation #)
(AREa #.#) (I22 #.#)
(1_Additional #) (1_Increment #)
(DONT_PRINT_Results)

Explanatory Notes

- The **BE2_2D** is a C^1 continuous frame element for planar analyses. The element
  - Contains two (2) nodes.
  - Has two (2) displacements and two (2) slope degrees of freedom at each node.
  - Employs a *cubic* interpolation for the transverse displacement.
- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENTS OFF** or the **ECHO ELEMENT_DESCRIPTIONS OFF** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with a **BE2_2D** elements is shown in Figure 20.1.
- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The *default* values for the **MATERIAL** number is one (1). The value of the elastic modulus associated with a **BE2_2D** element is specified in a **MATERIAL FLEXURAL ELASTIC** command.

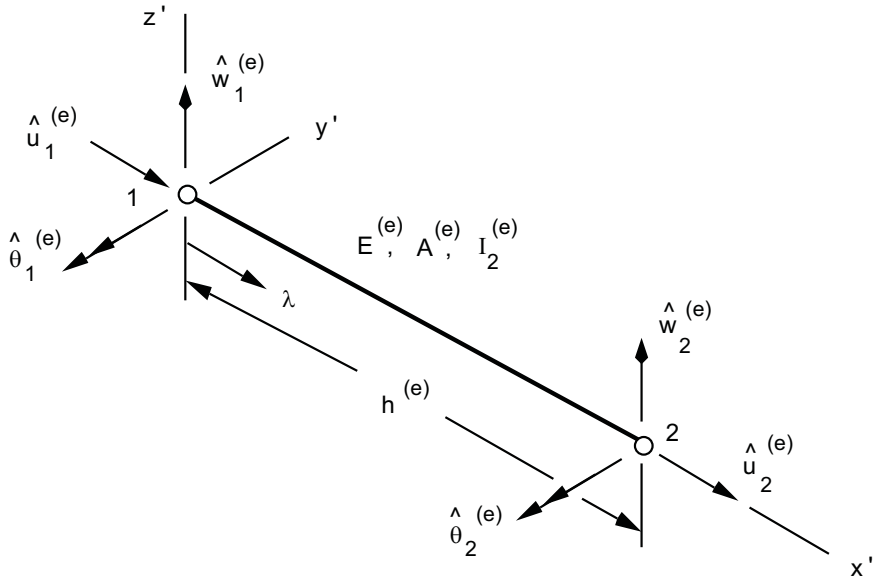

Figure 20.1: Node numbering associated with a typical 2-node cubic Bernoulli-Euler (BE2_2D) frame element.

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to, or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the *default* condition, no input is required in such a case. The condition of no excavation is likewise the default.

- The **AREA** keyword is used to specify the element's cross-sectional area. The *default* **AREA** value is equal to 1.0. Over a given element, the cross-sectional area is assumed to be *constant*.

- The **I22** keyword is used to specify the moment of inertia (second moment of area) for the cross-section with respect to to the element y'-axis (Figure 20.1). The *default* **I22** value is equal to 1.0. The moment of inertia is assumed to be *constant* over a given element.

- If the body being analyzed can be divided into a layer of elements, and if the characteristics of the frame element (i.e., the **MATERIAL**, the incremental **CONSTRUCTION** and **EXCAVATION** numbers, the **AREA**, and **I22**) are the *same* for several elements along a line, and if the nodes are numbered in a *consistent* fashion, then an element data generation option can be employed. To generate a sequence of frame elements along a line, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**.

- Specification of the keyword **DONT_PRINT_Results** indicates that the analyst does not desire to see output of secondary dependent variables (i.e., bending moments and

shear force) for this element. If *generation* is performed using this **ELEMENT FLEXURAL TYPE BE2_2D** command, then all the elements generated will be affected in a like manner by the above print control commands.

20.2 The ELEMENT FLEXURAL TYPE TB2_2D Command

Synopsis

The **ELEMENT FLEXURAL TYPE TB2_2D** command is used to describe all 2-node linear "Timoshenko" (C^0 continuous) frame elements that are to be used in planar mechanical analyses.[1]

Syntax

The following syntax is used to describe a typical **TB2_2D** frame element:

ELEment FLExural TYPe TB2_2D (DEScription "string")
NODes #:#:# (MATerial #) (CONstruction #) (EXCavation #) (INTcode #)
(AREa #.#) (I22 #.#) (SHEar_factor #.#)
(1_Additional #) (1_Increment #)
(DONT_PRINT_Results)

Explanatory Notes

- The **TB2_2D** is a C^0 continuous frame element for planar analyses. The element
  - Contains two (2) nodes.
  - Has two (2) displacements and one (1) rotational degree of freedom at each node.
  - Employs a *linear* interpolation for the axial displacement, the transverse displacement and the fiber rotation.
- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with a **TB2_2D** elements is shown in Figure 20.2.

[1] The range of applicability of standard Bernoulli-Euler beam theory was extended by S. P. Timoshenko [108; 109], who accounted for the effects of transverse shear deformations and, in the case of dynamic excitations and vibrating beams, rotary inertia.

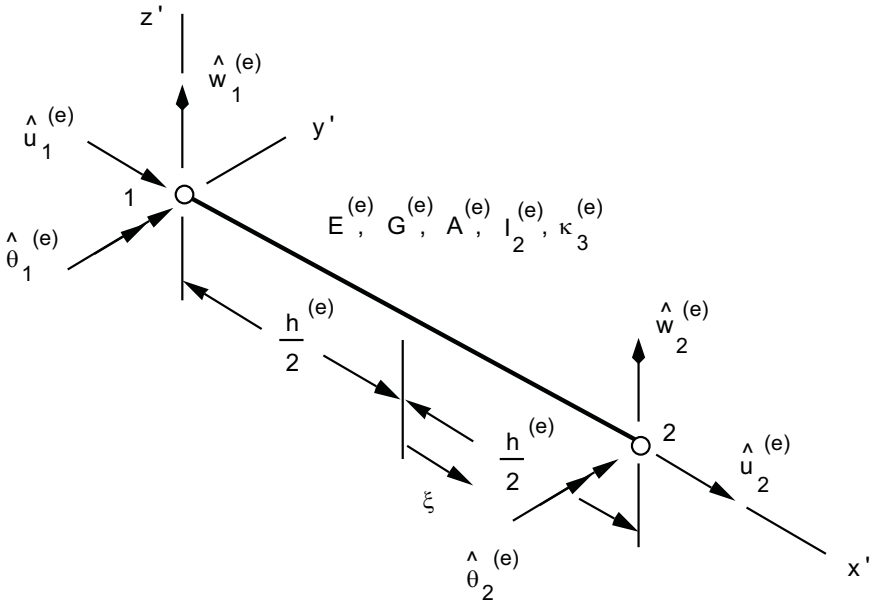

Figure 20.2: Node numbering associated with a typical 2-node linear "Timoshenko" (TB2_2D) frame element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The *default* values for the **MATERIAL** number is one (1). The value of the elastic modulus associated with a **TB2_2D** element is specified in a **MATERIAL FLEXURAL ELASTIC** command.

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the *default* condition, no input is required in such a case. The condition of no excavation is likewise the default.

- The **INTCODE** keyword is used to control selective reduced integration in evaluating the shear stiffness contribution to the element stiffness matrix. The *default* value for **INTCODE** is zero (0), which results in certain terms in the shear stiffness contribution being *under-integrated*. This results superior accuracy as compared to using full integration on all terms associated with the shear stiffness. If **INTCODE** is set equal to 1, *full integration* is indeed used on the shear stiffness contribution. Although this is known to give overly stiff response [44] and is thus generally undesirable, the option is included as a means by which to quantify the benefit of using selective reduced integration.

- The **AREA** keyword is used to specify the element's cross-sectional area. The *default* **AREA** value is equal to 1.0. Over a given element, the cross-sectional area is assumed to be *constant*.

- The **I22** keyword is used to specify the moment of inertia (second moment of area) for the cross-section with respect to to the element y'-axis (Figure 20.2). The *default* **I22** value is equal to 1.0. The moment of inertia is assumed to be *constant* over a given element.

- The **SHEAR_FACTOR** keyword is used to specify a value for shear area correction factor [21] for the cross-ection.

- If the body being analyzed can be divided into a layer of elements, and if the characteristics of the frame element (i.e., the **MATERIAL**, the incremental **CONSTRUCTION** and **EXCAVATION** numbers, the **AREA**, and **I22**) are the *same* for several elements along a line, and if the nodes are numbered in a *consistent* fashion, then an element data generation option can be employed. To generate a sequence of frame elements along a line, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**.

- Specification of the keyword **DONT_PRINT_Results** indicates that the analyst does not desire to see output of secondary dependent variables (i.e., bending moments and shear force) for this element. If *generation* is performed using this **ELEMENT FLEXURAL TYPE TB2_2D** command, then all the elements generated will be affected in a like manner by the above print control commands.

20.3 The ELEMENT FLEXURAL TYPE TB3_2D Command

Synopsis

The **ELEMENT FLEXURAL TYPE TB3_2D** command is used to describe all 3-node quadratic "Timoshenko" (C^0 continuous) frame elements that are to be used in planar mechanical analyses.[2]

Syntax

The following syntax is used to describe a typical **TB3_2D** frame element:

ELEment FLExural TYPe TB3_2D (DEScription "string")
NODes #:#:# (MATerial #) (CONstruction #) (EXCavation #) (INTcode #)
(AREa #.#) (I22 #.#) (SHEar_factor #.#)
(1_Additional #) (1_Increment #)
(DONT_PRINT_Results)

Explanatory Notes

- The **TB3_2D** is a C^0 continuous frame element for planar analyses. The element
  - Contains three (3) nodes.
  - Has two (2) displacements and one (1) rotational degree of freedom at each node.
  - Employs a *quadratic* interpolation for the axial displacement, the transverse displacement and the fiber rotation.
- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with a **TB3_2D** elements is shown in Figure 20.3.

[2]The range of applicability of standard Bernoulli-Euler beam theory was extended by S. P. Timoshenko [108; 109], who accounted for the effects of transverse shear deformations and, in the case of dynamic excitations and vibrating beams, rotary inertia.

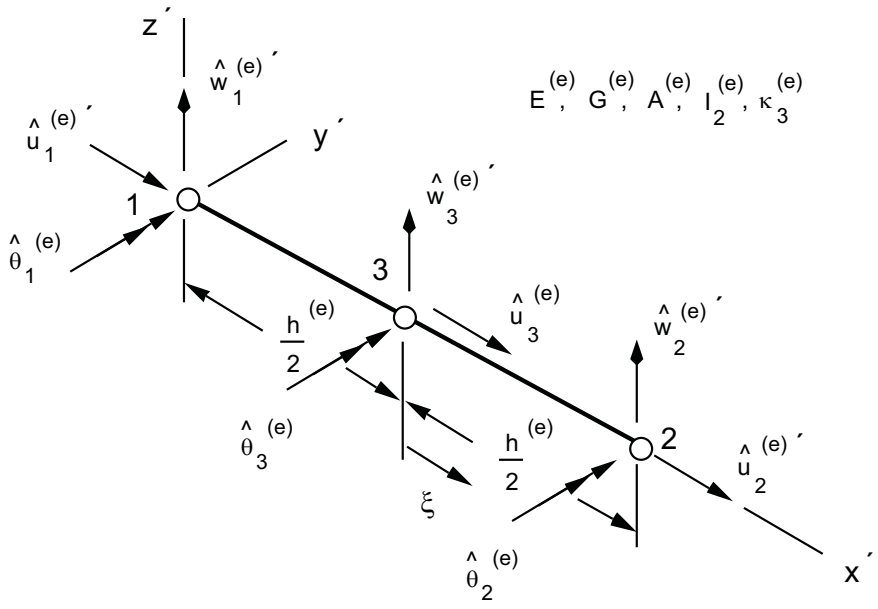

Figure 20.3: Node numbering associated with a typical quadratic "Timoshenko" (TB3_2D) frame element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The *default* values for the **MATERIAL** number is one (1). The value of the elastic modulus associated with a **TB3_2D** element is specified in a **MATERIAL FLEXURAL ELASTIC** command.

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the *default* condition, no input is required in such a case. The condition of no excavation is likewise the default.

- The **INTCODE** keyword is used to control selective reduced integration in evaluating the shear stiffness contribution to the element stiffness matrix. The *default* value for **INTCODE** is zero (0), which results in certain terms in the shear stiffness contribution being under integrated. This results superior accuracy as compared to using full integration on all terms associated with the shear stiffness. If **INTCODE** is set equal to 1, full integration is indeed used on the shear stiffness contribution. Although this is known to give overly stiff response [44] and is thus generally undesirable, the option is included as a means by which to quantify the benefit of using selective reduced integration.

- The **AREA** keyword is used to specify the element's cross-sectional area. The *default* **AREA** value is equal to 1.0. Over a given element, the cross-sectional area is assumed to be *constant*.

- The **I22** keyword is used to specify the moment of inertia (second moment of area) for

the cross-section with respect to to the element y'-axis (Figure 20.3). The *default* **I22** value is equal to 1.0. The moment of inertia is assumed to be *constant* over a given element.

- The **SHEAR_FACTOR** keyword is used to specify a value for shear area correction factor [21] for the cross-ection.
- If the body being analyzed can be divided into a layer of elements, and if the characteristics of the frame element (i.e., the **MATERIAL**, the incremental **CONSTRUCTION** and **EXCAVATION** numbers, the **AREA**, and **I22**) are the *same* for several elements along a line, and if the nodes are numbered in a *consistent* fashion, then an element data generation option can be employed. To generate a sequence of frame elements along a line, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**.
- Specification of the keyword **DONT_PRINT_Results** indicates that the analyst does not desire to see output of secondary dependent variables (i.e., bending moments and shear force) for this element. If *generation* is performed using this **ELEMENT FLEXURAL TYPE TB3_2D** command, then all the elements generated will be affected in a like manner by the above print control commands.

Chapter 21

ELEMENT INFINITE Commands

21.1 Introductory Remarks

Certain problems in engineering and other disciplines involve domains of infinite or semi-infinite extent, or problems in which the region of interest is small compared with the surrounding medium. For example, in the area of footing design, many of the standard formulas used are based on elasticity solutions involving loads applied to semi-infinite half-spaces such as the one shown schematically in Figure 21.1. There are two approaches for modeling such domains mathematically in order to conduct finite element analyses.

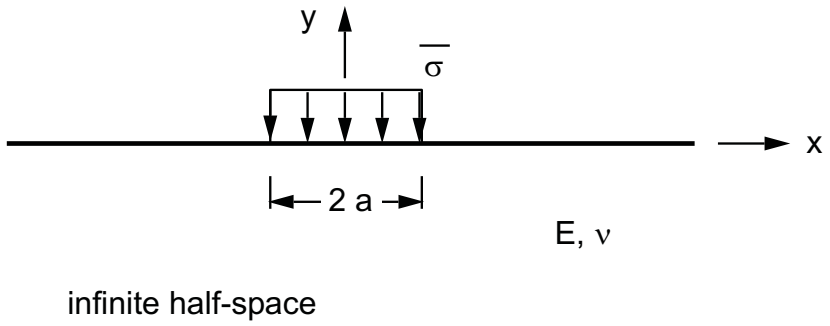

Figure 21.1: Elastic half-space subjected to pressure loading.

In the first approach, the exterior boundary of the solution domain is fixed at a large, but *finite* distance, where the influence of the surrounding medium on the region of interest is considered small enough to be neglected. The resulting domain is only discretized up to this exterior boundary. Figure 21.2 illustrates this approach for the elastic half-space shown in Figure 21.1. Here the extent of the mesh is defined by the dimensions b and d.

This approach involves experimentation with mesh sizes and assumed boundary conditions at the truncated edges of the mesh and has the potential of introducing new possible sources of error. For example, in quasi-static applications a finite domain has a stiffness that

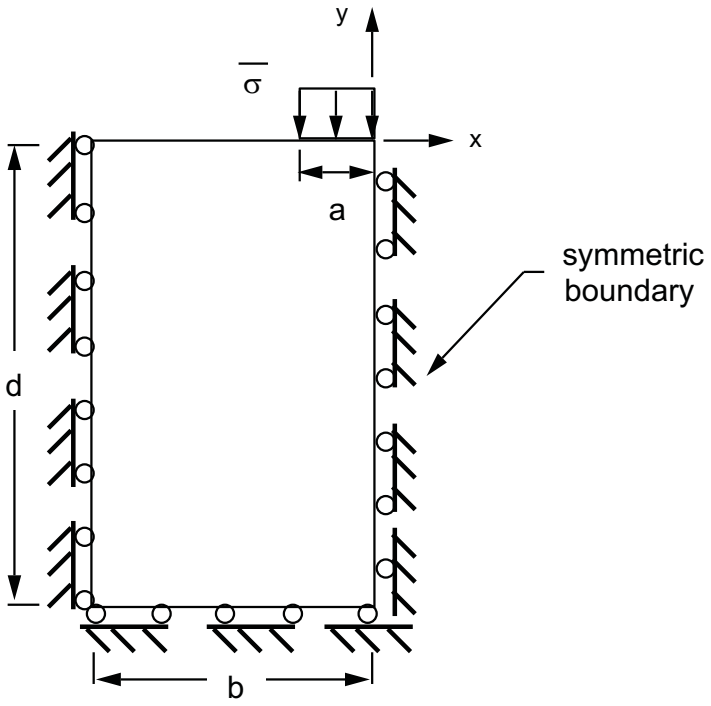

Figure 21.2: Mathematical model of elastic half-space subjected to pressure loading.

differs from an infinite one. In dynamic analyses finite boundaries may reflect energy back into the region being modeled, whereas infinite domains do not. Furthermore the analyst has to have some knowledge about the behavior of the system under consideration in order to judge whether the mathematical model is of sufficient extent. This may require an extensive number of trial analyses to judge the precision of the chosen mesh.

A second approach involves extending the domain to infinity directly. Although analytical solutions valid at large distances can be used, a more elegant approach is to map the infinite domain into a finite region, and to use *suitably modified* finite elements. Bettess [8] published the first paper describing efficient means of for developing such "infinite" elements. The use of infinite elements provides a good means of computational economy for modeling unbounded domains. Figure 21.3 illustrates the use of infinite elements in solving the classical Boussinesq problem of a concentrated load on an elastic half-space. Instead of using a large mesh consisting of standard finite elements, the mesh of the mesh is terminated following the infinite elements.

Since Bettess' 1977 paper, a great number of authors have refined this element type, so that today analysts can make use of this procedure for static and dynamic analyses in a rather straightforward manner. One big advantage of this method is that it merely implies an addition to the element library of the finite element program being used.

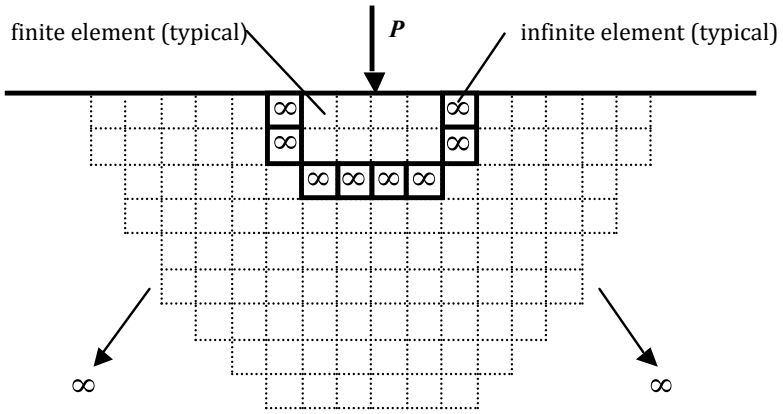

Figure 21.3: Economy provided by infinite elements (after Curnier [23]).

The available "infinite" elements are listed below. These elements are entered using the **ELEMENT INFINITE** commands.

- Type **INFQ6**: a 6-node irreducible "infinite" (linear/quadratic displacement) quadrilateral element. The command syntax for this element type is available in Section 21.2.
- Type **INFQ8**: an 8-node irreducible "infinite" (biquadratic displacement) quadrilateral element. The command syntax for this element type is available in Section 21.3.
- Type **INFQ9**: a 9-node irreducible "infinite" (biquadratic displacement) quadrilateral element. The command syntax for this element type is available in Section 21.4.

21.2 The ELEMENT INFINITE TYPE INFQ6 Command

Synopsis

The **ELEMENT INFINITE TYPE INFQ6** command is used to describe a six-node continuum element of semi-infinite extent for use in conjunction with mechanical analyses.

Syntax

The following syntax is used to describe a six-node continuum element of semi-infinite extent:

ELEment INFinite TYPe INFQ6 (DEScription "string")
NODes #:#:# (MATerial #) (INItial #)
(CONstruction #) (EXCavation #) (THIckness #.#)
(1_Additional #) (1_Increment #)
(2_Additional #) (2_Increment #)

Explanatory Notes

- The semi-infinite **INFQ6** element is compatible with *linear* continuum elements (e.g., **Q4P0** and **Q4P1d** quadrilaterals and **T3P0** triangles).
- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with a typical **INFQ6** element is shown in Figure 21.4. For each such element, the nodes must be specified in ascending numerical order $n1$ to $n6$.

  NOTE: The nodal specifications associated with nodes 5 and 6 of the **INFQ6** element are *automatically* set to *zero displacements* in the global and x_1 and x_2 directions. Consequently, the analyst does *not* need to include these nodes as part of the explicit specifications.

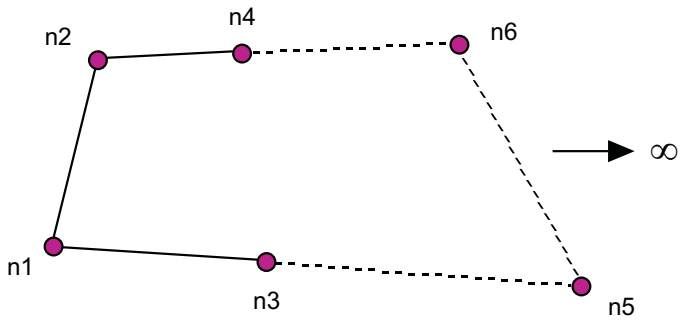

Figure 21.4: Node Numbering Associated with Typical 6-Node **INFQ6** Quadrilateral Semi-Infinite Element (Compatible with Linear Continuum Elements)

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The *default* values for the **MATERIAL** number is one (1).
- The **INITIAL** keyword is used to specify the initial state number associated with the element. The *default* value for the **INITIAL** is zero (0).
- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the *default* condition, no input is required in such a case. The condition of no excavation is likewise the default.
- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The *default* **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE STRAIN** idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.
- If the body being analyzed can be divided into a layer (or layers) of elements, and if the characteristics of the element (i.e., the **MATERIAL**, the **INITIAL** state, the incremental **CONSTRUCTION** and **EXCAVATION** numbers, and the **THICKNESS**) are the *same* for several elements within a layer, and if the nodes are numbered in a *consistent* fashion, then an element data generation option can be employed. To generate a sequence of elements within a single layer, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**. If several layers of elements are made of the *same* material, the above generation option can be carried one step further. This involves suitable use of the **2_ADDITIONAL** and **2_INCREMENT** keywords along with the **1_ADDITIONAL** and **1_INCREMENT** keywords.

Remark: The absence of any print control commands is explained by the observation that the accuracy of secondary dependent variables associated with semi-infinite elements is typically suspect, thus such quantities are simply *not* printed. ◄

Remark: Although the use of infinite elements facilitates the accurate computation of primary and secondary dependent variables in standard continuum elements such as the **Q4P0**, **T3P0**, etc., it is important to note that for an infinite element, the accuracy of the primary and secondary dependent variables is by no means assured. For these reason, these quantities are NOT printed for infinite elements. ◄

Remark: Another fact worthy of note has to do with element distortion. Apart from the "usual" rules concerning element distortion, it is imperative to ensure that for infinite elements those edges that extend to infinity are either parallel or are diverging. Consequently, the configuration shown in Figure 21.5 should be avoided. ◄

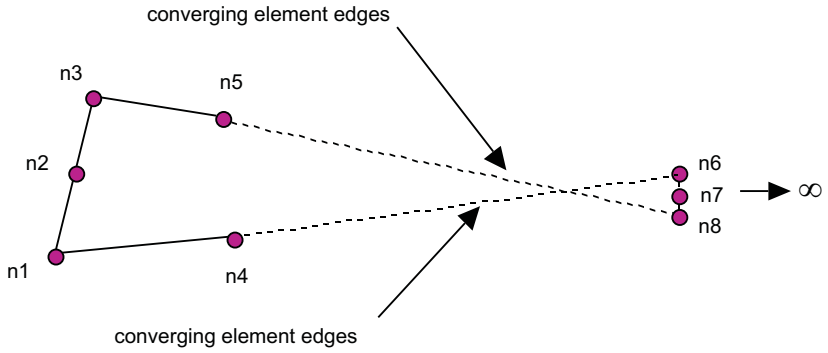

Figure 21.5: Improper Infinite Element Configuration

21.3 The ELEMENT INFINITE TYPE INFQ8 Command

Synopsis

The **ELEMENT INFINITE TYPE INFQ8** command is used to describe a eight-node continuum element of semi-infinite extent for use in conjunction with mechanical analyses.

Syntax

The following syntax is used to describe an eight-node continuum element of semi-infinite extent:

ELEment INFinite TYPe INFQ8 (DEScription "string")
NODes #:#:# (MATerial #) (INItial #)
(CONstruction #) (EXCavation #) (THIckness #.#)
(1_Additional #) (1_Increment #)
(2_Additional #) (2_Increment #)

Explanatory Notes

- The semi-infinite **INFQ8** element is compatible with *quadratic* continuum elements (e.g., **Q8P0**, **Q9P0**, **Q4P4c** quadrilaterals and **T6P0**, **T6P1d** triangular elements).
- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with a typical quadratic **INFQ8** element is shown in Figure 21.6. For each such element, the nodes must be specified in ascending numerical order $n1$ to $n8$.

  NOTE: For the **INFQ8** element the nodal specifications associated with nodes 6, 7 and 8 are *automatically* set to zero displacements in the global x_1 and x_2 directions. Consequently, the analyst does not need to include these nodes as part of the explicit specifications.

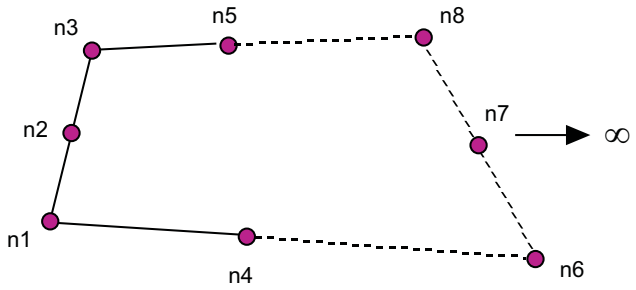

Figure 21.6: Node Numbering Associated with Typical 8-Node **INFQ8** Quadrilateral Semi-Infinite Element (Compatible with Quadratic Continuum Elements)

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The *default* values for the **MATERIAL** number is one (1).
- The **INITIAL** keyword is used to specify the initial state number associated with the element. The *default* value for the **INITIAL** is zero (0).
- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the *default* condition, no input is required in such a case. The condition of no excavation is likewise the default.
- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The *default* **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE STRAIN** idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.
- If the body being analyzed can be divided into a layer (or layers) of elements, and if the characteristics of the element (i.e., the **MATERIAL**, the **INITIAL** state, the incremental **CONSTRUCTION** and **EXCAVATION** numbers, and the **THICKNESS**) are the *same* for several elements within a layer, and if the nodes are numbered in a *consistent* fashion, then an element data generation option can be employed. To generate a sequence of elements within a single layer, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**. If several layers of elements are made of the *same* material, the above generation option can be carried one step further. This involves suitable use of the **2_ADDITIONAL** and **2_INCREMENT** keywords along with the **1_ADDITIONAL** and **1_INCREMENT** keywords.

Remark: The absence of any print control commands is explained by the observation that the accuracy of secondary dependent variables associated with semi-infinite elements is typically suspect, thus such quantities are simply *not* printed. ◄

Remark: Although the use of infinite elements facilitates the accurate computation of primary and secondary dependent variables in standard continuum elements such as the **Q8P0**, **T6P0**, etc., it is important to note that for an infinite element, the accuracy of the primary and secondary dependent variables is by no means assured. For these reason, these quantities are NOT printed for infinite elements. ◄

Remark: Another fact worthy of note has to do with element distortion. Apart from the "usual" rules concerning element distortion, it is imperative to ensure that for infinite elements those edges that extend to infinity are either parallel or are diverging. Consequently, the configuration shown in Figure 21.7 should be avoided. ◄

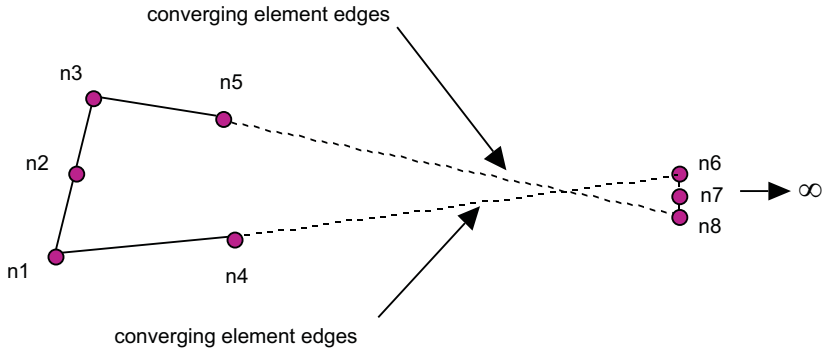

Figure 21.7: Improper Infinite Element Configuration

21.4 The ELEMENT INFINITE TYPE INFQ9 Command

Synopsis

The **ELEMENT INFINITE TYPE INFQ9** command is used to describe a nine-node continuum element of semi-infinite extent for use in conjunction with mechanical analyses.

Syntax

The following syntax is used to describe an nine-node continuum element of semi-infinite extent:

ELEment INFinite TYPe INFQ9 (DEScription "string")
NODes #:#:# (MATerial #) (INItial #)
(CONstruction #) (EXCavation #) (THIckness #.#)
(1_Additional #) (1_Increment #)
(2_Additional #) (2_Increment #)

Explanatory Notes

- The semi-infinite **INFQ9** element is compatible with *quadratic* continuum elements (e.g., **Q8P0**, **Q9P0**, **Q4P4c** quadrilaterals and **T6P0**, **T6P1d** triangular elements).
- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with a typical quadratic **INFQ9** element is shown in Figure 21.8. For each such element, the nodes must be specified in ascending numerical order $n1$ to $n9$.

  NOTE: For the **INFQ9** element the nodal specifications associated with nodes 7, 8 and 9 are *automatically* set to zero displacements in the global x_1 and x_2 directions. Consequently, the analyst does not need to include these nodes as part of the explicit specifications.

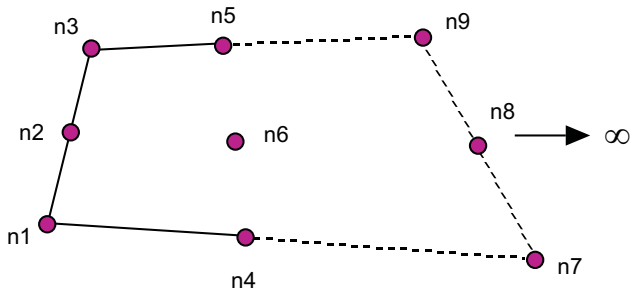

Figure 21.8: Node Numbering Associated with Typical 9-Node **INFQ9** Quadrilateral Semi-Infinite Element (Compatible with Quadratic Continuum Elements)

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The *default* values for the **MATERIAL** number is one (1).
- The **INITIAL** keyword is used to specify the initial state number associated with the element. The *default* value for the **INITIAL** is zero (0).
- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the *default* condition, no input is required in such a case. The condition of no excavation is likewise the default.
- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The *default* **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE STRAIN** idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.
- If the body being analyzed can be divided into a layer (or layers) of elements, and if the characteristics of the element (i.e., the **MATERIAL**, the **INITIAL** state, the incremental **CONSTRUCTION** and **EXCAVATION** numbers, and the **THICKNESS**) are the *same* for several elements within a layer, and if the nodes are numbered in a *consistent* fashion, then an element data generation option can be employed. To generate a sequence of elements within a single layer, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**. If several layers of elements are made of the *same* material, the above generation option can be carried one step further. This involves suitable use of the **2_ADDITIONAL** and **2_INCREMENT** keywords along with the **1_ADDITIONAL** and **1_INCREMENT** keywords.

Remark: The absence of any print control commands is explained by the observation that the accuracy of secondary dependent variables associated with semi-infinite elements is typically suspect, thus such quantities are simply *not* printed. ◄

Remark: Although the use of infinite elements facilitates the accurate computation of primary and secondary dependent variables in standard continuum elements such as the **Q8P0**, **T6P0**, etc., it is important to note that for an infinite element, the accuracy of the primary and secondary dependent variables is by no means assured. For these reason, these quantities are NOT printed for infinite elements. ◄

Remark: Another fact worthy of note has to do with element distortion. Apart from the "usual" rules concerning element distortion, it is imperative to ensure that for infinite elements those edges that extend to infinity are either parallel or are diverging. Consequently, the configuration shown in Figure 21.9 should be avoided. ◄

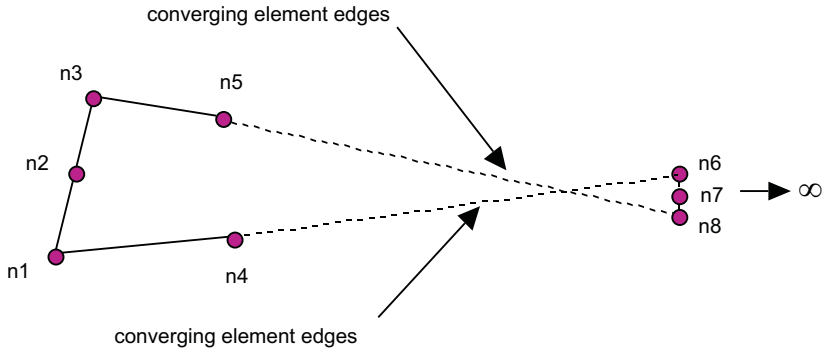

Figure 21.9: Improper Infinite Element Configuration

Chapter 22

ELEMENT INTERFACE Commands

The available interface elements for mechanical analyses are listed below. These elements are entered using the **ELEMENT INTERFACE** commands.

- Type **LK1**: a 4-node (linear) kinematically consistent *zero-thickness* interface element developed by Kaliakin and Li [57]. The command syntax for this element type is available in Section 22.1.
- Type **LRH**: a 4-node (linear) *zero-thickness* ("link") interface element originally developed by Herrmann [39]. The command syntax for this element type is available in Section 22.2.

22.1 The ELEMENT INTERFACE TYPE LK1 Command

Synopsis

The **ELEMENT INTERFACE TYPE LK1** command is used to describe all 4-node (linear) kinematically consistent *zero-thickness* interface element [57] that are to be used in mechanical analyses.

Syntax

The following syntax is used to describe an LK1 zero-thickness interface element:

ELEment INTerface TYPe LK1
NODes #:#:# (MATerial #) (INItial #)
(CONstruction #) (EXCavation #) (THIckness #.#)
(1_Additional #) (1_Increment #)
(2_Additional #) (2_Increment #)

Explanatory Notes

Figure 22.1 shows the numbering order of **NODES** associated with a typical **LK1** element.

Figure 22.1: Node numbering associated with typical 4-node (**LK1**) zero-thickness interface element [57].

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The *default* values for the **MATERIAL** number is one (1).
- The **INITIAL** keyword is used to specify the initial state number associated with the element. The *default* value for the **INITIAL** is zero (0).

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the *default* condition, no input is required in such a case. The condition of no excavation is likewise the default.

- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The *default* **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE STRAIN** idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.

- If the body being analyzed can be divided into a layer (or layers) of elements, and if the characteristics of the element (i.e., the **MATERIAL**, the **INITIAL** state, the incremental **CONSTRUCTION** and **EXCAVATION** numbers, and the **THICKNESS**) are the *same* for several elements within a layer, and if the nodes are numbered in a *consistent* fashion, then an element data generation option can be employed. To generate a sequence of elements within a single layer, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**. If several layers of elements are made of the *same* material, the above generation option can be carried one step further. This involves suitable use of the **2_ADDITIONAL** and **2_INCREMENT** keywords along with the **1_ADDITIONAL** and **1_INCREMENT** keywords.

22.2 The ELEMENT INTERFACE TYPE LRH Command

Synopsis

The **ELEMENT INTERFACE TYPE LRH** command is used to describe all 4-node (linear) *zero-thickness* ("link") interface elements [39] that are to be used in mechanical analyses.

Syntax

The following syntax is used to describe an LRH zero-thickness interface element:

ELEment INTerface TYPe LRH
NODes #:#:# (MATerial #) (INItial #)
(CONstruction #) (EXCavation #) (THIckness #.#)
(1_Additional #) (1_Increment #)
(2_Additional #) (2_Increment #)

Explanatory Notes

Figure 22.2 shows the numbering order of **NODES** associated with a typical **LRH** element.

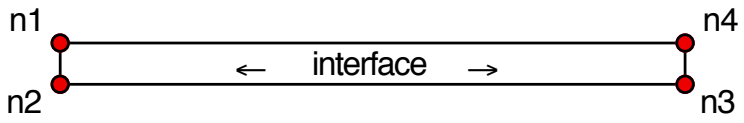

Figure 22.2: Node Numbering associated with typical 4-node (**LRH**) zero-thickness ("link") interface element [39].

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The *default* values for the **MATERIAL** number is one (1).

- The **INITIAL** keyword is used to specify the initial state number associated with the element. The *default* value for the **INITIAL** is zero (0).

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the *default* condition, no input is required in such a case. The condition of no excavation is likewise the default.

- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The *default* **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE STRAIN** idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.

- If the body being analyzed can be divided into a layer (or layers) of elements, and if the characteristics of the element (i.e., the **MATERIAL**, the **INITIAL** state, the incremental **CONSTRUCTION** and **EXCAVATION** numbers, and the **THICKNESS**) are the *same* for several elements within a layer, and if the nodes are numbered in a *consistent* fashion, then an element data generation option can be employed. To generate a sequence of elements within a single layer, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**. If several layers of elements are made of the *same* material, the above generation option can be carried one step further. This involves suitable use of the **2_ADDITIONAL** and **2_INCREMENT** keywords along with the **1_ADDITIONAL** and **1_INCREMENT** keywords.

Chapter 23

ELEMENT IRREDUCIBLE
Commands Involving Triangles

The available **TRIANGULAR** irreducible elements are listed below. These elements are entered using the **ELEMENT IRREDUCIBLE** commands.

Type **T3P0**: an irreducible, 3-node *linear* displacement isoparametric triangular element for mechanical analyses.

- The command syntax for this element type is available in Section 23.1.

Type **T6P0**: an irreducible, 6-node *quadratic* displacement parametrically mapped triangular element for mechanical analyses.

- The command syntax for this element type is available in Section 23.2.

Type **T7P0**: a 7-node irreducible, "enhanced" *quadratic* displacement parametrically mapped triangular element for mechanical analyses. This element adds a cubic "bubble" mode to the quadratic **T6P0** element. **NOTE**: this element is meant for research purposes and it is *not* recommended for general use.

- The command syntax for this element type is available in Section 23.3.

Type **T10P0**: a 10-node irreducible, *cubic* displacement parametrically mapped triangular element for mechanical analyses.

- The command syntax for this element type is available in Section 23.4.

Type **T15P0**: a 15-node irreducible, *quartic* displacement parametrically mapped triangular element for mechanical analyses.

- The command syntax for this element type is available in Section 23.5.

23.1 The ELEMENT IRREDUCIBLE TYPE T3P0 Command

Synopsis

The **ELEMENT IRREDUCIBLE TYPE T3P0** command is used to describe all irreducible, 3-node linear triangular continuum elements that are to be used in mechanical analyses.

Syntax

The following syntax is used to describe a typical **T3P0** irreducible quadrilateral continuum element:

ELEment IRReducible TYPe T3P0 (DEScription "string")
NODes #:#:# (MATerial #) (INItial #)
(CONstruction #) (EXCavation #) (THIckness #.#)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLUMETRIC_STRAIN)

Explanatory Notes

- The **T3P0** is an irreducible, linear, isoparametric triangular continuum element. The element
  - Contains three (3) vertex nodes.
  - Has two (2) displacements degrees of freedom at each node.
  - Possesses a total of six (6) displacement degrees of freedom.
- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.

- The numbering order of **NODES** associated with **T3P0** elements, which must be specified sequentially from 1 to 3, is shown in Figure 23.1.

  NOTE: Presently APES+ does *not* possess the ability to generate **T3P0** elements. It is assumed that the analyst will use some stand alone pre-processing software to accomplish this task. The resulting element and node data will then be translated to the format expected by APES*.

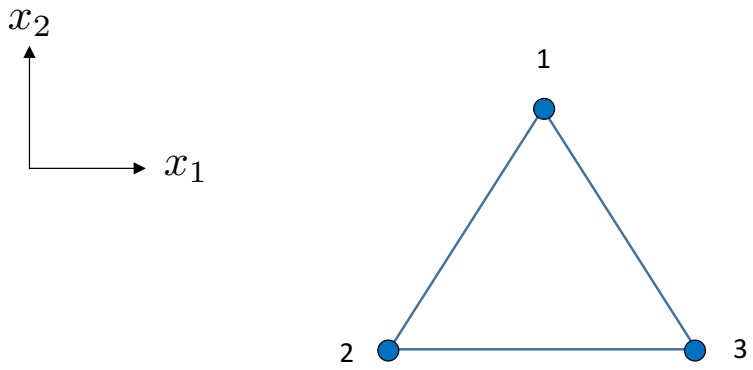

Figure 23.1: Node numbering associated with a typical irreducible linear 3-node (**T3P0**) triangular element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default values for the **MATERIAL** number is one (1).

- The **INITIAL** keyword is used to specify the initial state number associated with the element. The default value for the **INITIAL** is zero (0).

- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The default **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE STRAIN** idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to **zero** corresponds to a material in existence at the beginning of the analysis. Since this is the default condition, no input is required in such a case. The condition of no excavation is likewise the default.

- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES*. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates

inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.
- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the default condition both strains are printed.
- Specification of the **DONT_PRINT_STRESSES** keyword indicates that stresses are not to be printed. Under the default condition stresses are printed.
- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.
- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.
- The **PRINT_VOLUMETRIC_STRAIN** keyword causes the volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the volumetric strain to the absolute value of the minimum non-zero normal strain in the element is printed. That is,

$$\varepsilon_{ratio} = \frac{\left|\hat{\varepsilon}^{(e)}_{vol}\right|}{\sqrt{\left(\hat{\varepsilon}^{(e)}_{11}\right)^2 + \left(\hat{\varepsilon}^{(e)}_{22}\right)^2 + \left(\hat{\varepsilon}^{(e)}_{33}\right)^2}}$$

This ratio is instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. As such, this keyword would likely *not* be used in conjunction with the **T3P0** element. Under the default condition the volumetric strain and the aforementioned ratio are *not* computed and printed.

23.2 The ELEMENT IRREDUCIBLE TYPE T6P0 Command

Synopsis

The **ELEMENT IRREDUCIBLE TYPE T6P0** command is used to describe all irreducible, 6-node quadratic triangular continuum elements that are to be used in mechanical analyses.

Syntax

The following syntax is used to describe a typical irreducible continuum element:

ELEment IRReducible TYPe T6P0 (DEScription "string")
NODes #:#:# (MATerial #) (INItial #) (THIckness #.#)
(INTcode #) (MAPping_points ##)
(CONstruction #) (EXCavation #)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLUMETRIC_STRAIN)

Explanatory Notes

- The **T6P0** is an irreducible, quadratic, parametrically mapped triangular continuum element [52]. The element
  - Contains three (3) vertex nodes.
  - Contains three (3) mid-side nodes.
  - Has two (2) displacements degrees of freedom at each node.
  - Possesses a total of twelve (12) displacement degrees of freedom.
- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.

- The numbering order of **NODES** associated with **T6P0** elements, which must be specified sequentially from 1 to 6, is shown in Figure 23.2.

  NOTE: Presently APES+ does *not* possess the ability to generate **T6P0** elements. Two options are, however, available to help in the determination of nodal coordinates.

  First, the analyst can use some stand-alone pre-processing software to accomplish this task. The resulting element and node data must then be translated to the format expected by APES+. When nodal data is supplied to APES+, the ANALYSIS MESH EXTERNAL command must be specified (see Section 5.4).

  Second, if the coordinates of all *vertex* nodes are explicitly input or generated, and if the node numbers associated with all elements are specified using ELEMENT IRREDUCIBLE commands, the LOCATE_coords routines in APES+ will generate the nodal coordinates of all mid-side nodes. This is done assuming that the element has *straight sides*. When nodal data is computed in this manner, the ANALYSIS MESH INTERNAL command must be specified (see Section 5.4).

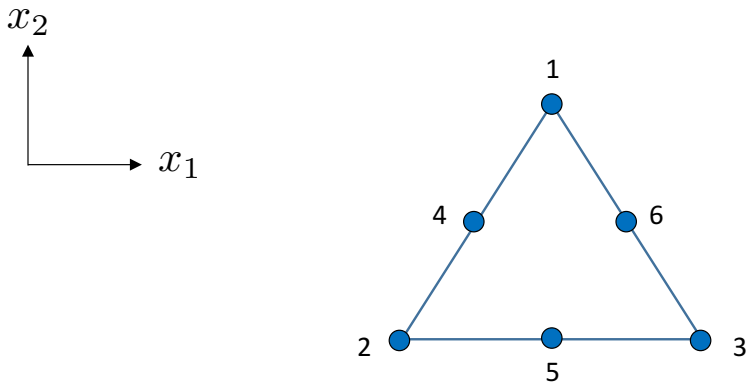

Figure 23.2: Node numbering associated with a typical irreducible quadratic 6-node (**T6P0**) triangular element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default values for the **MATERIAL** number is one (1).
- The **INITIAL** keyword is used to specify the initial state number associated with the element. The default value for the **INITIAL** is zero (0).
- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The default **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE**

STRAIN idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.

- The value specified in conjunction with the **INTCODE** keyword describes the order of numerical integration scheme to be used in developing the element equations for the element.

  The "commonly" used numerical integration rule for **T6P0** elements corresponds to a 3-point numerical integration scheme (degree of precision equal to 2) for the primary dependent variables (i.e., nodal displacements) and a 1-point scheme (degree of precision equal to 1) for the secondary dependent variables (i.e., strains and stresses). This is the default condition and requires no input using the **INTCODE** keyword. If a quadrature order different from the default condition is desired, the following integer values are associated with this keyword:

  INTCODE = 11: A 1-point numerical integration scheme (degree of precision equal to 1) is used to compute *both* the primary dependent variables (i.e., nodal displacements) and for the secondary dependent variables (i.e., strains and stresses).

  This serves to uniformly *under integrate* the element. The disadvantage of uniform reduced integration is that the rank of the element stiffness matrix $\mathbf{K}^{(e)}$ may be reduced, resulting in the singularity or near singularity of the global $\mathbf{K}$ [52]. The associated element instabilities manifest themselves in the form of *mechanisms* or *hourglass modes* [44]. Consequently, the specification of **INTCODE** = 11 should only be used to investigate the possible pathologies associated with uniform reduced integration and *not* in standard analyses.

  INTCODE = 31: a 3-point numerical integration scheme (degree of precision equal to 2) is used to compute the primary dependent variables (i.e., nodal displacements) and a 1-point scheme (degree of precision equal to 1) is used to compute the secondary dependent variables (i.e., strains and stresses). This is equivalent to the aforementioned default setting.

  INTCODE = 34: a 3-point numerical integration scheme (degree of precision equal to 2) is used to compute the primary dependent variables (i.e., nodal displacements). A 4-point scheme (degree of precision equal to 3) is used is used to compute the secondary dependent variables (i.e., strains and stresses). These points are very close to the *optimal* flux sampling points for quadratic triangular elements [89].

  INTCODE = 63: a 6-point numerical integration scheme (degree of precision equal to 4) is used to compute the primary dependent variables (i.e., nodal displacements). A 3-point scheme (degree of precision equal to 2) is used is used to compute the secondary dependent variables (i.e., strains and stresses).

  INTCODE = 64: a 6-point numerical integration scheme (degree of precision equal to 4) is used to compute the primary dependent variables (i.e., nodal displace-

ments). A 4-point scheme (degree of precision equal to 3) is used is used to compute the secondary dependent variables (i.e., strains and stresses).

- The value specified in conjunction with the **MAPPING_POINTS** keyword describes the parametric element mapping [52] to be used in describing the element geometry. The following integer values are associated with this keyword:

  MAPPING_POINTS = 6: An *isoparametric* mapping, corresponding to a *quadratic* representation of the element geometry, is used. Since this is the default condition, the **MAPPING_POINTS** command can be omitted.

  MAPPING_POINTS = 3: A *sub-parametric* mapping, involving a *linear* representation of the element geometry, is used.

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to **zero** corresponds to a material in existence at the beginning of the analysis. Since this is the default condition, no input is required in such a case. The condition of no excavation is likewise the default.

- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the default condition both strains are printed.

- Specification of the **DONT_PRINT_STRESSES** keyword indicates that stresses are not to be printed. Under the default condition stresses are printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_VOLUMETRIC_STRAIN** keyword causes the volumetric strain (ε_{vol}) to be computed and printed for the element. In addition, the following quantities are computed and printed:

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33}) \neq 0$$

$$\varepsilon_{ratio} = \frac{\left|\hat{\varepsilon}_{vol}^{(e)}\right|}{\sqrt{\left(\hat{\varepsilon}_{11}^{(e)}\right)^2 + \left(\hat{\varepsilon}_{22}^{(e)}\right)^2 + \left(\hat{\varepsilon}_{33}^{(e)}\right)^2}}$$

The first quantity is the ratio of the absolute value of ε_{vol} to the absolute value of the minimum non-zero normal strain in the element. The second quantity is the ratio of the absolute value of ε_{vol} to the square root of the sum of squares of the normal strain components. These ratio are instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. As such, this keyword would likely *not* be used in conjunction with the **T6P0** element. Under the default condition the volumetric strain and the aforementioned ratios are *not* computed and printed.

23.3 The ELEMENT IRREDUCIBLE TYPE T7P0 Command

Synopsis

The **ELEMENT IRREDUCIBLE T7P0** command is used to describe all irreducible, 7-node "enhanced" quadratic triangular continuum elements that are to be used in mechanical analyses.

Syntax

The following syntax is used to describe a typical irreducible continuum element:

ELEment IRReducible TYPe T7P0 (DEScription "string")
NODes #:#:# (MATerial #) (INItial #) (THIckness #.#)
(INTcode #) (MAPping_points ##)
(CONstruction #) (EXCavation #)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_AVG_STRAins) (PRINT_AVG_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLUMETRIC_STRAIN)

Explanatory Notes

- The **T7P0** is an irreducible, "enhanced" quadratic, parametrically mapped triangular continuum element [52]. The element
  - Contains three (3) vertex nodes.
  - Contains three (3) mid-side nodes.
  - Contains one (1) mid-side node.
  - Has two (2) displacements degrees of freedom at each node.
  - Possesses a total of fourteen (14) displacement degrees of freedom.
- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.

- The numbering order of **NODES** associated with **T7P0** elements, which must be specified sequentially from 1 to 7, is shown in Figure 23.3.

  NOTE: Presently APES+ does *not* possess the ability to generate **T7P0** elements. Two options are, however, available to help in the determination of nodal coordinates.

  First, the analyst can use some stand-alone pre-processing software to accomplish this task. The resulting element and node data must then be translated to the format expected by APES+. When nodal data is supplied to APES+, the ANALYSIS MESH EXTERNAL command must be specified (see Section 5.4).

  Second, if the coordinates of all *vertex* nodes are explicitly input or generated, and if the node numbers associated with all elements are specified using ELEMENT IRREDUCIBLE commands, the LOCATE_coords routines in APES+ will generate the nodal coordinates of all mid-side nodes, as well as the interior node. This is done assuming that the element has *straight sides*. When nodal data is computed in this manner, the ANALYSIS MESH INTERNAL command must be specified (see Section 5.4).

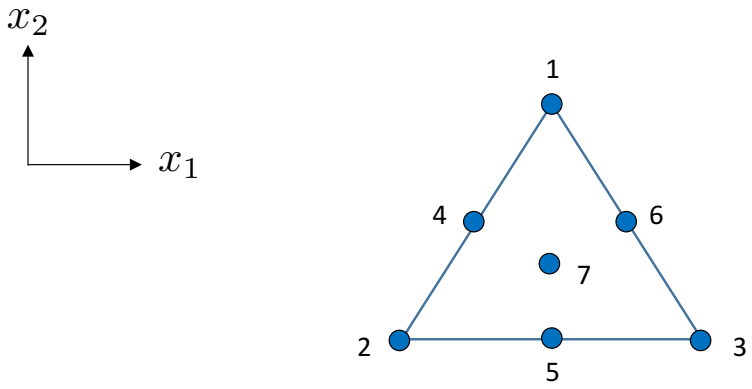

Figure 23.3: Node numbering associated with a typical irreducible 7-node (**T7P0**) triangular element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default values for the **MATERIAL** number is one (1).
- The **INITIAL** keyword is used to specify the initial state number associated with the element. The default value for the **INITIAL** is zero (0).
- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The default **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE**

STRAIN idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.

- The value specified in conjunction with the **INTCODE** keyword describes the order of numerical integration scheme to be used in developing the element equations for the element.

  The "commonly" used numerical integration rule for **T7P0** elements corresponds to a 6-point numerical integration scheme (degree of precision equal to 4) for the primary dependent variables (i.e., nodal displacements) and a 4-point scheme (degree of precision equal to 3) for the secondary dependent variables (i.e., strains and stresses). This is the default condition and requires no input using the **INTCODE** keyword. If a quadrature order different from the default condition is desired, the following integer values are associated with this keyword:

  INTCODE = 31: a 3-point numerical integration scheme (degree of precision equal to 2) is used to compute the primary dependent variables (i.e., nodal displacements) and a 1-point scheme (degree of precision equal to 1) is used to compute the secondary dependent variables (i.e., strains and stresses).

  INTCODE = 63: a 6-point numerical integration scheme (degree of precision equal to 4) is used to compute the primary dependent variables (i.e., nodal displacements). A 3-point scheme (degree of precision equal to 2) is used is used to compute the secondary dependent variables (i.e., strains and stresses).

  INTCODE = 64: a 6-point numerical integration scheme (degree of precision equal to 4) is used to compute the primary dependent variables (i.e., nodal displacements). A 4-point scheme (degree of precision equal to 3) is used is used to compute the secondary dependent variables (i.e., strains and stresses). This is equivalent to the aforementioned default setting.

- The value specified in conjunction with the **MAPPING_POINTS** keyword describes the parametric element mapping [52] to be used in describing the element geometry. The following integer values are associated with this keyword:

  MAPPING_POINTS = 7: An *isoparametric* mapping, corresponding to a *quadratic* representation of the element geometry with a *cubic* term associated with the 7th node, is used. Since this is the default condition, the **MAPPING_POINTS** command can be omitted.

  MAPPING_POINTS = 6: An *sub-parametric* mapping, corresponding to a *quadratic* representation of the element geometry, is used.

  MAPPING_POINTS = 3: A *sub-parametric* mapping, involving a linear representation of the element geometry, is used.

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from

the model. A **CONSTRUCTION** number equal to **zero** corresponds to a material in existence at the beginning of the analysis. Since this is the default condition, no input is required in such a case. The condition of no excavation is likewise the default.

- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the default condition both strains are printed.

- Specification of the **DONT_PRINT_STRESSES** keyword indicates that stresses are not to be printed. Under the default condition stresses are printed.

- The **PRINT_AVG_STRAINS** keyword indicates that average strains (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the default condition average strains are not computed and printed.

- The **PRINT_AVG_STRESSES** keyword indicates that average stresses (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the default condition average stresses are not computed and printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_VOLUMETRIC_STRAIN** keyword causes the volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the volumetric strain to the absolute value of the minimum non-zero normal strain in the element is printed. That is,

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33}) \neq 0$$

  This ratio is instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. As such, this keyword would

likely *not* be used in conjunction with the **T7P0** element. Under the default condition the volumetric strain and the aforementioned ratio are *not* computed and printed.

23.4 The ELEMENT IRREDUCIBLE TYPE T10P0 Command

Synopsis

The **ELEMENT IRREDUCIBLE TYPE T10P0** command is used to describe all irreducible, 10-node cubic triangular continuum elements that are to be used in mechanical analyses.

Syntax

The following syntax is used to describe a typical irreducible continuum element:

ELEment IRReducible TYPe T10P0 (DEScription "string")
NODes #:#:# (MATerial #) (INItial #) (THIckness #.#)
(INTcode #) (MAPping_points ##)
(CONstruction #) (EXCavation #)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_AVG_STRAins) (PRINT_AVG_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLUMETRIC_STRAIN)

Explanatory Notes

- The **T10P0** is an irreducible, cubic, parametrically mapped triangular continuum element. The element
  - Contains three (3) vertex nodes.
  - Contains three (6) mid-side nodes.
  - Contains one (1) interior node.
  - Has two (2) displacements degrees of freedom at each node.
  - Possesses a total of twenty (20) displacement degrees of freedom.
- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.

- The numbering order of **NODES** associated with **T10P0** elements, which must be specified sequentially from 1 to 10, is shown in Figure 23.4.

  NOTE: Presently APES+ does *not* possess the ability to generate **T10P0** elements. Two options are, however, available to help in the determination of nodal coordinates.

  First, the analyst can use some stand-alone pre-processing software to accomplish this task. The resulting element and node data must then be translated to the format expected by APES+. When nodal data is supplied to APES+, the ANALYSIS MESH EXTERNAL command must be specified (see Section 5.4).

  Second, if the coordinates of all *vertex* nodes are explicitly input or generated, and if the node numbers associated with all elements are specified using ELEMENT IRREDUCIBLE commands, the LOCATE_coords routines in APES+ will generate the nodal coordinates of all mid-side nodes, as well as the interior node. This is done assuming that the element has *straight sides*. When nodal data is computed in this manner, the ANALYSIS MESH INTERNAL command must be specified (see Section 5.4).

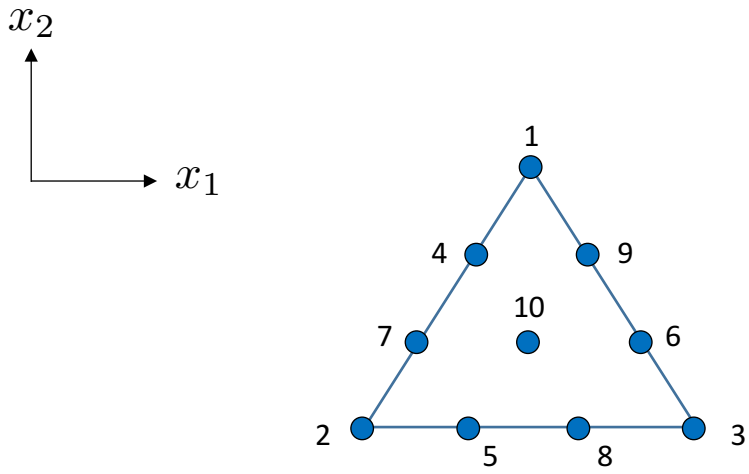

Figure 23.4: Node numbering associated with a typical irreducible cubic 10-node (**T10P0**) triangular element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default values for the **MATERIAL** number is one (1).
- The **INITIAL** keyword is used to specify the initial state number associated with the element. The default value for the **INITIAL** is zero (0).
- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The default

THICKNESS value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE STRAIN** idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.

- The value specified in conjunction with the **INTCODE** keyword describes the order of numerical integration scheme to be used in developing the element equations for the element.

  The "commonly" used numerical integration rule for **T10P0** elements corresponds to a 6-point numerical integration scheme (degree of precision equal to 4) for the primary dependent variables (i.e., nodal displacements) and a 4-point scheme (degree of precision equal to 3) for the secondary dependent variables (i.e., strains and stresses). This is the default condition and requires no input using the **INTCODE** keyword. If a quadrature order different from the default condition is desired, the following integer values are associated with this keyword:

  INTCODE = 43: a 4-point numerical integration scheme (degree of precision equal to 3) is used to compute the primary dependent variables (i.e., nodal displacements) and a 3-point scheme (degree of precision equal to 2) is used to compute the secondary dependent variables (i.e., strains and stresses).

  INTCODE = 63: a 6-point numerical integration scheme (degree of precision equal to 4) is used to compute the primary dependent variables (i.e., nodal displacements). A 3-point scheme (degree of precision equal to 2) is used is used to compute the secondary dependent variables (i.e., strains and stresses).

  INTCODE = 64: a 6-point numerical integration scheme (degree of precision equal to 4) is used to compute the primary dependent variables (i.e., nodal displacements). A 4-point scheme (degree of precision equal to 3) is used is used to compute the secondary dependent variables (i.e., strains and stresses). This is equivalent to the aforementioned default setting.

- The value specified in conjunction with the **MAPPING_POINTS** keyword describes the parametric element mapping [52] to be used in describing the element geometry. The following integer values are associated with this keyword:

  MAPPING_POINTS = 10: An *isoparametric* mapping, corresponding to a *cubic* representation of the element geometry, is used. Since this is the default condition, the **MAPPING_POINTS** command can be omitted.

  MAPPING_POINTS = 6: A *sub-parametric* mapping, involving a *quadratic* representation of the element geometry, is used.

  MAPPING_POINTS = 3: A *sub-parametric* mapping, involving a *linear* representation of the element geometry, is used.

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from

the model. A **CONSTRUCTION** number equal to **zero** corresponds to a material in existence at the beginning of the analysis. Since this is the default condition, no input is required in such a case. The condition of no excavation is likewise the default.

- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the keyword **DONT_PRINT_Results** indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the default condition both strains are printed.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the default condition both strains are printed.

- Specification of the keyword **DONT_PRINT_STRESSES** indicates that stresses are not to be printed. Under the default condition stresses are printed.

- The **PRINT_AVG_STRAINS** keyword indicates that average strains (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the default condition average strains are not computed and printed.

- The **PRINT_AVG_STRESSES** keyword indicates that average stresses (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the default condition average stresses are not computed and printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_VOLUMETRIC_STRAIN** keyword causes the volumetric strain (ε_{vol}) to be computed and printed for the element. In addition, the following quantities are computed and printed:

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33}) \neq 0$$

$$\varepsilon_{ratio} = \frac{\left|\hat{\varepsilon}_{vol}^{(e)}\right|}{\sqrt{\left(\hat{\varepsilon}_{11}^{(e)}\right)^2 + \left(\hat{\varepsilon}_{22}^{(e)}\right)^2 + \left(\hat{\varepsilon}_{33}^{(e)}\right)^2}}$$

The first quantity is the ratio of the absolute value of ε_{vol} to the absolute value of the minimum non-zero normal strain in the element. The second quantity is the ratio of the absolute value of ε_{vol} to the square root of the sum of squares of the normal strain components. These ratio are instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. As such, this keyword would likely *not* be used in conjunction with the **T10P0** element. Under the default condition the volumetric strain and the aforementioned ratios are *not* computed and printed.

23.5 The ELEMENT IRREDUCIBLE TYPE T15P0 Command

Synopsis

The **ELEMENT IRREDUCIBLE TYPE T15P0** command is used to describe all irreducible, 15-node quartic triangular continuum elements that are to be used in mechanical analyses.

Syntax

The following syntax is used to describe a typical irreducible continuum element:

ELEment IRReducible TYPe T15P0 (DEScription "string")
NODes #:#:# (MATerial #) (INItial #) (THIckness #.#)
(INTcode #) (MAPping_points ##)
(CONstruction #) (EXCavation #)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_AVG_STRAins) (PRINT_AVG_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLUMETRIC_STRAIN)

Explanatory Notes

- The **T15P0** is an irreducible, quartic, parametrically mapped triangular continuum element. The element
  - Contains three (3) vertex nodes.
  - Contains three (9) mid-side nodes.
  - Contains one (3) interior node.
  - Has two (2) displacements degrees of freedom at each node.
  - Possesses a total of twenty (30) displacement degrees of freedom.
- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.

- The numbering order of **NODES** associated with **T15P0** elements, which must be specified sequentially from 1 to 15, is shown in Figure 23.5.

  NOTE: Presently APES+ does *not* possess the ability to generate **T15P0** elements. Two options are, however, available to help in the determination of nodal coordinates.

  First, the analyst can use some stand-alone pre-processing software to accomplish this task. The resulting element and node data must then be translated to the format expected by APES+. When nodal data is supplied to APES+, the ANALYSIS MESH EXTERNAL command must be specified (see Section 5.4).

  Second, if the coordinates of all *vertex* nodes are explicitly input or generated, and if the node numbers associated with all elements are specified using ELEMENT IRREDUCIBLE commands, the LOCATE_coords routines in APES+ will generate the nodal coordinates of all mid-side and interior nodes. This is done assuming that the element has *straight sides*. When nodal data is computed in this manner, the ANALYSIS MESH INTERNAL command must be specified (see Section 5.4).

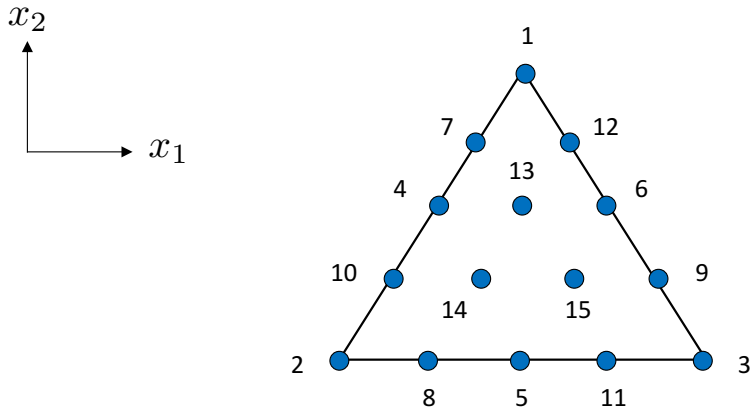

Figure 23.5: Node numbering associated with a typical irreducible quartic 15-node (**T15P0**) triangular element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default values for the **MATERIAL** number is one (1).

- The **INITIAL** keyword is used to specify the initial state number associated with the element. The default value for the **INITIAL** is zero (0).

- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The default **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE**

STRAIN idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.

- The value specified in conjunction with the **INTCODE** keyword describes the order of numerical integration scheme to be used in developing the element equations for the element.

  The "commonly" used numerical integration rule for **T15P0** elements corresponds to a 12-point numerical integration scheme (degree of precision equal to 6) for the primary dependent variables (i.e., nodal displacements) and a 6-point scheme (degree of precision equal to 4) for the secondary dependent variables (i.e., strains and stresses). This is the default condition and requires no input using the **INTCODE** keyword. If a quadrature order different from the default condition is desired, the following integer values are associated with this keyword:

  INTCODE = 64: a 6-point numerical integration scheme (degree of precision equal to 4) is used to compute the primary dependent variables (i.e., nodal displacements). A 4-point scheme (degree of precision equal to 3) is used is used to compute the secondary dependent variables (i.e., strains and stresses).

- The value specified in conjunction with the **MAPPING_POINTS** keyword describes the parametric element mapping [52] to be used in describing the element geometry. The following integer values are associated with this keyword:

  MAPPING_POINTS = 15: An *isoparametric* mapping, corresponding to a *quartic* representation of the element geometry, is used. Since this is the default condition, the **MAPPING_POINTS** command can be omitted.

  MAPPING_POINTS = 6: A *sub-parametric* mapping, involving a *quadratic* representation of the element geometry, is used.

  MAPPING_POINTS = 3: A *sub-parametric* mapping, involving a *linear* representation of the element geometry, is used.

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to **zero** corresponds to a material in existence at the beginning of the analysis. Since this is the default condition, no input is required in such a case. The condition of no excavation is likewise the default.

- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the keyword **DONT_PRINT_Results** indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.
- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the default condition both strains are printed.
- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the default condition both strains are printed.
- Specification of the **DONT_PRINT_STRESSES** keyword indicates that stresses are not to be printed. Under the default condition stresses are printed.
- The **PRINT_AVG_STRAINS** keyword indicates that average strains (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the default condition average strains are not computed and printed.
- The **PRINT_AVG_STRESSES** keyword indicates that average stresses (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the default condition average stresses are not computed and printed.
- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.
- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.
- The keyword **PRINT_VOLUMETRIC_STRAIN** causes the volumetric strain (ε_{vol}) to be computed and printed for the element. In addition, the following quantities are computed and printed:

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33}) \neq 0$$

$$\varepsilon_{ratio} = \frac{\left|\hat{\varepsilon}^{(e)}_{vol}\right|}{\sqrt{\left(\hat{\varepsilon}^{(e)}_{11}\right)^2 + \left(\hat{\varepsilon}^{(e)}_{22}\right)^2 + \left(\hat{\varepsilon}^{(e)}_{33}\right)^2}}$$

The first quantity is the ratio of the absolute value of ε_{vol} to the absolute value of the minimum non-zero normal strain in the element. The second quantity is the ratio of the absolute value of ε_{vol} to the square root of the sum of squares of the normal strain components. These ratio are instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. As such, this keyword would likely *not* be used in conjunction with the **T15P0** element. Under the default condition the volumetric strain and the aforementioned ratios are *not* computed and printed.

Chapter 24

ELEMENT IRREDUCIBLE
Commands Involving Quadrilaterals

The available **QUADRILATERAL** irreducible elements are listed below. These elements are entered using the **ELEMENT IRREDUCIBLE** commands.

Type **Q4P0**: a 4-node irreducible, *bilinear* displacement isoparametric Lagrangian quadrilateral element for mechanical analyses.

- The command syntax for this element type is available in Section 24.1.

Type **Q8P0**: an 8-node irreducible, *quadratic* displacement parametrically mapped "serendipity" quadrilateral element for mechanical analyses.

- The command syntax for this element type is available in Section 24.2.

Type **Q9P0**: a 9-node irreducible, *quadratic* displacement parametrically mapped Lagrangian quadrilateral element for mechanical analyses.

- The command syntax for this element type is available in Section 24.3.

Type **Q12P0**: an 12-node irreducible, *cubic* displacement parametrically mapped "serendipity" quadrilateral element for mechanical analyses.

- The command syntax for this element type is available in Section 24.4.

Type **Q16P0**: a 16-node irreducible, *cubic* displacement parametrically mapped Lagrangian quadrilateral element for mechanical analyses.

- The command syntax for this element type is available in Section 24.5.

Type **Q17P0**: a 17-node irreducible, *quartic* displacement parametrically mapped "serendip-

ity" quadrilateral element for mechanical analyses.

- The command syntax for this element type is available in Section 24.6.

24.1 The ELEMENT IRREDUCIBLE TYPE Q4P0 Command

Synopsis

The **ELEMENT IRREDUCIBLE TYPE Q4P0** command is used to describe all irreducible, 4-node bilinear quadrilateral continuum elements that are to be used in mechanical analyses.

Syntax

The following syntax is used to describe a typical **Q4P0** irreducible quadrilateral continuum element:

ELEment IRReducible TYPe Q4P0 (DEScription "string") NODes #:#:#
(MATerial #) (INItial #)
(CONstruction #) (EXCavation #) (THIckness #.#)
(INTcode #)
(1_Additional #) (1_Increment #)
(2_Additional #) (2_Increment #)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLUMETRIC_STRAIN)

Explanatory Notes

- The **Q4P0** is an irreducible, bilinear, isoparametric Lagrangian continuum element [52]. The element
  - Contains four (4) vertex nodes.
  - Has two (2) displacement degrees of freedom at each node.

– Possesses a total of eight (8) displacement degrees of freedom.

- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.

- The numbering order of **NODES** associated with a **Q4P0** element, which must be specified sequentially from 1 to 4, is shown in Figure 24.1.

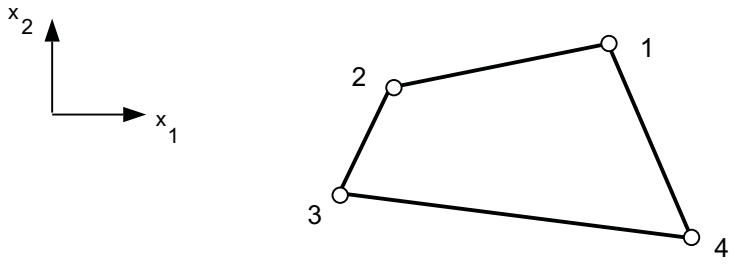

Figure 24.1: Node numbering associated with a typical irreducible, 4-node bilinear (**Q4P0**) quadrilateral continuum element.

- A linear (3-node) triangular elements for mechanical analyses can be developed by *degenerating* a four-node (**Q4P0**) quadrilateral. This is realized by repeating the first node number as the fourth; i.e., for a typical **Q4P0** element, the nodes would be specified as follows:

```
nodes  n1  n2  n3  n1
```

  The results obtained using degenerated quadrilaterals are *identical* to those from true 3-node triangular (**T3P0**) elements [5; 44]. Since the **T3P0** element is implemented in the APES+ computer program (see Section **??**), if use of a 3-node triangular element is desired, it is simper to use the **T3P0** directly.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default values for the **MATERIAL** number is one (1).

- The **INITIAL** keyword is used to specify the initial state number associated with the element. The default value for the **INITIAL** is zero (0).

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from

the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the default condition, no input is required in such a case. The condition of no excavation is likewise the default.

- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The default **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE STRAIN** idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.

- The value specified in conjunction with the **INTCODE** keyword describes the order of numerical integration scheme to be used in developing the element equations for the element.

  The "commonly" used numerical integration rule for **Q4P0** elements corresponds to a 2 by 2 Gauss-Legendre quadrature scheme (degree of precision equal to 3) for the primary dependent variables (i.e., nodal displacements) and a 1-point Gauss-Legendre scheme (degree of precision equal to 1) for the secondary dependent variables (i.e., strains and stresses). This is the default condition and requires no input using the **INTCODE** keyword. If a quadrature order different from the default condition is desired, the following integer values are associated with this keyword:

  INTCODE = 11: A 1-point Gauss-Legendre quadrature scheme (degree of precision equal to 1) is used to compute *both* the primary dependent variables (i.e., nodal displacements) and for the secondary dependent variables (i.e., strains and stresses).

  This serves to uniformly *under integrate* the element. The disadvantage of uniform reduced integration is that the rank of the element stiffness matrix $\mathbf{K}^{(e)}$ may be reduced, resulting in the singularity or near singularity of the global $\mathbf{K}$ [52]. The associated element instabilities manifest themselves in the form of *mechanisms* or *hourglass modes* [44]. Consequently, the specification of **INTCODE** = 11 should only be used to investigate the possible pathologies associated with uniform reduced integration and *not* in standard analyses.

  INTCODE = 21: a 2 by 2 Gauss-Legendre quadrature scheme (degree of precision equal to 3) is used to compute the primary dependent variables (i.e., nodal displacements) and a 1-point Gauss-Legendre scheme (degree of precision equal to 1) is used to compute the secondary dependent variables (i.e., strains and stresses). This is equivalent to the aforementioned default setting.

- If the body being analyzed can be divided into a layer (or layers) of elements, and if the characteristics of the element (i.e., the **MATERIAL**, the **INITIAL** state, the incremental **CONSTRUCTION** and **EXCAVATION** numbers, the **THICKNESS**, and the **INTCODE**) are the *same* for several elements within a layer, and if the nodes are numbered in a *consistent* fashion, then an element data generation option

can be employed. To generate a sequence of elements within a single layer, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**.

For example, assuming that the coordinates of all boundary nodes have been specified, the bottom row of elements in the mesh shown in Figure 24.2 could be established by entering the node numbers associated with element *a* and the keywords/values **1_ADDITIONAL** 4 and **1_INCREMENT** 5. Assuming a **MATERIAL** number equal to one (1) and default values for all of the other element parameters, the input record corresponding to this specification would be:

```
element irreducible type "q4p0" nodes  1  6  7  2  material  1 &
   1_add  4   1_incr 5
```

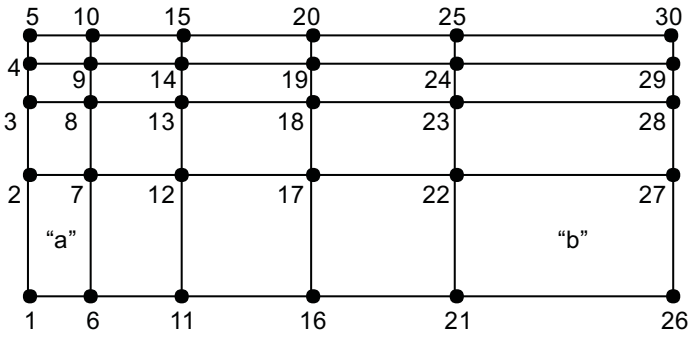

Figure 24.2: Hypothetical mesh consisting of **Q4P0** elements.

Alternately, this bottom row of elements could be established by entering the node numbers associated with element *b* and the keywords/values **1_ADDITIONAL** 4 and **1_INCREMENT** -5 (i.e., a decrement). Assuming once again a **MATERIAL** number equal to one (1) and default values for all of the other element parameters, the input record corresponding to this specification would be as follows:

```
element irreducible type "q4p0" nodes  21  26  27  22 material 1 &
   1_add  4   1_incr -5
```

In a similar manner, the left most column of elements in the mesh could be established by entering the node numbers associated with element *a* and the keywords/values **1_ADDITIONAL** 2 and **1_INCREMENT** 1. The corresponding input record would be as follows:

```
element irreducible type "q4p0" nodes  1  6  7  2  material  1 &
   1_add  2   1_incr 1
```

If several layers of elements have the *same* attributes, the above generation option can be carried one step further. For example, the entire mesh shown in Figure 24.2 could be established by entering the node numbers of element *a* and the keywords/values **1_ADDITIONAL** 4, **1_INCREMENT** 5, **2_ADDITIONAL** 3, and **2_INCREMENT** 1. The associated input record would thus be as follows:

```
element irreducible type "q4p0" nodes  1  6  7  2  material  1 &
   1_add  4   1_incr 5  2_add  3   2_incr 1
```

Note that the 1- and 2-directions need not be orthogonal, nor be aligned with the global coordinate directions. Alternately, the mesh shown in Figure 24.2 could be generated by specifying the node points associated with element *a* and the keywords/values **1_ADDITIONAL** 3, **1_INCREMENT** 1, **2_ADDITIONAL** 4, and **2_INCREMENT** 5. The associated input record would thus be as follows:

```
element irreducible type "q4p0" nodes  1  6  7  2  material  1 &
   1_add  2   1_incr 1  2_add  4   2_incr 5
```

The mesh generated with these two specifications would differ only in the order in which the elements were numbered by the APES program.

- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the keyword **DONT_PRINT_Results** indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the default condition both strains are printed.

- Specification of the **DONT_PRINT_STRESSES** keyword indicates that stresses are not to be printed. Under the default condition stresses are printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_VOLUMETRIC_STRAIN** keyword causes the volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the volumetric strain to the absolute value of the minimum non-zero normal strain in the element is printed. That is,

$$\varepsilon_{ratio} = \frac{\left|\hat{\varepsilon}_{vol}^{(e)}\right|}{\sqrt{\left(\hat{\varepsilon}_{11}^{(e)}\right)^2 + \left(\hat{\varepsilon}_{22}^{(e)}\right)^2 + \left(\hat{\varepsilon}_{33}^{(e)}\right)^2}}$$

  This ratio is instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. As such, this keyword would likely *not* be used in conjunction with the **Q4P0** element. Under the default condition the volumetric strain and the aforementioned ratio are *not* computed and printed.

24.2 The ELEMENT IRREDUCIBLE TYPE Q8P0 Command

Synopsis

The **ELEMENT IRREDUCIBLE TYPE Q8P0** command is used to describe all irreducible, 8-node quadratic quadrilateral continuum elements that are to be used in mechanical analyses.

Syntax

The following syntax is used to describe a typical **Q8P0** irreducible quadrilateral continuum element:

ELEment IRReducible TYPe Q8P0 (DEScription "string")
NODes #:#:# (MATerial #) (INItial #) (THIckness #.#)
(INTcode #) (MAPping_points ##)
(CONstruction #) (EXCavation #)
(1_Additional #) (1_Increment #)
(2_Additional #) (2_Increment #)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_AVG_STRAins) (PRINT_AVG_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLUMETRIC_STRAIN) (PRINT_AVG_VOLumetric_strain)

Explanatory Notes

- The **Q8P0** is an irreducible, quadratic, parametrically mapped "serendipity" continuum element [52]. The element
  - Contains four (4) vertex nodes.
  - Contains four (4) mid-side nodes.
  - Has two (2) displacements degrees of freedom at each node.
  - Possesses a total of sixteen (16) displacement degrees of freedom.
- The numbering order of **NODES** associated with **Q8P0** elements, which must be specified sequentially from 1 to 8, is shown in Figure 24.3.

- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.

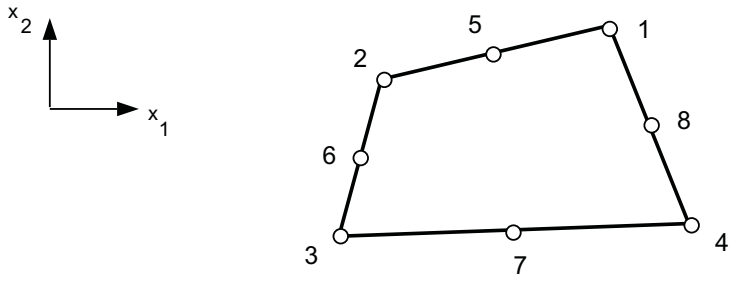

Figure 24.3: Node numbering associated with a typical irreducible, 8-node quadratic "serendipity" (**Q8P0**) quadrilateral continuum element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default values for the **MATERIAL** number is one (1).
- The **INITIAL** keyword is used to specify the initial state number associated with the element. The default value for the **INITIAL** is zero (0).
- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The default **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE STRAIN** idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.
- The value specified in conjunction with the **INTCODE** keyword describes the order of numerical integration scheme to be used in developing the element equations for the element.

  The "commonly" used numerical integration rule for **Q8P0** elements corresponds to a 3 by 3 Gauss-Legendre quadrature scheme (degree of precision equal to 5) for the primary dependent variables (i.e., nodal displacements) and a 2 by 2 Gauss-Legendre scheme (degree of precision equal to 3) for the secondary dependent variables (i.e., strains and stresses). This is the default condition and requires no input using the **INTCODE** keyword. If a quadrature order different from the default condition is desired, the following integer values are associated with this keyword:

  INTCODE = 32: a 3 by 3 Gauss-Legendre quadrature scheme (degree of precision equal to 5) is used to compute the primary dependent variables (i.e., nodal displacements) and a 2 by 2 Gauss-Legendre scheme (degree of precision equal to 3)

is used to compute the secondary dependent variables (i.e., strains and stresses). This is equivalent to the aforementioned default setting.

INTCODE = 21: a 2 by 2 Gauss-Legendre quadrature scheme (degree of precision equal to 3) is used to compute the primary dependent variables (i.e., nodal displacements) and a 1-point Gauss-Legendre scheme (degree of precision equal to 1) is used to compute the secondary dependent variables (i.e., strains and stresses).

- The value specified in conjunction with the **MAPPING_POINTS** keyword describes the parametric element mapping [52] to be used in describing the element geometry. The following integer values are associated with this keyword:

  MAPPING_POINTS = 8: An *isoparametric* mapping, corresponding to a *quadratic* representation of the element geometry, is used. Since this is the default condition, the **MAPPING_POINTS** command can be omitted.

  MAPPING_POINTS = 4: A *sub-parametric* mapping, involving a *linear* representation of the element geometry, is used.

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the default condition, no input is required in such a case. The condition of no excavation is likewise the default.

- If the body being analyzed can be divided into a layer (or layers) of elements, and if the characteristics of the element (i.e., the **MATERIAL**, the **INITIAL** state, the incremental **CONSTRUCTION** and **EXCAVATION** numbers, the **THICKNESS**, and the **INTCODE**) are the *same* for several elements within a layer, and if the nodes are numbered in a *consistent* fashion, then an element data generation option can be employed. To generate a sequence of elements within a single layer, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**.

  If several layers of elements have the *same* attributes, the above generation option can be carried one step further by making use of the **2_ADDITIONAL**, and **2_INCREMENT** keywords.

- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the default condition both strains are printed.

- Specification of the **DONT_PRINT_STRESSES** keyword indicates that stresses are not to be printed. Under the default condition stresses are printed.

- The **PRINT_AVG_STRAINS** keyword indicates that average strains (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the default condition average strains are not computed and printed.

- The **PRINT_AVG_STRESSES** keyword indicates that average stresses (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the default condition average stresses are not computed and printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_VOLUMETRIC_STRAIN** keyword causes the volumetric strain (ε_{vol}) to be computed and printed for the element. In addition, the following quantities are computed and printed:

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33}) \neq 0$$

$$\varepsilon_{ratio} = \frac{\left|\hat{\varepsilon}^{(e)}_{vol}\right|}{\sqrt{\left(\hat{\varepsilon}^{(e)}_{11}\right)^2 + \left(\hat{\varepsilon}^{(e)}_{22}\right)^2 + \left(\hat{\varepsilon}^{(e)}_{33}\right)^2}}$$

  The first quantity is the ratio of the absolute value of ε_{vol} to the absolute value of the minimum non-zero normal strain in the element. The second quantity is the ratio of the absolute value of ε_{vol} to the square root of the sum of squares of the normal strain components. These ratio are instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. As such, this keyword would likely *not* be used in conjunction with the **Q9P0** element. Under the default condition the volumetric strain and the aforementioned ratios are *not* computed and printed.

- The keyword **PRINT_AVG_VOLUMETRIC_STRAIN** causes the *average* (over all the quadrature points used to compute secondary dependent variables) volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the *average* volumetric strain to the absolute value of the minimum *average* non-zero normal strain in the element

$$\varepsilon_{ratio} = \frac{\left|\hat{\varepsilon}_{vol}^{(e)}\right|}{\sqrt{\left(\hat{\varepsilon}_{11}^{(e)}\right)^2 + \left(\hat{\varepsilon}_{22}^{(e)}\right)^2 + \left(\hat{\varepsilon}_{33}^{(e)}\right)^2}}$$

is printed. Under the default condition the average volumetric strain and the aforementioned ratio are *not* computed and printed.

24.3 The ELEMENT IRREDUCIBLE TYPE Q9P0 Command

Synopsis

The **ELEMENT IRREDUCIBLE TYPE Q9P0** command is used to describe all irreducible, 9-node bi-quadratic quadrilateral Lagrangian continuum elements that are to be used in mechanical analyses.

Syntax

The following syntax is used to describe a typical **Q9P0** irreducible quadrilateral continuum element:

ELEment IRReducible TYPe Q9P0 (DEScription "string")
NODes #:#:# (MATerial #) (INItial #) (THIckness #.#)
(INTcode #) (MAPping_points ##)
(CONstruction #) (EXCavation #)
(1_Additional #) (1_Increment #)
(2_Additional #) (2_Increment #)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_AVG_STRAins) (PRINT_AVG_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLUMETRIC_STRAIN) (PRINT_AVG_VOLumetric_strain)

Explanatory Notes

- The **Q9P0** is an irreducible, bi-quadratic, parametrically mapped Lagrangian continuum element [52]. The element
  - Contains four (4) vertex nodes.
  - Contains four (4) mid-side nodes.
  - Contains one (1) interior node.
  - Has two (2) displacements degrees of freedom at each node.
  - Possesses a total of eighteen (18) displacement degrees of freedom.

- The numbering order of **NODES** associated with **Q9P0** elements, which must be specified sequentially from 1 to 9, is shown in Figure 24.4.
- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.

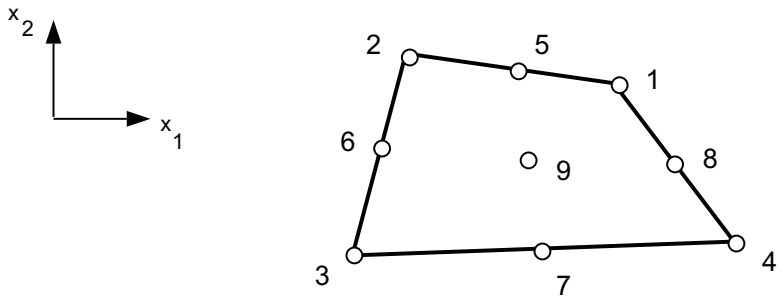

Figure 24.4: Node numbering associated with a typical irreducible, 9-node bi-quadratic Lagrangian (**Q9P0**) quadrilateral continuum element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default values for the **MATERIAL** number is one (1).
- The **INITIAL** keyword is used to specify the initial state number associated with the element. The default value for the **INITIAL** is zero (0).
- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The default **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE STRAIN** idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.
- The value specified in conjunction with the **INTCODE** keyword describes the order of numerical integration scheme to be used in developing the element equations for the element.

  The "commonly" used numerical integration rule for **Q9P0** elements corresponds to a 3 by 3 Gauss-Legendre quadrature scheme (degree of precision equal to 5) for the primary dependent variables (i.e., nodal displacements) and a 2 by 2 Gauss-Legendre scheme (degree of precision equal to 3) for the secondary dependent variables (i.e., strains and stresses). This is the default condition and requires no input using the **INTCODE** keyword. If a quadrature order different from the default condition is desired, the following integer values are associated with this keyword:

INTCODE = 32: a 3 by 3 Gauss-Legendre quadrature scheme (degree of precision equal to 5) is used to compute the primary dependent variables (i.e., nodal displacements) and a 2 by 2 Gauss-Legendre scheme (degree of precision equal to 3) is used to compute the secondary dependent variables (i.e., strains and stresses). This is equivalent to the aforementioned default setting.

INTCODE = 21: a 2 by 2 Gauss-Legendre quadrature scheme (degree of precision equal to 3) is used to compute the primary dependent variables (i.e., nodal displacements) and a 1-point Gauss-Legendre scheme (degree of precision equal to 1) is used to compute the secondary dependent variables (i.e., strains and stresses).

- The value specified in conjunction with the **MAPPING_POINTS** keyword describes the parametric element mapping [52] to be used in describing the element geometry. The following integer values are associated with this keyword:

  MAPPING_POINTS = 9: An *isoparametric* mapping, corresponding to a *bi-quadratic* representation of the element geometry, is used. Since this is the default condition, the **MAPPING_POINTS** command can be omitted.

  MAPPING_POINTS = 4: A *sub-parametric* mapping, involving a *bi-linear* representation of the element geometry, is used.

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the default condition, no input is required in such a case. The condition of no excavation is likewise the default.

- If the body being analyzed can be divided into a layer (or layers) of elements, and if the characteristics of the element (i.e., the **MATERIAL**, the **INITIAL** state, the incremental **CONSTRUCTION** and **EXCAVATION** numbers, the **THICKNESS**, and the **INTCODE**) are the *same* for several elements within a layer, and if the nodes are numbered in a *consistent* fashion, then an element data generation option can be employed. To generate a sequence of elements within a single layer, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**.

  If several layers of elements have the *same* attributes, the above generation option can be carried one step further by making use of the **2_ADDITIONAL**, and **2_INCREMENT** keywords.

- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the default condition both strains are printed.

- Specification of the keyword **DONT_PRINT_STRESSES** indicates that stresses are not to be printed. Under the default condition stresses are printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_AVG_STRAINS** keyword indicates that average strains (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the default condition average strains are not computed and printed.

- The **PRINT_AVG_STRESSES** keyword indicates that average stresses (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the default condition average stresses are not computed and printed.

- The **PRINT_VOLUMETRIC_STRAIN** keyword causes the volumetric strain (ε_{vol}) to be computed and printed for the element. In addition, the following quantities are computed and printed:

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33}) \neq 0$$

$$\varepsilon_{ratio} = \frac{\left|\hat{\varepsilon}^{(e)}_{vol}\right|}{\sqrt{\left(\hat{\varepsilon}^{(e)}_{11}\right)^2 + \left(\hat{\varepsilon}^{(e)}_{22}\right)^2 + \left(\hat{\varepsilon}^{(e)}_{33}\right)^2}}$$

  The first quantity is the ratio of the absolute value of ε_{vol} to the absolute value of the minimum non-zero normal strain in the element. The second quantity is the ratio of the absolute value of ε_{vol} to the square root of the sum of squares of the normal strain components. These ratio are instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. As such, this keyword would likely *not* be used in conjunction with the **Q9P0** element. Under the default condition the volumetric strain and the aforementioned ratios are *not* computed and printed.

- The **PRINT_AVG_VOLUMETRIC_STRAIN** keyword causes the *average* (over all the quadrature points used to compute secondary dependent variables) volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the *average* volumetric strain to the absolute value of the minimum *average* non-zero normal strain in the element

$$\varepsilon_{ratio} = \frac{\left|\hat{\varepsilon}^{(e)}_{vol}\right|}{\sqrt{\left(\hat{\varepsilon}^{(e)}_{11}\right)^2 + \left(\hat{\varepsilon}^{(e)}_{22}\right)^2 + \left(\hat{\varepsilon}^{(e)}_{33}\right)^2}}$$

  is printed. Under the default condition the average volumetric strain and the aforementioned ratio are *not* computed and printed.

24.4 The ELEMENT IRREDUCIBLE TYPE Q12P0 Command

Synopsis

The **ELEMENT IRREDUCIBLE TYPE Q12P0** command is used to describe all irreducible, 12-node cubic "serendipity" quadrilateral continuum elements that are to be used in mechanical analyses.

Syntax

The following syntax is used to describe a typical **Q12P0** irreducible quadrilateral continuum element:

ELEment IRReducible TYPe Q12P0 (DEScription "string")
NODes #:#:# (MATerial #) (INItial #) (THIckness #.#)
(INTcode #) (MAPping_points ##)
(CONstruction #) (EXCavation #)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_AVG_STRAins) (PRINT_AVG_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLUMETRIC_STRAIN) (PRINT_AVG_VOLumetric_strain)

Explanatory Notes

- The **Q12P0** is an irreducible, cubic, parametrically mapped "serendipity" element [52]. The element
  - Contains four (4) vertex nodes.
  - Contains eight (8) mid-side nodes.
  - Has two (2) displacements degrees of freedom at each node.
  - Possesses a total of twenty-four (24) displacement degrees of freedom.
- The numbering order of **NODES** associated with **Q12P0** elements, which must be specified sequentially from 1 to 12, is shown in Figure 24.5.
- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.

  NOTE: Presently APES+ does *not* possess the ability to generate **Q12P0** elements. Two options are, however, available to help in the determination of nodal coordinates.

  First, the analyst can use some stand-alone pre-processing software to accomplish this task. The resulting element and node data must then be translated to the format expected by APES+. When nodal data is supplied to APES+, the ANALYSIS MESH EXTERNAL command must be specified (see Section 5.4).

  Second, if the coordinates of all *vertex* nodes are explicitly input or generated, and if the node numbers associated with all elements are specified using ELEMENT IRREDUCIBLE commands, the LOCATE_coords routines in APES+ will generate the nodal coordinates of all mid-side nodes. This is done assuming that the element has *straight sides*. When nodal data is computed in this manner, the ANALYSIS MESH INTERNAL command must be specified (see Section 5.4).
- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default values for the **MATERIAL** number is one (1).
- The **INITIAL** keyword is used to specify the initial state number associated with the element. The default value for the **INITIAL** is zero (0).
- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The default

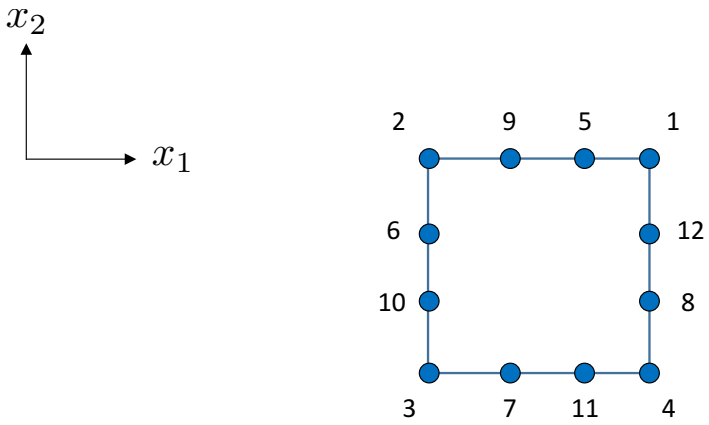

Figure 24.5: Node numbering associated with a typical irreducible, 12-node cubic "serendipity" (**Q12P0**) quadrilateral continuum element.

THICKNESS value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE STRAIN** idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.

- The value specified in conjunction with the **INTCODE** keyword describes the order of numerical integration scheme to be used in developing the element equations for the element.

  The "commonly" used numerical integration rule for **Q12P0** elements corresponds to a 4 by 4 Gauss-Legendre quadrature scheme (degree of precision equal to 7) for the primary dependent variables (i.e., nodal displacements) and a 3 by 3 Gauss-Legendre scheme (degree of precision equal to 5) for the secondary dependent variables (i.e., strains and stresses). This is the default condition and requires no input using the **INTCODE** keyword. If a quadrature order different from the default condition is desired, the following integer values are associated with this keyword:

  INTCODE = 43: a 4 by 4 Gauss-Legendre quadrature scheme (degree of precision equal to 7) is used to compute the primary dependent variables (i.e., nodal displacements) and a 3 by 3 Gauss-Legendre scheme (degree of precision equal to 5) is used to compute the secondary dependent variables (i.e., strains and stresses). This is equivalent to the aforementioned default setting.

  INTCODE = 32: a 3 by 3 Gauss-Legendre quadrature scheme (degree of precision equal to 5) is used to compute the primary dependent variables (i.e., nodal displacements) and a 2 by 2 Gauss-Legendre scheme (degree of precision equal to 3) is used to compute the secondary dependent variables (i.e., strains and stresses).

  INTCODE = 21: a 2 by 2 Gauss-Legendre quadrature scheme (degree of precision equal to 3) is used to compute the primary dependent variables (i.e., nodal dis-

placements) and a 1-point Gauss-Legendre scheme (degree of precision equal to 1) is used to compute the secondary dependent variables (i.e., strains and stresses).

- The value specified in conjunction with the **MAPPING_POINTS** keyword describes the parametric element mapping [52] to be used in describing the element geometry. The following integer values are associated with this keyword:

  MAPPING_POINTS = 12: An *isoparametric* mapping, corresponding to a *cubic* representation of the element geometry, is used. Since this is the default condition, the **MAPPING_POINTS** command can be omitted.

  MAPPING_POINTS = 4: A *sub-parametric* mapping, involving a *linear* representation of the element geometry, is used.

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the default condition, no input is required in such a case. The condition of no excavation is likewise the default.

- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the default condition both strains are printed.

- Specification of the **DONT_PRINT_STRESSES** keyword indicates that stresses are not to be printed. Under the default condition stresses are printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_AVG_STRAINS** keyword indicates that average strains (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the default condition average strains are not computed and printed.

- The **PRINT_AVG_STRESSES** keyword indicates that average stresses (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the default condition average stresses are not computed and printed.

- The **PRINT_VOLUMETRIC_STRAIN** keyword causes the volumetric strain (ε_{vol}) to be computed and printed for the element. In addition, the following quantities are computed and printed:

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33}) \neq 0$$

$$\varepsilon_{ratio} = \frac{\left|\hat{\varepsilon}_{vol}^{(e)}\right|}{\sqrt{\left(\hat{\varepsilon}_{11}^{(e)}\right)^2 + \left(\hat{\varepsilon}_{22}^{(e)}\right)^2 + \left(\hat{\varepsilon}_{33}^{(e)}\right)^2}}$$

  The first quantity is the ratio of the absolute value of ε_{vol} to the absolute value of the minimum non-zero normal strain in the element. The second quantity is the ratio of the absolute value of ε_{vol} to the square root of the sum of squares of the normal strain components. These ratio are instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. As such, this keyword would likely *not* be used in conjunction with the **Q9P0** element. Under the default condition the volumetric strain and the aforementioned ratios are *not* computed and printed.

- The **PRINT_AVG_VOLUMETRIC_STRAIN** keyword causes the *average* (over all the quadrature points used to compute secondary dependent variables) volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the *average* volumetric strain to the absolute value of the minimum *average* non-zero normal strain in the element

$$\varepsilon_{ratio} = \frac{\left|\hat{\varepsilon}_{vol}^{(e)}\right|}{\sqrt{\left(\hat{\varepsilon}_{11}^{(e)}\right)^2 + \left(\hat{\varepsilon}_{22}^{(e)}\right)^2 + \left(\hat{\varepsilon}_{33}^{(e)}\right)^2}}$$

  is printed. Under the default condition the average volumetric strain and the aforementioned ratio are *not* computed and printed.

24.5 The ELEMENT IRREDUCIBLE TYPE Q16P0 Command

Synopsis

The **ELEMENT IRREDUCIBLE TYPE Q16P0** command is used to describe all irreducible, 16-node cubic Lagrangian quadrilateral continuum elements that are to be used in mechanical analyses.

Syntax

The following syntax is used to describe a typical **Q16P0** irreducible quadrilateral continuum element:

ELEment IRReducible TYPe Q16P0 (DEScription "string")
NODes #:#:# (MATerial #) (INItial #) (THIckness #.#)
(INTcode #) (MAPping_points ##)
(CONstruction #) (EXCavation #)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_AVG_STRAins) (PRINT_AVG_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLUMETRIC_STRAIN) (PRINT_AVG_VOLumetric_strain)

Explanatory Notes

- The **Q16P0** is an irreducible, cubic, parametrically mapped Lagrangian element [52]. The element
  - Contains four (4) vertex nodes.
  - Contains eight (8) mid-side nodes.
  - Contains four (4) interior nodes.
  - Has two (2) displacements degrees of freedom at each node.
  - Possesses a total of thirty-two (32) displacement degrees of freedom.
- The numbering order of **NODES** associated with **Q16P0** elements, which must be specified sequentially from 1 to 16, is shown in Figure 24.6.
- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.

  NOTE: Presently APES+ does *not* possess the ability to generate **Q16P0** elements. Two options are, however, available to help in the determination of nodal coordinates.

  First, the analyst can use some stand-alone pre-processing software to accomplish this task. The resulting element and node data must then be translated to the format expected by APES+. When nodal data is supplied to APES+, the ANALYSIS MESH EXTERNAL command must be specified (see Section 5.4).

  Second, if the coordinates of all *vertex* nodes are explicitly input or generated, and if the node numbers associated with all elements are specified using ELEMENT IRREDUCIBLE commands, the LOCATE_coords routines in APES+ will generate the nodal coordinates of all mid-side and interior nodes. This is done assuming that the element has *straight sides*. When nodal data is computed in this manner, the ANALYSIS MESH INTERNAL command must be specified (see Section 5.4).
- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default values for the **MATERIAL** number is one (1).
- The **INITIAL** keyword is used to specify the initial state number associated with the element. The default value for the **INITIAL** is zero (0).

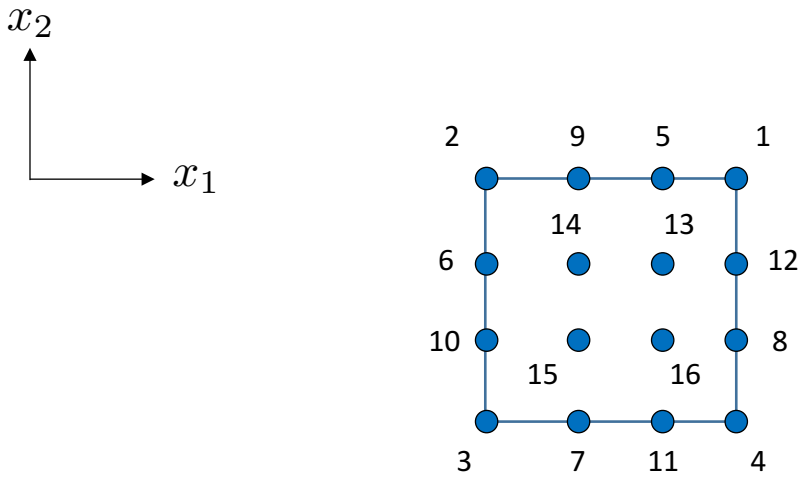

Figure 24.6: Node numbering associated with a typical irreducible, 16-node cubic Lagrangian (**Q16P0**) quadrilateral continuum element.

- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The default **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE STRAIN** idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.

- The value specified in conjunction with the **INTCODE** keyword describes the order of numerical integration scheme to be used in developing the element equations for the element.

  The "commonly" used numerical integration rule for **Q16P0** elements corresponds to a 4 by 4 Gauss-Legendre quadrature scheme (degree of precision equal to 7) for the primary dependent variables (i.e., nodal displacements) and a 3 by 3 Gauss-Legendre scheme (degree of precision equal to 5) for the secondary dependent variables (i.e., strains and stresses). This is the default condition and requires no input using the **INTCODE** keyword. If a quadrature order different from the default condition is desired, the following integer values are associated with this keyword:

  INTCODE = 43: a 4 by 4 Gauss-Legendre quadrature scheme (degree of precision equal to 7) is used to compute the primary dependent variables (i.e., nodal displacements) and a 3 by 3 Gauss-Legendre scheme (degree of precision equal to 5) is used to compute the secondary dependent variables (i.e., strains and stresses). This is equivalent to the aforementioned default setting.

  INTCODE = 32: a 3 by 3 Gauss-Legendre quadrature scheme (degree of precision equal to 5) is used to compute the primary dependent variables (i.e., nodal displacements) and a 2 by 2 Gauss-Legendre scheme (degree of precision equal to 3)

is used to compute the secondary dependent variables (i.e., strains and stresses).

INTCODE = 21: a 2 by 2 Gauss-Legendre quadrature scheme (degree of precision equal to 3) is used to compute the primary dependent variables (i.e., nodal displacements) and a 1-point Gauss-Legendre scheme (degree of precision equal to 1) is used to compute the secondary dependent variables (i.e., strains and stresses).

- The value specified in conjunction with the **MAPPING_POINTS** keyword describes the parametric element mapping [52] to be used in describing the element geometry. The following integer values are associated with this keyword:

  MAPPING_POINTS = 16: An *isoparametric* mapping, corresponding to a *bi-cubic* representation of the element geometry, is used. Since this is the default condition, the **MAPPING_POINTS** command can be omitted.

  MAPPING_POINTS = 4: A *sub-parametric* mapping, involving a *bi-linear* representation of the element geometry, is used.

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the default condition, no input is required in such a case. The condition of no excavation is likewise the default.

- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the default condition both strains are printed.

- Specification of the **DONT_PRINT_STRESSES** keyword indicates that stresses are not to be printed. Under the default condition stresses are printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_AVG_STRAINS** keyword indicates that average strains (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the default condition average strains are not computed and printed.

- The **PRINT_AVG_STRESSES** keyword indicates that average stresses (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the default condition average stresses are not computed and printed.

- The **PRINT_VOLUMETRIC_STRAIN** keyword causes the volumetric strain (ε_{vol}) to be computed and printed for the element. In addition, the following quantities are computed and printed:

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33}) \neq 0$$

$$\varepsilon_{ratio} = \frac{\left|\hat{\varepsilon}^{(e)}_{vol}\right|}{\sqrt{\left(\hat{\varepsilon}^{(e)}_{11}\right)^2 + \left(\hat{\varepsilon}^{(e)}_{22}\right)^2 + \left(\hat{\varepsilon}^{(e)}_{33}\right)^2}}$$

  The first quantity is the ratio of the absolute value of ε_{vol} to the absolute value of the minimum non-zero normal strain in the element. The second quantity is the ratio of the absolute value of ε_{vol} to the square root of the sum of squares of the normal strain components. These ratio are instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. As such, this keyword would likely *not* be used in conjunction with the **Q16P0** element. Under the default condition the volumetric strain and the aforementioned ratios are *not* computed and printed.

- The **PRINT_AVG_VOLUMETRIC_STRAIN** keyword causes the *average* (over all the quadrature points used to compute secondary dependent variables) volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the *average* volumetric strain to the absolute value of the minimum *average* non-zero normal strain in the element

$$\varepsilon_{ratio} = \frac{\left|\hat{\varepsilon}^{(e)}_{vol}\right|}{\sqrt{\left(\hat{\varepsilon}^{(e)}_{11}\right)^2 + \left(\hat{\varepsilon}^{(e)}_{22}\right)^2 + \left(\hat{\varepsilon}^{(e)}_{33}\right)^2}}$$

  is printed. Under the default condition the average volumetric strain and the aforementioned ratio are *not* computed and printed.

24.6 The ELEMENT IRREDUCIBLE TYPE Q17P0 Command

Synopsis

The **ELEMENT IRREDUCIBLE TYPE Q17P0** command is used to describe all irreducible, 17-node quartic "serendipity" quadrilateral continuum elements that are to be used in mechanical analyses.

Syntax

The following syntax is used to describe a typical **Q17P0** irreducible quadrilateral continuum element:

ELEment IRReducible TYPe Q17P0 (DEScription "string")
NODes #:#:# (MATerial #) (INItial #) (THIckness #.#)
(INTcode #) (MAPping_points ##)
(CONstruction #) (EXCavation #)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_AVG_STRAins) (PRINT_AVG_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLUMETRIC_STRAIN) (PRINT_AVG_VOLumetric_strain)

Explanatory Notes

- The **Q17P0** is an irreducible, quartic, parametrically mapped "serendipity" element [52]. The element
  - Contains four (4) vertex nodes.
  - Contains twelve (12) mid-side nodes.
  - Contains one (1) interior node.
  - Has two (2) displacements degrees of freedom at each node.
  - Possesses a total of thirty-four (34) displacement degrees of freedom.
- The numbering order of **NODES** associated with **Q17P0** elements, which must be specified sequentially from 1 to 17, is shown in Figure 24.7.
- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.

  NOTE: Presently APES+ does *not* possess the ability to generate **Q17P0** elements. Two options are, however, available to help in the determination of nodal coordinates.

  First, the analyst can use some stand-alone pre-processing software to accomplish this task. The resulting element and node data must then be translated to the format expected by APES+. When nodal data is supplied to APES+, the ANALYSIS MESH EXTERNAL command must be specified (see Section 5.4).

  Second, if the coordinates of all *vertex* nodes are explicitly input or generated, and if the node numbers associated with all elements are specified using ELEMENT IRREDUCIBLE commands, the LOCATE_coords routines in APES+ will generate the nodal coordinates of all mid-side nodes as well as the interior node. This is done assuming that the element has *straight sides*. When nodal data is computed in this manner, the ANALYSIS MESH INTERNAL command must be specified (see Section 5.4).
- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default values for the **MATERIAL** number is one (1).
- The **INITIAL** keyword is used to specify the initial state number associated with the element. The default value for the **INITIAL** is zero (0).

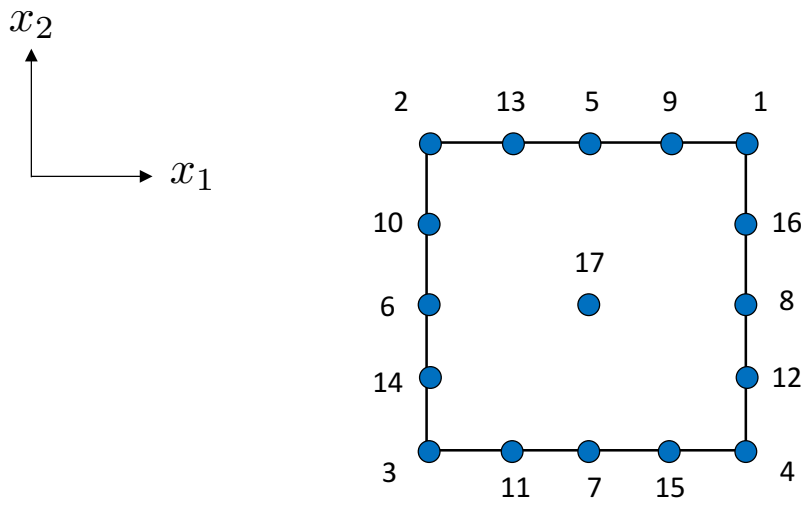

Figure 24.7: Node numbering associated with a typical irreducible, 17-node quartic "serendipity" (**Q17P0**) quadrilateral continuum element.

- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The default **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE STRAIN** idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.

- The value specified in conjunction with the **INTCODE** keyword describes the order of numerical integration scheme to be used in developing the element equations for the element.

  The "commonly" used numerical integration rule for **Q17P0** elements corresponds to a 5 by 5 Gauss-Legendre quadrature scheme (degree of precision equal to 9) for the primary dependent variables (i.e., nodal displacements) and a 4 by 4 Gauss-Legendre scheme (degree of precision equal to 5) for the secondary dependent variables (i.e., strains and stresses). This is the default condition and requires no input using the **INTCODE** keyword. If a quadrature order different from the default condition is desired, the following integer values are associated with this keyword:

  INTCODE = 54: a 5 by 5 Gauss-Legendre quadrature scheme (degree of precision equal to 7) is used to compute the primary dependent variables (i.e., nodal displacements) and a 4 by 4 Gauss-Legendre scheme (degree of precision equal to 5) is used to compute the secondary dependent variables (i.e., strains and stresses). This is equivalent to the aforementioned default setting.

  INTCODE = 43: a 4 by 4 Gauss-Legendre quadrature scheme (degree of precision equal to 7) is used to compute the primary dependent variables (i.e., nodal dis-

placements) and a 3 by 3 Gauss-Legendre scheme (degree of precision equal to 5) is used to compute the secondary dependent variables (i.e., strains and stresses).

INTCODE = 32: a 3 by 3 Gauss-Legendre quadrature scheme (degree of precision equal to 5) is used to compute the primary dependent variables (i.e., nodal displacements) and a 2 by 2 Gauss-Legendre scheme (degree of precision equal to 3) is used to compute the secondary dependent variables (i.e., strains and stresses).

INTCODE = 21: a 2 by 2 Gauss-Legendre quadrature scheme (degree of precision equal to 3) is used to compute the primary dependent variables (i.e., nodal displacements) and a 1-point Gauss-Legendre scheme (degree of precision equal to 1) is used to compute the secondary dependent variables (i.e., strains and stresses).

- The value specified in conjunction with the **MAPPING_POINTS** keyword describes the parametric element mapping [52] to be used in describing the element geometry. The following integer values are associated with this keyword:

  MAPPING_POINTS = 17: An *isoparametric* mapping, corresponding to a *quartic* representation of the element geometry, is used. Since this is the default condition, the **MAPPING_POINTS** command can be omitted.

  MAPPING_POINTS = 4: A *sub-parametric* mapping, involving a *bi-linear* representation of the element geometry, is used.

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the default condition, no input is required in such a case. The condition of no excavation is likewise the default.

- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the default condition both strains are printed.

- Specification of the **DONT_PRINT_STRESSES** keyword indicates that stresses are not to be printed. Under the default condition stresses are printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_AVG_STRAINS** keyword indicates that average strains (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the default condition average strains are not computed and printed.

- The **PRINT_AVG_STRESSES** keyword indicates that average stresses (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the default condition average stresses are not computed and printed.

- The **PRINT_VOLUMETRIC_STRAIN** keyword causes the volumetric strain (ε_{vol}) to be computed and printed for the element. In addition, the following quantities are computed and printed:

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33}) \neq 0$$

$$\varepsilon_{ratio} = \frac{\left|\hat{\varepsilon}^{(e)}_{vol}\right|}{\sqrt{\left(\hat{\varepsilon}^{(e)}_{11}\right)^2 + \left(\hat{\varepsilon}^{(e)}_{22}\right)^2 + \left(\hat{\varepsilon}^{(e)}_{33}\right)^2}}$$

  The first quantity is the ratio of the absolute value of ε_{vol} to the absolute value of the minimum non-zero normal strain in the element. The second quantity is the ratio of the absolute value of ε_{vol} to the square root of the sum of squares of the normal strain components. These ratio are instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. As such, this keyword would likely *not* be used in conjunction with the **Q17P0** element. Under the default condition the volumetric strain and the aforementioned ratios are *not* computed and printed.

- The **PRINT_AVG_VOLUMETRIC_STRAIN** keyword causes the *average* (over all the quadrature points used to compute secondary dependent variables) volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the *average* volumetric strain to the absolute value of the minimum *average* non-zero normal strain in the element

$$\varepsilon_{ratio} = \frac{\left|\hat{\varepsilon}^{(e)}_{vol}\right|}{\sqrt{\left(\hat{\varepsilon}^{(e)}_{11}\right)^2 + \left(\hat{\varepsilon}^{(e)}_{22}\right)^2 + \left(\hat{\varepsilon}^{(e)}_{33}\right)^2}}$$

is printed. Under the default condition the average volumetric strain and the aforementioned ratio are *not* computed and printed.

Chapter 25

ELEMENT IRREDUCIBLE
Commands Involving Tetrahedrals

The available **TETRAHEDRAL** irreducible elements are listed below. These elements are entered using the **ELEMENT IRREDUCIBLE** commands.

Type **TET4P0**: a irreducible, 4-node *linear* displacement isoparametric tetrahedral element.

- The command syntax for this element type is available in Section 25.1.

Type **TET10P0**: a 4-node irreducible, *linear* displacement parametrically mapped tetrahedral element.

- The command syntax for this element type is available in Section 25.2.

25.1 The ELEMENT IRREDUCIBLE TYPE TET4P0 Command

Synopsis

The **ELEMENT IRREDUCIBLE TYPE TET4P0** command is used to describe all irreducible, 4-node liner tetrahedral continuum elements that are to be used in mechanical analyses.

Syntax

The following syntax is used to describe a typical **TET4P0** irreducible quadrilateral continuum element:

ELEment IRReducible TYPe TET4P0 (DEScription "string")
NODes #:#:# (MATerial #) (INItial #)
(CONstruction #) (EXCavation #)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)

Explanatory Notes

- The **TET4P0** is an irreducible, linear, isoparametric tetrahedral element. It possesses twelve (12) displacement degrees of freedom.
- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with **TET4P0** elements, which should proceed in the sequence 1 to 4, is shown in Figure 25.1.

  NOTE: Presently APES+ does *not* possess the ability to generate **TET4P0** elements. It is assumed that the analyst will use some stand alone pre-processing software to accomplish this task. The resulting element and node data will then be translated to the format expected by APES+.

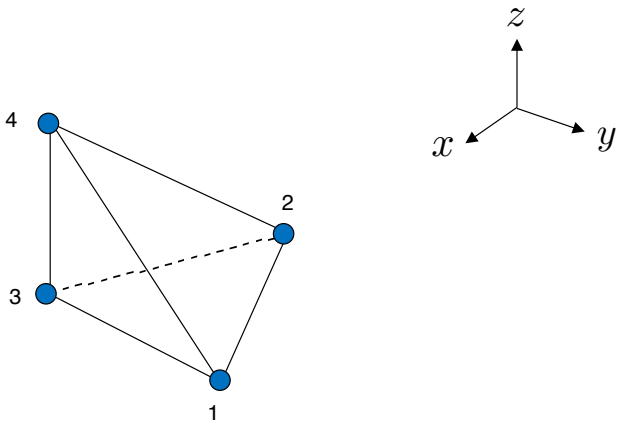

Figure 25.1: Node Numbering associated with a typical irreducible, 4-node linear (**TET4P0**) tetrahedral continuum element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default values for the **MATERIAL** number is one (1).
- The **INITIAL** keyword is used to specify the initial state number associated with the element. The default value for the **INITIAL** is zero (0).
- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the default condition, no input is required in such a case. The condition of no excavation is likewise the default.
- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step.
- Specification of the keyword **DONT_PRINT_Results** indicates that the analyst does not desire to see output (strains and stresses) for this element. Specification of the **DONT_PRINT_STRAINS** keyword indicates that only stresses are to be printed. Similarly, specification of the keyword **DONT_PRINT_STRESSES** indicates that only strains are to be printed. Under the default condition, both strains and stresses are printed.
- The **PRINT_PRIN_STRAINS** and **PRINT_PRIN_STRESSES** keywords indicate that principal strains and stresses, respectively are to be printed. Under the default condition, neither of these quantities are printed.

25.2 The ELEMENT IRREDUCIBLE TYPE TET10P0 Command

Synopsis

The **ELEMENT IRREDUCIBLE TYPE TET10P0** command is used to describe all irreducible, 10-node quadratic tetrahedral continuum elements that are to be used in mechanical analyses.

Syntax

The following syntax is used to describe a typical **TET10P0** irreducible quadrilateral continuum element:

ELEment IRReducible TYPe TET10P0 (DEScription "string")
NODes #:#:# (MATerial #) (INItial #) (MAPping_points ##)
(CONstruction #) (EXCavation #)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)

Explanatory Notes

- The **TET10P0** is an irreducible, quadratic, parametrically mapped tetrahedral element. It possesses thirty (30) displacement degrees of freedom.
- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with **TET10P0** elements, which should proceed in the sequence 1 to 10, is shown in Figure 25.2.

  NOTE: Presently APES+ does *not* possess the ability to generate **TET10P0** elements. It is assumed that the analyst will use some stand alone pre-processing software to accomplish this task. The resulting element and node data will then be translated to the format expected by APES+.

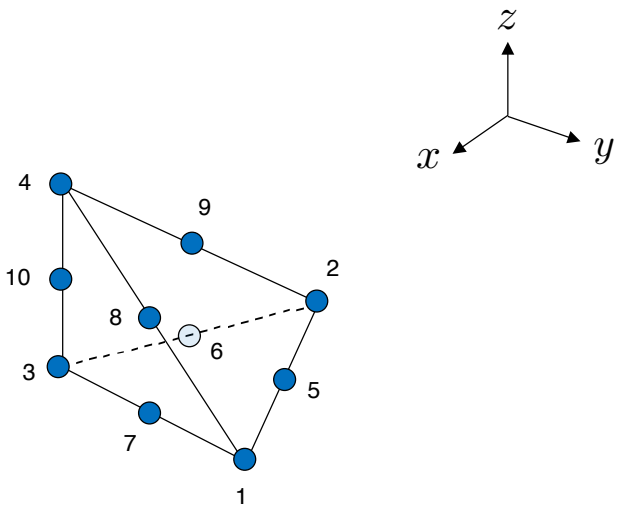

Figure 25.2: Node numbering associated with a typical irreducible, 10-node quadratic (**TET10P0**) tetrahedral continuum element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default values for the **MATERIAL** number is one (1).
- The **INITIAL** keyword is used to specify the initial state number associated with the element. The default value for the **INITIAL** is zero (0).
- The value specified in conjunction with the **MAPPING_POINTS** keyword describes the parametric element mapping [52] to be used in describing the element geometry. The following integer values are associated with this keyword:

  MAPPING_POINTS = 10: An *isoparametric* mapping, corresponding to a *quadratic* representation of the element geometry, is used. Since this is the default condition, the **MAPPING_POINTS** command can be omitted.

  MAPPING_POINTS = 4: A *sub-parametric* mapping, involving a *linear* representation of the element geometry, is used.

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the default condition, no input is required in such a case. The condition of no excavation is likewise the default.
- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step.

- Specification of the keyword **DONT_PRINT_Results** indicates that the analyst does not desire to see output (strains and stresses) for this element. Specification of the **DONT_PRINT_STRAINS** keyword indicates that only stresses are to be printed. Similarly, specification of the keyword **DONT_PRINT_STRESSES** indicates that only strains are to be printed. Under the default condition, both strains and stresses are printed.

- The **PRINT_PRIN_STRAINS** and **PRINT_PRIN_STRESSES** keywords indicate that principal strains and stresses, respectively are to be printed. Under the default condition, neither of these quantities are printed.

Chapter 26

ELEMENT IRREDUCIBLE
Commands Involving Hexahedrals

The available **HEXAHEDRAL** irreducible elements are listed below. These elements are entered using the **ELEMENT IRREDUCIBLE** commands.

Type **H8P0**: an irreducible, 8-node *trilinear* displacement, isoparametric Lagrangian hexahedral element.

- The command syntax for this element type is available in Section 26.1.

Type **H20P0**: an irreducible, 20-node *quadratic* displacement, parametrically mapped "serendipity" hexahedral element.

- The command syntax for this element type is available in Section 26.2.

Type **H27P0**: an irreducible, 27-node *triquadratic* displacement, parametrically Lagrangian hexahedral element.

- The command syntax for this element type is available in Section 26.3.

26.1 The ELEMENT IRREDUCIBLE TYPE H8P0 Command

Synopsis

The **ELEMENT IRREDUCIBLE TYPE H8P0** command is used to describe all irreducible 8-node trilinear hexahedral continuum elements that are to be used in mechanical analyses.

Syntax

The following syntax is used to describe a typical **H8P0** irreducible hexahedral continuum element:

ELEment IRReducible TYPe H8P0 (DEScription "string")
NODes #:#:# (MATerial #) (INItial #) (INTcode #)
(CONstruction #) (EXCavation #)
(1_Additional #) (1_Increment #)
(2_Additional #) (2_Increment #)
(3_Additional #) (3_Increment #)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLumetric_strain)

Explanatory Notes

- The **H8P0** is an irreducible, trilinear, isoparametric hexahedral continuum element [52]. The element
  - Contains eight (8) vertex nodes.
  - Has three (3) displacements degrees of freedom at each node.
  - Possesses a total of twenty-four (24) displacement degrees of freedom.
- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.

- The numbering order of **NODES** associated with **H8P0** elements, which must be specified sequentially from 1 to 8, is shown in Figure 26.1.

  NOTE: Presently APES+ does *not* possess the ability to automatically generate all nodes in **H8P0** elements. It is assumed that the analyst will thus use some stand-alone pre-processing software to accomplish this task. The resulting element and node data will then be translated to the format expected by APES+.

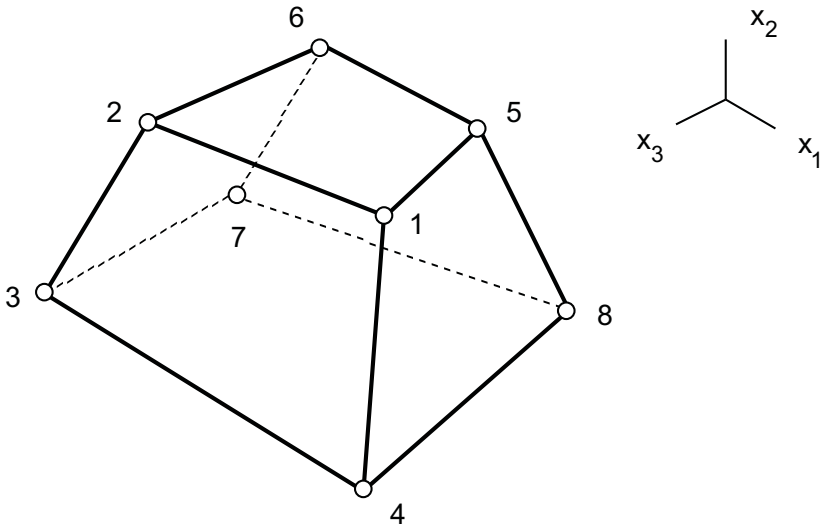

Figure 26.1: Node numbering associated with a typical irreducible, 8-node hexahedral trilinear (**H8P0**) continuum element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default values for the **MATERIAL** number is one (1).
- The **INITIAL** keyword is used to specify the initial state number associated with the element. The default value for the **INITIAL** is zero (0).
- The value specified in conjunction with the **INTCODE** keyword describes the order of numerical integration scheme to be used in developing the element equations for the element.

  The "commonly" used numerical integration rule for **H8P0** elements corresponds to a 2 by 2 by 2 Gauss-Legendre quadrature scheme (degree of precision equal to 3) for the primary dependent variables (i.e., nodal displacements) and a 1-point Gauss-Legendre scheme (degree of precision equal to 1) for the secondary dependent variables (i.e., strains and stresses). This is the default condition and requires no input using the **INTCODE** keyword. If a quadrature order different from the default condition is desired, the following integer values are associated with this keyword:

  INTCODE = 11: A 1-point Gauss-Legendre quadrature scheme (degree of precision equal to 1) is used to compute *both* the primary dependent variables (i.e.,

nodal displacements) and for the secondary dependent variables (i.e., strains and stresses).

This serves to uniformly *under integrate* the element. The disadvantage of uniform reduced integration is that the rank of the element stiffness matrix $\mathbf{K}^{(e)}$ may be reduced, resulting in the singularity or near singularity of the global $\mathbf{K}$ [52]. The associated element instabilities manifest themselves in the form of *mechanisms* or *hourglass modes* [44]. Consequently, the specification of **INTCODE** = 11 should only be used to investigate the possible pathologies associated with uniform reduced integration and *not* in standard analyses.

INTCODE = 21: a 2 by 2 by 2 Gauss-Legendre quadrature scheme (degree of precision equal to 3) is used to compute the primary dependent variables (i.e., nodal displacements) and a 1-point Gauss-Legendre scheme (degree of precision equal to 1) is used to compute the secondary dependent variables (i.e., strains and stresses). This is equivalent to the aforementioned default setting.

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the default condition, no input is required in such a case. The condition of no excavation is likewise the default.

- If the body being analyzed can be divided into a layer (or layers) of elements, and if the characteristics of the element (i.e., the **MATERIAL**, the **INITIAL** state, the incremental **CONSTRUCTION** and **EXCAVATION** numbers, the **THICKNESS**, and the **INTCODE**) are the *same* for several elements within a layer, and if the nodes are numbered in a *consistent* fashion, then an element data generation option can be employed. To generate a sequence of elements within a single layer, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**.

  If several layers of elements have the *same* attributes, the above generation option can be carried one step further by making use of the **2_ADDITIONAL**, and **2_INCREMENT** keywords, and possibly the **3_ADDITIONAL**, and **3_INCREMENT** keywords.

- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the default condition both strains are printed.

- Specification of the **DONT_PRINT_STRESSES** keyword indicates that stresses are not to be printed. Under the default condition stresses are printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_VOLUMETRIC_STRAIN** keyword causes the volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the volumetric strain to the absolute value of the minimum non-zero normal strain in the element is printed. That is,

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33}) \neq 0$$

  This ratio is instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. As such, this keyword would likely *not* be used in conjunction with the **H8P0** element. Under the default condition the volumetric strain and the aforementioned ratio are *not* computed and printed.

26.2 The ELEMENT IRREDUCIBLE TYPE H20P0 Command

Synopsis

The **ELEMENT IRREDUCIBLE TYPE H20P0** command is used to describe all irreducible 20-node quadratic, hexahedral continuum elements that are to be used in mechanical analyses.

Syntax

The following syntax is used to describe a typical **H20P0** irreducible hexahedral continuum element:

ELEment IRReducible TYPe H20P0 (DEScription "string")
NODes #:#:# (MATerial #) (INItial #) (INTcode #) (MAPping_points ##)
(CONstruction #) (EXCavation #)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_AVG_STRAins) (PRINT_AVG_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLUMETRIC_STRAIN) (PRINT_AVG_VOLumetric_strain)

Explanatory Notes

- The **H20P0** is an irreducible, quadratic, parametrically mapped "serendipity" hexahedral continuum element [52]. The element
  - Has six (6) faces.
  - Contains eight (8) vertex nodes.
  - Contains twelve (12) mid-side nodes.
  - Has three (3) displacements degrees of freedom at each node.
  - Possesses a total of sixty (60) displacement degrees of freedom.
- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.

- The numbering order of **NODES** associated with **H20P0** elements, which must be specified sequentially from 1 to 20, is shown in Figure 26.2.

  NOTE: Presently APES+ does *not* possess the ability to generate **H20P0** elements. It is assumed that the analyst will thus use some stand-alone pre-processing software to accomplish this task. The resulting element and node data will then be translated to the format expected by APES+.

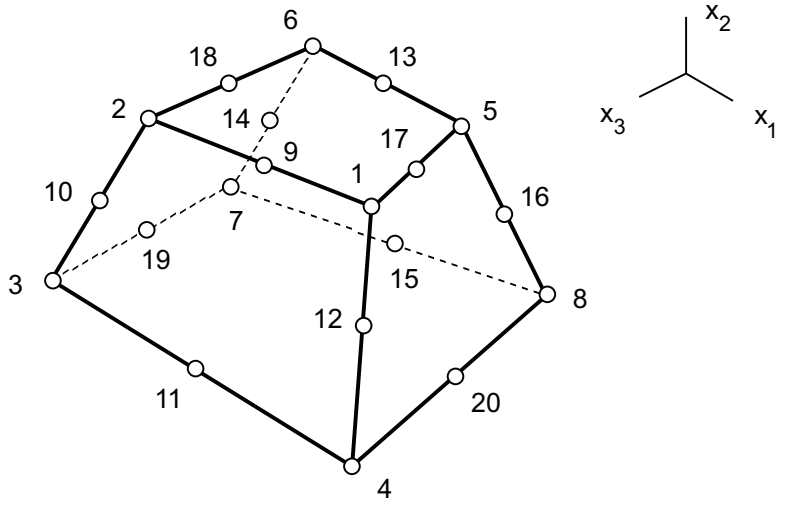

Figure 26.2: Node numbering associated with a typical irreducible, 20-node quadratic (**H20P0**) hexahedral continuum element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default values for the **MATERIAL** number is one (1).

- The **INITIAL** keyword is used to specify the initial state number associated with the element. The default value for the **INITIAL** is zero (0).

- The value specified in conjunction with the **INTCODE** keyword describes the order of numerical integration scheme to be used in developing the element equations for the element.

  The "commonly" used numerical integration rule for **H20P0** elements corresponds to a 3 by 3 by 3 Gauss-Legendre quadrature scheme (degree of precision equal to 5) for the primary dependent variables (i.e., nodal displacements) and a 2 by 2 by 2 Gauss-Legendre scheme (degree of precision equal to 3) for the secondary dependent variables (i.e., strains and stresses). This is the default condition and requires no input using the **INTCODE** keyword.

  If a quadrature order different from the default condition is desired, the following integer values are associated with this keyword:

INTCODE = 32: a 3 by 3 by 3 Gauss-Legendre quadrature scheme (degree of precision equal to 5) is used to compute the primary dependent variables (i.e., nodal displacements) and a 2 by 2 by 2 Gauss-Legendre scheme (degree of precision equal to 3) is used to compute the secondary dependent variables (i.e., strains and stresses). This is equivalent to the aforementioned default setting.

INTCODE = 21: A 2 by 2 by 2 Gauss-Legendre quadrature scheme (degree of precision equal to 3) is used for the primary dependent variables (i.e., nodal displacements) and a 1-point Gauss-Legendre scheme for the secondary dependent variables (i.e., strains and stresses).

- The value specified in conjunction with the **MAPPING_POINTS** keyword describes the parametric element mapping [52] to be used in describing the element geometry. The following integer values are associated with this keyword:

  MAPPING_POINTS = 20: An *isoparametric* mapping, corresponding to a *tri-quadratic* representation of the element geometry, is used. Since this is the default condition, the **MAPPING_POINTS** command can be omitted.

  MAPPING_POINTS = 8: A *sub-parametric* mapping, involving a *tri-linear* representation of the element geometry, is used.

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the default condition, no input is required in such a case. The condition of no excavation is likewise the default.

- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the default condition both strains are printed.

- Specification of the **DONT_PRINT_STRESSES** keyword indicates that stresses are not to be printed. Under the default condition stresses are printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_AVG_STRAINS** keyword indicates that average strains (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the default condition average strains are not computed and printed.

- The **PRINT_AVG_STRESSES** keyword indicates that average stresses (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the default condition average stresses are not computed and printed.

- The **PRINT_VOLUMETRIC_STRAIN** keyword causes the volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the volumetric strain to the absolute value of the minimum non-zero normal strain in the element is printed. That is,

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33}) \neq 0$$

  This ratio is instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. As such, this keyword would likely *not* be used in conjunction with the **H20P0** element. Under the default condition the volumetric strain and the aforementioned ratio are *not* computed and printed.

26.3 The ELEMENT IRREDUCIBLE TYPE H27P0 Command

Synopsis

The **ELEMENT IRREDUCIBLE TYPE H27P0** command is used to describe all irreducible, 27-node quadratic, hexahedral continuum elements that are to be used in mechanical analyses.

Syntax

The following syntax is used to describe a typical **H27P0** irreducible hexahedral continuum element:

ELEment IRReducible TYPe H27P0 (DEScription "string")
NODes #:#:# (MATerial #) (INItial #) (INTcode #) (MAPping_points ##)
(CONstruction #) (EXCavation #)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_AVG_STRAins) (PRINT_AVG_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLUMETRIC_STRAIN) (PRINT_AVG_VOLumetric_strain)

Explanatory Notes

- The **H27P0** is an irreducible, triquadratic, parametrically mapped Lagrangian hexahedral continuum element [52]. The element
  - Has six (6) faces.
  - Contains eight (8) vertex nodes.
  - Contains twelve (12) mid-side nodes.
  - Contains seven (7) interior nodes.
  - Has three (3) displacements degrees of freedom at each node.
  - Possesses a total of eighty-one (81) displacement degrees of freedom.

- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.

- The numbering order of **NODES** associated with the **H27P0** element is shown in Figure 26.3 (see also Figure 26.4). For this element the numbers 1 to 27 must be input sequentially.

  NOTE: Presently APES+ does *not* possess the ability to generate **H27P0** elements. It is assumed that the analyst will use some stand alone pre-processing software to accomplish this task. The resulting element and node data will then be translated to the format expected by APES+.

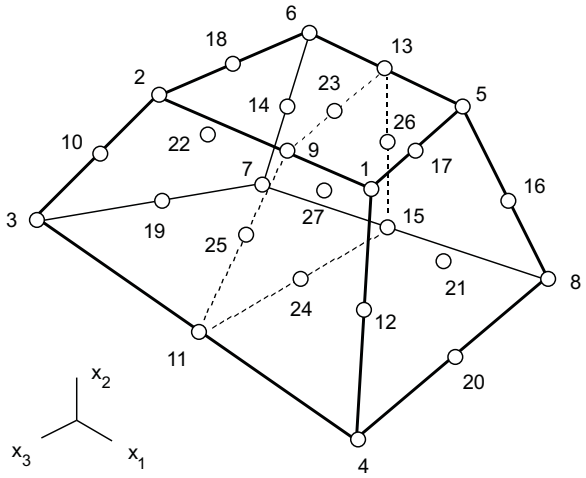

Figure 26.3: Node numbering associated with a typical irreducible, 27-node triquadratic (**H27P0**) hexahedral continuum element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default values for the **MATERIAL** number is one (1).

- The **INITIAL** keyword is used to specify the initial state number associated with the element. The default value for the **INITIAL** is zero (0).

- The value specified in conjunction with the **INTCODE** keyword describes the order of numerical integration scheme to be used in developing the element equations for the element.

  The "commonly" used numerical integration rule for *H27P0* elements corresponds to a 3 by 3 by 3 Gauss-Legendre quadrature scheme (degree of precision equal to 5) for

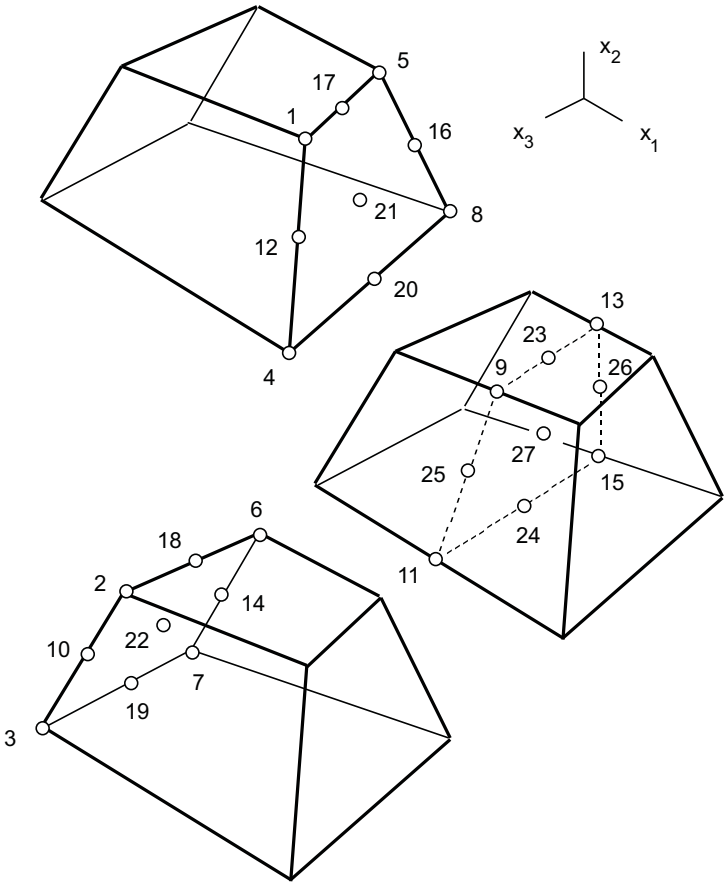

Figure 26.4: Exploded view of node numbering associated with a typical irreducible, 27-node triquadratic (**H27P0**) hexahedral continuum element.

the primary dependent variables (i.e., nodal displacements) and a 2 by 2 by 2 Gauss-Legendre scheme (degree of precision equal to 3) for the secondary dependent variables (i.e., strains and stresses). This is the default condition and requires no input using the **INTCODE** keyword.

If a quadrature order different from the default condition is desired, the following integer values are associated with this keyword:

INTCODE = 32: a 3 by 3 by 3 Gauss-Legendre quadrature scheme (degree of precision equal to 5) is used to compute the primary dependent variables (i.e., nodal displacements) and a 2 by 2 by 2 Gauss-Legendre scheme (degree of precision equal to 3) is used to compute the secondary dependent variables (i.e., strains and stresses). This is equivalent to the aforementioned default setting.

INTCODE = 21: A 2 by 2 by 2 Gauss-Legendre quadrature scheme (degree of precision equal to 3) is used for the primary dependent variables (i.e., nodal displacements) and a 1-point Gauss-Legendre scheme for the secondary dependent variables (i.e., strains and stresses).

- The value specified in conjunction with the **MAPPING_POINTS** keyword describes the parametric element mapping [52] to be used in describing the element geometry. The following integer values are associated with this keyword:

  MAPPING_POINTS = 27: An *isoparametric* mapping, corresponding to a *tri-quadratic* representation of the element geometry, is used. Since this is the default condition, the **MAPPING_POINTS** command can be omitted.

  MAPPING_POINTS = 8: A *sub-parametric* mapping, involving a *tri-linear* representation of the element geometry, is used.

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the default condition, no input is required in such a case. The condition of no excavation is likewise the default.

- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the default condition both strains are printed.

- Specification of the **DONT_PRINT_STRESSES** keyword indicates that stresses are not to be printed. Under the default condition stresses are printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_AVG_STRAINS** keyword indicates that average strains (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the default condition average strains are not computed and printed.

- The **PRINT_AVG_STRESSES** keyword indicates that average stresses (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the default condition average stresses are not computed and printed.

- The **PRINT_VOLUMETRIC_STRAIN** keyword causes the volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the volumetric strain to the absolute value of the minimum non-zero normal strain in the element is printed. That is,

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33}) \neq 0$$

  This ratio is instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. As such, this keyword would likely *not* be used in conjunction with the **H27P0** element. Under the default condition the volumetric strain and the aforementioned ratio are *not* computed and printed.

Chapter 27

ELEMENT MIXED Commands

The available *mixed* continuum elements that account for the flow of pore fluid for mechanical analyses are listed below. In these elements, the pressure approximation is *discontinuous*. Such elements are entered using the **ELEMENT MIXED** commands.

Type **T6P1d**: a 6-node mixed or mixed-penalty isoparametric triangular element. The approximation for displacement is quadratic; a constant (discontinuous) pressure is assumed.

- The command syntax is available in Section 27.1.

Type **T7P3d**: a 7-node mixed or mixed-penalty isoparametric triangular element. The approximation for displacement is quadratic with an additional "bubble" node. A constant (discontinuous) pressure approximation is assumed.

- The command syntax is available in Section 27.2.

Type **Q4P1d**: a 4-node mixed or mixed-penalty isoparametric quadrilateral element. The approximation for displacement is bilinear in displacement; a constant (discontinuous) pressure is assumed.

- The command syntax is available in Section 27.3.

Type **Q8P3d**: an 8-node mixed or mixed-penalty isoparametric quadrilateral element. The approximation for displacement is quadratic, of "serendipity" type; a linear (discontinuous) pressure is assumed.

- The command syntax is available in Section 27.4.

Type **Q9P3d**: a 9-node mixed or mixed-penalty isoparametric quadrilateral element. The approximation for displacement is quadratic, of Lagrangian type; a linear (discontinuous) pressure is assumed.

- The command syntax is available in Section 27.5.

Type **H8P1d**: an 8-node mixed or mixed-penalty isoparametric hexahedral element. The approximation for displacement is trilinear in displacement; a constant (discontinuous) pressure is assumed.

- Thecommand syntax is available in Section 27.6.

27.1 The ELEMENT MIXED TYPE T6P1d Command

Synopsis

The **ELEMENT MIXED TYPE T6P1d** command is used to describe all mixed, 6-node quadratic triangular continuum elements with a constant *discontinuous* pressure approximation.

Remarks

- The **T6P1d** is a mixed, quadratic, isoparametric triangular continuum element. The element
  - Contains three (3) vertex nodes.
  - Contains three (3) mid-side nodes.
  - Has two (2) displacements degrees of freedom at each node, for a total of twelve (12) displacement degrees of freedom.
  - Employs a *quadratic* approximation for the displacement field.
  - Employs a *constant*, discontinuous approximation for the pressure (Figure 27.1). The pressure degrees of freedom are "condensed" out [52] from the element prior to its assembly into the global system of equations.
- Since mixed or mixed-penalty continuum elements are typically used in simulating incompressible or nearly incompressible material response, **T6P1d** elements should be either *plane strain* or *torsionless axisymmetry* anaylses. No benefits are realized by using **T6P1d** elements for *plane stress* idealizations. Indeed, more accurate results are obtained by using the **T6P0** element, which is the irreducible counterpart to the **T6P1d** element.
- For analyses involving standard (compressible) materials no benefit is gained from using **T6P1d** elements. In such cases the irreducible **T6P0** element is preferred.
- The **T6P1d** element satisfies the Babuška-Brezzi condition [44]. The accuracy is $O(h)$ (or the rate of convergence is 1). As such, this is a *sub-optimal* element.
- Because of the constant pressure approximation, the **T6P1d** element simulates incompressibility *in the mean*. Consequently, accurate pressure approximations may require rather fine discretizations [5].

Syntax

The following syntax is used to describe a typical mixed or mixed-penalty **T6P1d** continuum element:

ELEment MIXed TYPe [T6P1d] (DEScription "string")
NODes #:#:# (MATerial #) (INItial #)
(CONstruction #) (EXCavation #) (THIckness #.#) (PENalty)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLumetric_strain)

Explanatory Notes

- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.

- The numbering order of **NODES** associated with **T6P1d** elements, which must be specified sequentially from 1 to 6, is shown in Figure 27.1.

  NOTE: Presently APES+ does *not* possess the ability to generate **T6P1d** elements. It is assumed that the analyst will thus use some stand-alone pre-processing software to accomplish this task. The resulting element and node data will then be translated to the format expected by APES+.

- The **INITIAL** keyword is used to specify the initial state number associated with the element. The default value for the **INITIAL** is zero (0).

- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The default **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE STRAIN** idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the default condition, no input is required in such a case. The condition of no excavation is likewise the default.

- If the **PENALTY** keyword is specified, the *mixed/penalty* version of the element is instead adopted in lieu of a traditional mixed version. If the mixed element is to be used

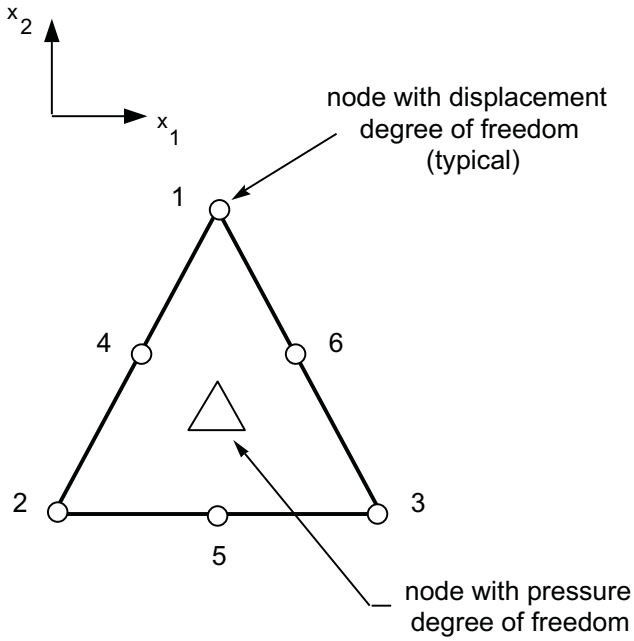

Figure 27.1: Typical mixed **T6P1d** element (discontinuous pressure field).

to simulate relatively compressible response (e.g., Poisson's ratio in the range from 0.0 to 0.40), then the *mixed* version of the element should be used. As the incompressible limit is approached, both the mixed and mixed/penalty formulations give essentially *identical* results.

- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT MIXED** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the keyword **DONT_PRINT_Results** indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the default condition both strains are printed.

- Specification of the keyword **DONT_PRINT_STRESSES** indicates that stresses are not to be printed. Under the default condition stresses are printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The keyword **PRINT_VOLUMETRIC_STRAIN** causes the volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the volumetric strain to the absolute value of the minimum non-zero normal strain in the element is printed. That is,

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33}) \neq 0$$

  This ratio is instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the default condition the volumetric strain and the aforementioned ratio are not computed and printed.

27.2 The ELEMENT MIXED TYPE T7P3d Command

Synopsis

The **ELEMENT MIXED TYPE T7P3d** command is used to describe all mixed, 7-node quadratic triangular continuum elements with a constant *discontinuous* pressure field.

Remarks

- The **T7P3d** is a mixed, triangular isoparametric continuum element. The element
  - Contains four (3) vertex nodes.
  - Contains four (3) mid-side nodes.
  - Contains one (1) interior node.
  - Has two (2) displacements degrees of freedom at each node, for a total of fourteen (14) displacement degrees of freedom.
  - Employs a *bi-quadratic* approximation for the displacement field that is enhanced by a cubic "bubble" function.
  - Employs a *linear*, discontinuous approximation for the pressure (Figure 27.2). The pressure degrees of freedom are "condensed" out [52] from the element prior to its assembly into the global system of equations.
- Since mixed or mixed-penalty continuum elements are typically used in simulating incompressible or nearly incompressible material response, **T7P3d** elements should be either *plane strain* or *torsionless axisymmetry* anaylses. No benefits are realized by using **T7P3d** elements for *plane stress* idealizations. Indeed, more accurate results are obtained by using the **T7P0** element, which is the irreducible counterpart to the **T7P3d** element.
- For analyses involving standard (compressible) materials no benefit is gained from using **T7P3d** elements. In such cases the irreducible **T7P0** element, or possibly **T6P0** element is preferred.
- The **T7P3d** element adds a cubic "bubble function" to the T6P3d element (Figure 27.2). The development of this "P2 bubble/P1 discontinuous" element is attributed to Fortin [33].
- The **T7P3d** element satisfies the Babuška-Brezzi condition [44]. The accuracy of the element is $O(h^2)$. The T7P3d is thus arguably the optimal triangular element [5].

Syntax

The following syntax is used to describe a typical mixed or mixed-penalty **T7P3d** continuum element:

ELEment MIXed TYPe [T7P3d] (DEScription "string")
NODes #:#:# (MATerial #) (INItial #)
(CONstruction #) (EXCavation #) (THIckness #.#) (PENalty)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLumetric_strain)
(PRINT_AVG_PREssure)

Explanatory Notes

- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.

- The numbering order of **NODES** associated with **T7P3d** elements, which must be specified sequentially from 1 to 7, is shown in Figure 27.2.

  NOTE: Presently APES+ does *not* possess the ability to generate **T7P3d** elements. It is assumed that the analyst will thus use some stand-alone pre-processing software to accomplish this task. The resulting element and node data will then be translated to the format expected by APES+.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default values for the **MATERIAL** number is one (1).

- The **INITIAL** keyword is used to specify the initial state number associated with the element. The default value for the **INITIAL** is zero (0).

- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The default **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE STRAIN** idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.

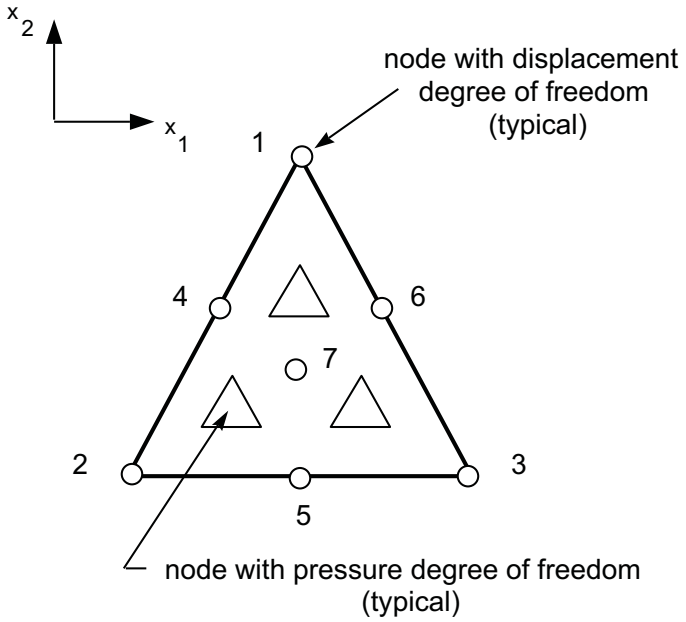

Figure 27.2: Typical mixed **T7P3d** element (discontinuous pressure field).

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the default condition, no input is required in such a case. The condition of no excavation is likewise the default.

- If the **PENALTY** keyword is specified, the *mixed/penalty* version of the element is instead adopted in lieu of a traditional mixed version. If the mixed element is to be used to simulate relatively compressible response (e.g., Poisson's ratio in the range from 0.0 to 0.40), then the *mixed* version of the element should be used. As the incompressible limit is approached, both the mixed and mixed/penalty formulations give essentially *identical* results.

- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT MIXED** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the keyword **DONT_PRINT_Results** indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains

are not to be printed. Under the default condition both strains are printed.

- Specification of the keyword **DONT_PRINT_STRESSES** indicates that stresses are not to be printed. Under the default condition stresses are printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_AVG_STRAINS** keyword indicates that average strains (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the default condition average strains are not computed and printed.

- The **PRINT_AVG_STRESSES** keyword indicates that average stresses (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the default condition average stresses are not computed and printed.

- The keyword **PRINT_VOLUMETRIC_STRAIN** causes the volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the volumetric strain to the absolute value of the minimum non-zero normal strain in the element is printed. That is,

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33}) \neq 0$$

  This ratio is instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the default condition the volumetric strain and the aforementioned ratio are not computed and printed.

- The keyword **PRINT_AVG_PRESSURE** causes the *average* pressure to be computed and printed for the element. This value represents the average of the approximate pressures at the three interior pressure nodes.

27.3 The ELEMENT MIXED TYPE Q4P1d Command

Synopsis

The **ELEMENT MIXED TYPE Q4P1d** command is used to describe all mixed, 4-node bi-linear quadrilateral continuum elements with a constant *discontinuous* pressure approximation.

Remarks

- The **Q4P1d** is a mixed, quadrilateral isoparametric "Lagrangian" continuum element [52]. The element
  - Contains four (4) vertex nodes.
  - Has two (2) displacements degrees of freedom at each node, for a total of eight (8) displacement degrees of freedom.
  - Employs a *bi-linear* approximation for the displacement field.
  - Employs a *constant*, discontinuous approximation for the pressure (Figure 27.3). The pressure degrees of freedom are "condensed" out [52] from the element prior to its assembly into the global system of equations.
- Since mixed or mixed-penalty continuum elements are typically used in simulating incompressible or nearly incompressible material response, **Q4P1d** elements should be either *plane strain* or *torsionless axisymmetry* anaylses. No benefits are realized by using **Q4P1d** elements for *plane stress* idealizations. Indeed, more accurate results are obtained by using the **Q4P0** element, which is the irreducible counterpart to the **Q4P1d** element.
- For analyses involving standard (compressible) materials no benefit is gained from using **Q4P1d** elements. In such cases the irreducible **Q4P0** element is preferred.
- The **Q4P1d** element does *not* satisfy the Babuška-Brezzi condition. [44]
- Because of the constant pressure approximation, the **Q4P1d** element simulates incompressibility *in the mean*.
- Meshes of **Q4P1d** elements, analyzed in the incompressible limit, are known to be *conditionally* stable [97; 44]. More precisely, the stability is mesh-dependent: the optimal convergence rate (linear) can be achieved by creating "macroelements" from four adjacent Q4P1d elements, with *straight* edges along the sides of each macroelement. However, such restrictions limit the model's usefulness in simulations in which the boundary is allowed to move.

- Instability of the **Q4P1d** element is manifested through spurious pressure modes in the solution. The element is, however, included in the APES+ computer program to facilitate comparisons with other mixed-penalty elements.

Syntax

The following syntax is used to describe a typical mixed or mixed-penalty **Q4P1d** continuum element:

ELEment MIXed TYPe [Q4P1d] (DEScription "string")
NODes #:#:# (MATerial #) (INItial #)
(CONstruction #) (EXCavation #) (THIckness #.#) (PENalty)
(1_Additional #) (1_Increment #)
(2_Additional #) (2_Increment #)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLumetric_strain)

Explanatory Notes

- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with **Q4P1d** elements, which must be specified sequentially from 1 to 4, is shown in Figure 27.3).
- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default values for the **MATERIAL** number is one (1).
- The **INITIAL** keyword is used to specify the initial state number associated with the element. The default value for the **INITIAL** is zero (0).
- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The default **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE**

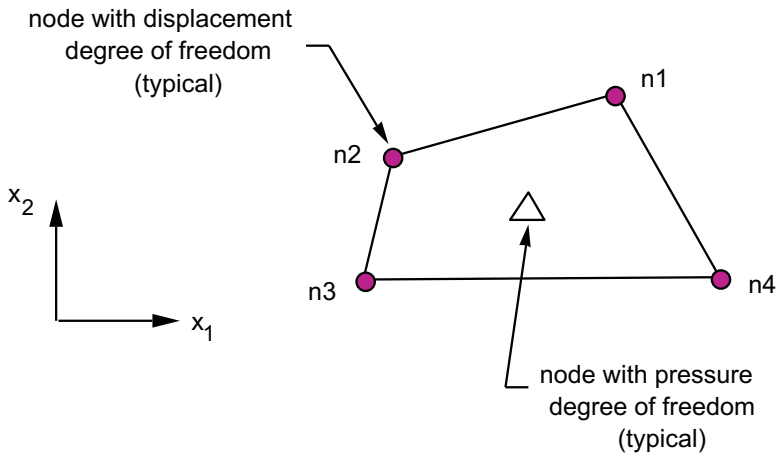

Figure 27.3: Typical mixed **Q4P1d** element (discontinuous constant pressure field).

STRAIN idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the default condition, no input is required in such a case. The condition of no excavation is likewise the default.

- If the **PENALTY** keyword is specified, the *mixed/penalty* version of the element is instead adopted in lieu of a traditional mixed version. If the mixed element is to be used to simulate relatively compressible response (e.g., Poisson's ratio in the range from 0.0 to 0.40), then the *mixed* version of the element should be used. As the incompressible limit is approached, both the mixed and mixed/penalty formulations give essentially *identical* results.

- If the body being analyzed can be divided into a layer (or layers) of elements, and if the characteristics of the element (i.e., the **MATERIAL**, the **INITIAL** state, the incremental **CONSTRUCTION** and **EXCAVATION** numbers, the **THICKNESS**, and the **INTCODE**) are the *same* for several elements within a layer, and if the nodes are numbered in a *consistent* fashion, then an element data generation option can be employed. To generate a sequence of elements within a single layer, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**.

  For example, assuming that the coordinates of all boundary nodes have been specified, the bottom row of elements in the mesh shown in Figure 27.4 could be established by entering the node numbers associated with element *a* and the keywords/values **1_ADDITIONAL** 4 and **1_INCREMENT** 5. Assuming a **MATERIAL** number

equal to one (1) and default values for all of the other element parameters, the input record corresponding to this specification would be:

```
element mixed type "q4p1" nodes  1  6  7  2  material  1 &
   1_add  4   1_incr 5
```

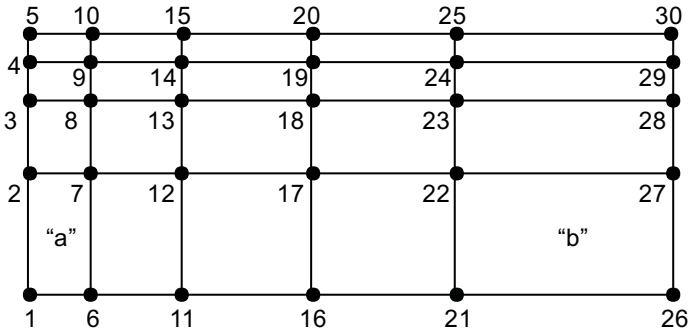

Figure 27.4: Hypothetical mesh consisting of **Q4P1d** elements.

Alternately, this bottom row of elements could be established by entering the node numbers associated with element *b* and the keywords/values **1_ADDITIONAL** 4 and **1_INCREMENT** -5 (i.e., a decrement). Assuming once again a **MATERIAL** number equal to one (1) and default values for all of the other element parameters, the input record corresponding to this specification would be as follows:

```
element mixed type "q4p1" nodes  21  26  27  22 material 1 &
   1_add  4   1_incr -5
```

In a similar manner, the left most column of elements in the mesh could be established by entering the node numbers associated with element *a* and the keywords/values **1_ADDITIONAL** 2 and **1_INCREMENT** 1. The corresponding input record would be as follows:

```
element mixed type "q4p1" nodes  1  6  7  2  material  1 &
   1_add  2   1_incr 1
```

If several layers of elements have the *same* attributes, the above generation option can be carried one step further. For example, the entire mesh shown in Figure 27.4 could be established by entering the node numbers of element *a* and the keywords/values **1_ADDITIONAL** 4, **1_INCREMENT** 5, **2_ADDITIONAL** 3, and **2_INCREMENT** 1. The associated input record would thus be as follows:

```
element mixed type "q4p1" nodes  1  6  7  2  material  1 &
   1_add  4   1_incr 5  2_add  3   2_incr 1
```

Note that the 1- and 2-directions need not be orthogonal, nor be aligned with the global coordinate directions. Alternately, the mesh shown in Figure 27.4 could be generated by specifying the node points associated with element a and the keywords/values **1_ADDITIONAL** 3, **1_INCREMENT** 1, **2_ADDITIONAL** 4, and **2_INCREMENT** 5. The associated input record would thus be as follows:

```
element mixed type "q4p1" nodes  1  6  7  2  material  1 &
   1_add  2   1_incr 1  2_add  4   2_incr 5
```

The mesh generated with these two specifications would differ only in the order in which the elements were numbered by the APES+ program.

- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT MIXED** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the keyword **DONT_PRINT_Results** indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the default condition both strains are printed.

- Specification of the keyword **DONT_PRINT_STRESSES** indicates that stresses are not to be printed. Under the default condition stresses are printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The keyword **PRINT_VOLUMETRIC_STRAIN** causes the volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the volumetric strain to the absolute value of the minimum non-zero normal strain in the element is printed. That is,

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33}) \neq 0$$

This ratio is instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the default condition the volumetric strain and the aforementioned ratio are not computed and printed.

27.4 The ELEMENT MIXED TYPE Q8P3d Command

Synopsis

The **ELEMENT MIXED TYPE Q8P3d** command is used to describe all mixed, 8-node quadratic quadrilateral continuum elements with a linear *discontinuous* pressure approximation.

Remarks

- The **Q8P3d** is a mixed, quadrilateral isoparametric "serendipity" continuum element [52]. The element
  - Contains four (4) vertex nodes.
  - Contains four (4) mid-side nodes.
  - Has two (2) displacements degrees of freedom at each node, for a total of sixteen (16) displacement degrees of freedom.
  - Employs a *quadratic* approximation for the displacement field.
  - Employs a *linear*, discontinuous approximation for the pressure (Figure 27.5). The pressure degrees of freedom are "condensed" out [52] from the element prior to its assembly into the global system of equations.
- Since mixed or mixed-penalty continuum elements are typically used in simulating incompressible or nearly incompressible material response, **Q8P3d** elements should be either *plane strain* or *torsionless axisymmetry* anaylses. No benefits are realized by using **Q8P3d** elements for *plane stress* idealizations. Indeed, more accurate results are obtained by using the **Q8P0** element, which is the irreducible counterpart to the **Q8P3d** element.
- For analyses involving standard (compressible) materials no benefit is gained from using **Q8P3d** elements. In such cases the irreducible **Q8P0** element is preferred.
- The **Q8P3d** element does *not* satisfy the Babuška-Brezzi condition and is considered to be a "borderline" element [44]. Consequently, the use of the bi-quadratic **Q9P3d** element is generally preferred. The **Q8P3d** element is, however, included in the APES+ computer program to facilitate comparisons with the **Q9P3d** element, as well as other mixed-penalty elements.

Syntax

The following syntax is used to describe a typical mixed or mixed-penalty **Q8P3d** continuum element:

ELEment MIXed TYPe [Q8P3d] (DEScription "string")
NODes #:#:# (MATerial #) (INItial #)
(CONstruction #) (EXCavation #) (THIckness #.#) (PENalty)
(1_Additional #) (1_Increment #)
(2_Additional #) (2_Increment #)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_AVG_STRAins) (PRINT_AVG_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLumetric_strain) (PRINT_AVG_VOLumetric_strain)
(PRINT_AVG_PREssure)

Explanatory Notes

- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with **Q8P3d** elements, which must be specified sequentially from 1 to 8, is shown in Figure 27.5).

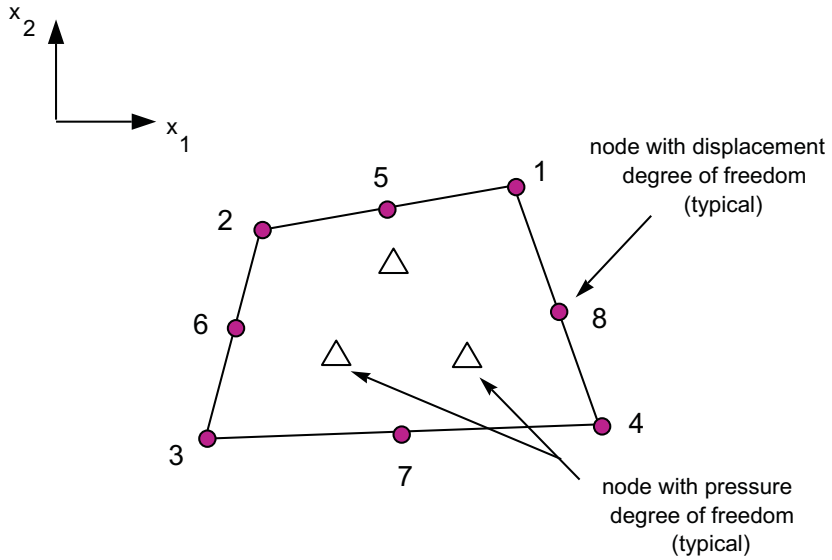

Figure 27.5: Typical mixed **Q8P3d** element (discontinuous pressure field).

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default values for the **MATERIAL** number is one (1).

- The **INITIAL** keyword is used to specify the initial state number associated with the element. The default value for the **INITIAL** is zero (0).

- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The default **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE STRAIN** idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the default condition, no input is required in such a case. The condition of no excavation is likewise the default.

- If the **PENALTY** keyword is specified, the *mixed/penalty* version of the element is instead adopted in lieu of a traditional mixed version. If the mixed element is to be used to simulate relatively compressible response (e.g., Poisson's ratio in the range from 0.0 to 0.40), then the *mixed* version of the element should be used. As the incompressible limit is approached, both the mixed and mixed/penalty formulations give essentially *identical* results.

- If the body being analyzed can be divided into a layer (or layers) of elements, and if the characteristics of the element (i.e., the **MATERIAL**, the **INITIAL** state, the incremental **CONSTRUCTION** and **EXCAVATION** numbers, and the **THICKNESS** are the *same* for several elements within a layer, and if the nodes are numbered in a *consistent* fashion, then an element data generation option can be employed. To generate a sequence of elements within a single layer, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**.

  If several layers of elements have the *same* attributes, the above generation option can be carried one step further by making use of the **2_ADDITIONAL**, and **2_INCREMENT** keywords.

- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT MIXED** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the keyword **DONT_PRINT_Results** indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.
- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the default condition both strains are printed.
- Specification of the keyword **DONT_PRINT_STRESSES** indicates that stresses are not to be printed. Under the default condition stresses are printed.
- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.
- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.
- The **PRINT_AVG_STRAINS** keyword indicates that average strains (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the default condition average strains are not computed and printed.
- The **PRINT_AVG_STRESSES** keyword indicates that average stresses (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the default condition average stresses are not computed and printed.
- The keyword **PRINT_VOLUMETRIC_STRAIN** causes the volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the volumetric strain to the absolute value of the minimum non-zero normal strain in the element is printed. That is,

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33}) \neq 0$$

  This ratio is instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the default condition the volumetric strain and the aforementioned ratio are *not* computed and printed.
- The keyword **PRINT_AVG_VOLUMETRIC_STRAIN** causes the *average* volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the *average* volumetric strain to the absolute value of the minimum *average* non-zero normal strain in the element is printed. This ratio is also instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the default condition the average volumetric strain and the aforementioned ratio are *not* computed and printed.
- The keyword **PRINT_AVG_PRESSURE** causes the *average* pressure to be computed and printed for the element. This value represents the average of the approximate pressures at the three interior pressure nodes.

27.5 The ELEMENT MIXED TYPE Q9P3d Command

Synopsis

The **ELEMENT MIXED Q9P3d** command is used to describe all mixed, 9-node bi-quadratic quadrilateral continuum elements with a linear *discontinuous* pressure approximation.

Remarks

- The **Q9P3d** is a mixed, quadrilateral isoparametric "Lagrangian" continuum element [52]. The element
  - Contains four (4) vertex nodes.
  - Contains four (4) mid-side nodes.
  - Contains one (1) interior node.
  - Has two (2) displacements degrees of freedom at each node, for a total of sixteen (16) displacement degrees of freedom.
  - Employs a *bi-quadratic* approximation for the displacement field.
  - Employs a *linear*, discontinuous approximation for the pressure (Figure 27.6). The pressure degrees of freedom are "condensed" out [52] from the element prior to its assembly into the global system of equations.
- Since mixed or mixed-penalty continuum elements are typically used in simulating incompressible or nearly incompressible material response, **Q9P3d** elements should be either *plane strain* or *torsionless axisymmetry* anaylses. No benefits are realized by using **Q9P3d** elements for *plane stress* idealizations. Indeed, more accurate results are obtained by using the **Q9P0** element, which is the irreducible counterpart to the **Q9P3d** element.
- For analyses involving standard (compressible) materials no benefit is gained from using **Q9P3d** elements. In such cases the irreducible **Q9P0** element is preferred.
- The **Q9P3d** element satisfies the Babuška-Brezzi condition [44]. The accuracy of the element is $O(h^2)$. The **Q9P3d** is thus an *optimal* element. This implies that convergence will be *quadratic* for $\hat{\mathbf{u}}^{(\mathbf{e})}$ in the H^1 norm; it will likewise be quadratic for $\hat{p}^{(e)}$ in the L_2 norm [44].

Syntax

The following syntax is used to describe a typical mixed or mixed-penalty **Q9P3d** continuum element:

ELEment MIXed TYPe [Q9P3d] (DEScription "string")
NODes #:#:# (MATerial #) (INItial #)
(CONstruction #) (EXCavation #) (THIckness #.#) (PENalty)
(1_Additional #) (1_Increment #)
(2_Additional #) (2_Increment #)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_AVG_STRAins) (PRINT_AVG_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLumetric_strain) (PRINT_AVG_VOLumetric_strain)
(PRINT_AVG_PREssure)

Explanatory Notes

- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with **Q9P3d** elements, which must be specified sequentially from 1 to 9, is shown in Figure 27.6.

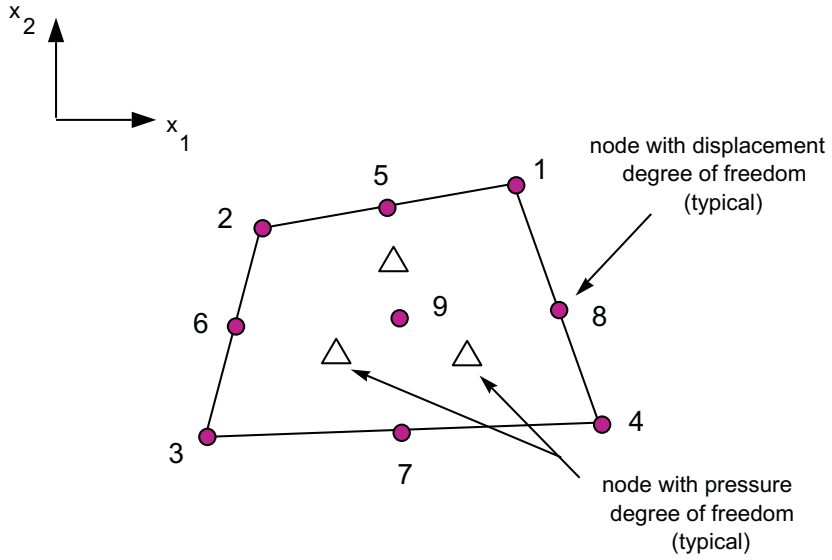

Figure 27.6: Typical mixed **Q9P3d** element (discontinuous pressure field).

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default values for the **MATERIAL** number is one (1).
- The **INITIAL** keyword is used to specify the initial state number associated with the element. The default value for the **INITIAL** is zero (0).
- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The default **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE STRAIN** idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.
- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the default condition, no input is required in such a case. The condition of no excavation is likewise the default.
- If the **PENALTY** keyword is specified, the *mixed/penalty* version of the element is instead adopted in lieu of a traditional mixed version. If the mixed element is to be used to simulate relatively compressible response (e.g., Poisson's ratio in the range from 0.0 to 0.40), then the *mixed* version of the element should be used. As the incompressible limit is approached, both the mixed and mixed/penalty formulations give essentially *identical* results.
- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT MIXED** command, then all the elements generated will be affected in a like manner by the above print control commands.
- Specification of the keyword **DONT_PRINT_Results** indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.
- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the default condition both strains are printed.
- Specification of the keyword **DONT_PRINT_STRESSES** indicates that stresses are not to be printed. Under the default condition stresses are printed.
- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_AVG_STRAINS** keyword indicates that average strains (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the default condition average strains are not computed and printed.

- The **PRINT_AVG_STRESSES** keyword indicates that average stresses (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the default condition average stresses are not computed and printed.

- The keyword **PRINT_VOLUMETRIC_STRAIN** causes the volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the volumetric strain to the absolute value of the minimum non-zero normal strain in the element is printed. That is,

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33}) \neq 0$$

  This ratio is instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the default condition the volumetric strain and the aforementioned ratio are *not* computed and printed.

- The keyword **PRINT_AVG_VOLUMETRIC_STRAIN** causes the *average* volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the *average* volumetric strain to the absolute value of the minimum *average* non-zero normal strain in the element is printed. This ratio is also instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the default condition the average volumetric strain and the aforementioned ratio are *not* computed and printed.

- The keyword **PRINT_AVG_PRESSURE** causes the *average* pressure to be computed and printed for the element. This value represents the average of the approximate pressures at the three interior pressure nodes.

27.6 The ELEMENT MIXED TYPE H8P1d Command

Synopsis

The **ELEMENT MIXED TYPE H8P1d** command is used to describe all mixed, 8-node trilinear hexahedral continuum elements with a constant *discontinuous* pressure field.

Remarks

- The **H8P1d** is a mixed, hexahedral isoparametric "Lagrangian" continuum element [52]. The element
  - Contains eight (8) vertex nodes.
  - Has three (3) displacements degrees of freedom at each node, for a total of twenty-four (24) displacement degrees of freedom.
  - Employs a *tri-linear* approximation for the displacement field.
  - Employs a *constant*, discontinuous approximation for the pressure (Figure 27.7). The pressure degrees of freedom are "condensed" out [52] from the element prior to its assembly into the global system of equations.
- For analyses involving standard (compressible) materials no benefit is gained from using **H8P1d** elements. In such cases the irreducible **H8P0** element is preferred.
- The **H8P1d** element does *not* satisfy the Babuška-Brezzi condition. [44]
- Because of the constant pressure approximation, the **H8P1d** element simulates incompressibility *in the mean*.
- Similar to the **Q4P1d** element **??**, instability of the **H8P1d** element is manifested through spurious pressure modes in the solution. The element is, however, included in the APES+ computer program to facilitate comparisons with other three-dimensional mixed-penalty elements.

Syntax

The following syntax is used to describe a typical mixed or mixed-penalty **H8P1d** continuum element:

ELEment MIXed TYPe [H8P1d] (DEScription "string")
NODes #:#:# (MATerial #) (INItial #)
(CONstruction #) (EXCavation #) (PENalty)

(1_Additional #) (1_Increment #)
(2_Additional #) (2_Increment #)
(3_Additional #) (3_Increment #)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLumetric_strain)

Explanatory Notes

- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with **H8P1d** elements, which must be specified sequentially from 1 to 8, is shown in Figure 27.7).

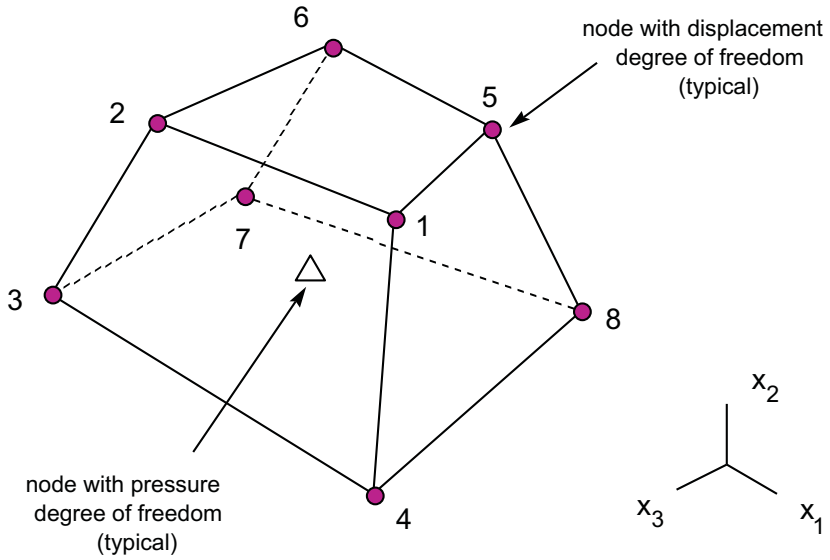

Figure 27.7: Typical mixed **H8P1d** element (discontinuous constant pressure field).

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default values for the **MATERIAL** number is one (1).
- The **INITIAL** keyword is used to specify the initial state number associated with the element. The default value for the **INITIAL** is zero (0).

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the default condition, no input is required in such a case. The condition of no excavation is likewise the default.

- If the **PENALTY** keyword is specified, the *mixed/penalty* version of the element is instead adopted in lieu of a traditional mixed version. If the mixed element is to be used to simulate relatively compressible response (e.g., Poisson's ratio in the range from 0.0 to 0.40), then the *mixed* version of the element should be used. As the incompressible limit is approached, both the mixed and mixed/penalty formulations give essentially *identical* results.

- If the body being analyzed can be divided into a layer (or layers) of elements, and if the characteristics of the element (i.e., the **MATERIAL**, the **INITIAL** state, the incremental **CONSTRUCTION** and **EXCAVATION** numbers, the **THICKNESS**, and the **INTCODE**) are the *same* for several elements within a layer, and if the nodes are numbered in a *consistent* fashion, then an element data generation option can be employed. To generate a sequence of elements within a single layer, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**.

  If several layers of elements have the *same* attributes, the above generation option can be carried one step further by making use of the **2_ADDITIONAL**, and **2_INCREMENT** keywords, and possibly the **3_ADDITIONAL**, and **3_INCREMENT** keywords.

- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT MIXED** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the keyword **DONT_PRINT_Results** indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the default condition both strains are printed.

- Specification of the keyword **DONT_PRINT_STRESSES** indicates that stresses are not to be printed. Under the default condition stresses are printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the default condition these quantities are not computed and printed.

- The keyword **PRINT_VOLUMETRIC_STRAIN** causes the volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the volumetric strain to the absolute value of the minimum non-zero normal strain in the element is printed. That is,

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33}) \neq 0$$

  This ratio is instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the default condition the volumetric strain and the aforementioned ratio are not computed and printed.

Chapter 28

ELEMENT MIXED FLOW
Commands for Elements with Continuous Pressure Approximations

The available mixed continuum elements with continuous pressure approximations for analyses that include the flow of pore fluid are listed below. These elements are entered using the **ELEMENT MIXED_FLOW** commands.

Type **T4P3c**: a conforming 4-node mixed isoparametric triangular element. Uses an "enhanced" *linear* displacement interpolation and a continuous *linear* pressure interpolation.

- The command syntax for this element type is available in Section 28.1.

Type **T6P3c**: a conforming 6-node mixed isoparametric triangular element. It uses a *quadratic* displacement interpolation and a continuous *linear* pressure interpolation.

- The command syntax for this element type is available in Section 28.2.

Type **T7P3c**: a conforming 7-node mixed isoparametric triangular element. It uses a *enhanced quadratic* displacement interpolation and a continuous *linear* pressure interpolation.

- The command syntax for this element type is available in Section 28.3.

Type **T15P6c**: a conforming 15-node mixed isoparametric triangular element. It uses a *quartic* displacement interpolation and a continuous *quadratic* pressure interpolation.

- The command syntax for this element type is available in Section 28.4.

Type **Q5P4c**: a conforming 5-node mixed isoparametric quadrilateral element. Uses an "enhanced" *bilinear* displacement interpolation and a continuous *bilinear* pressure interpolation.

- The command syntax for this element type is available in Section 28.5.

Type **Q8P4c**: a conforming 8-node mixed isoparametric quadrilateral element. It uses a *quadratic* displacement interpolation and a continuous *bilinear* pressure interpolation.

- The command syntax for this element type is available in Section 28.6.

Type **Q9P4c**: a conforming 9-node mixed isoparametric quadrilateral element. It uses a *biquadratic* displacement interpolation and a continuous *bilinear* pressure interpolation.

- The command syntax for this element type is available in Section 28.7.

Type **Q17P8c**: a conforming 17-node mixed isoparametric quadrilateral element. It uses a *quartic* displacement interpolation and a continuous *biquadratic* pressure interpolation.

- The command syntax for this element type is available in Section 28.8.

Type **H9P8c**: a conforming 9-node mixed isoparametric hexahedral element. Uses an "enhanced" *trilinear* displacement interpolation and a continuous *trilinear* pressure interpolation.

- The command syntax for this element type is available in Section 28.9.

Type **H20P8c**: a conforming 20-node mixed isoparametric hexahedral element. Uses a *quadratic* displacement interpolation and a continuous *trilinear* pressure interpolation.

- The command syntax for this element type is available in Section 28.10.

Type **H27P8c**: a conforming 27-node mixed isoparametric hexahedral element. Uses a *tri-quadratic* displacement interpolation and a continuous *trilinear* pressure interpolation.

- The command syntax for this element type is available in Section 28.11.

Type **Q4P4c**: a conforming 4-node mixed isoparametric quadrilateral equal-order interpolation (EOI) element with *hybrid* selective reduced integration. It uses a *bilinear* displacement interpolation and a continuous *bilinear* pressure interpolation.

- The command syntax for this element type is available in Section 28.12.

Type **H8P8c**: a conforming 8-node mixed isoparametric hexahedral equal-order interpolation (EOI) element with *hybrid* selective reduced integration. It uses a *trilinear* displacement interpolation and a continuous *trilinear* pressure interpolation.

- The command syntax for this element type is available in Section 28.13.

28.1 The ELEMENT MIXED_FLOW T4P3c Command

Synopsis

The **ELEMENT MIXED_FLOW T4P3c** command is used to describe all conforming 4-node mixed "enhanced" triangular continuum elements that add a *cubic* displacement "bubble" function to the linear triangular element. To this is added a continuous linear pressure approximation.

Remarks

- The **T4P3c** is a mixed, isoparametric triangular element [52].
- The element
  - Contains four (3) vertex nodes.
  - Contains one (1) interior node. Associated with this node is a *cubic* interpolation function that "enhances" the element.
  - Has two (2) displacements degrees of freedom at each node, for a total of eight (8) displacement degrees of freedom.
  - Employs a *bilinear*, continuous approximation for the pressure (Fig. 28.1).
  - Has a total of three (3) pressure degrees of freedom.
- The **T4P3c** element should be used for coupled displacement-flow analyses of porous media. The element is based on a generalized Biot formulation and employs a $u - p$ form of the governing equations.

Syntax

The following syntax is used to describe a typical **T4P3c** mixed-flow quadrilateral continuum element:

ELEment MIXed_Flow TYPe T4P3c (DEScription "string")
NODes #:#:# (MATerial #) (HYDraulic #.#) (INItial #)
(CONstruction #) (EXCavation #) (THIckness #.#)
(FLUID_Density #) (SOLID_Density #) (FLUID_Bulk #) (SOLID_Bulk #)
(INCompressible_constituents)
(1_Additional #) (1_Increment #) (2_Additional #) (2_Increment #)
(DONT_PRINT_Results)

(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLumetric_strain) (PRINT_VELocity)

Explanatory Notes

- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENTS OFF** or the **ECHO ELEMENT_DESCRIPTIONS OFF** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with the **T4P3c** element is shown in Figure 28.1. For this element the nodes are numbered sequentially from 1 to 4.

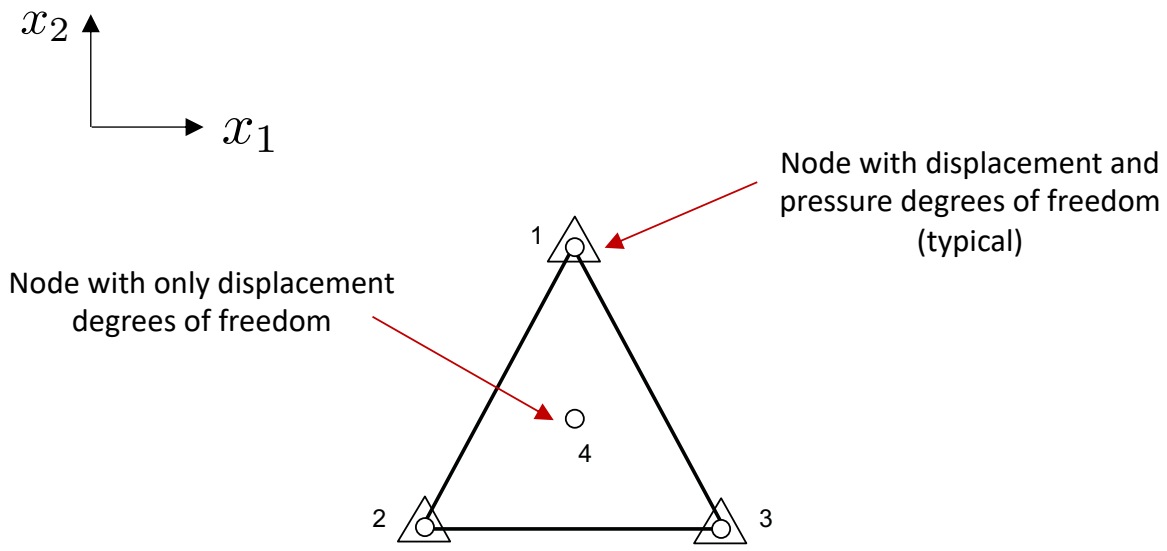

Figure 28.1: Node numbering associated with a typical mixed 4-node "enhanced" T4P3c quadrilateral continuum element.

- The **HYDRAULIC** keyword is used to specify the number of the hydraulic conductivity (permeability) idealization associated with the element. In order to maximize flexibility, the **HYDRAULIC** number differs from the **MATERIAL** number. The default value for the **HYDRAULIC** conductivity number is one (1).
- The **INITIAL** keyword is used to specify the initial state number associated with the element. The *default* value for the **INITIAL** is zero (0).

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to, or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the *default* condition, no input is required in such a case. The condition of no excavation is likewise the default. In the current version of APES+ the hydraulic boundary condition at the temporary top of a soil mass that is being constructed is *zero flow*; i.e., an *impervious* surface. The conditions on top of the final surface can, however, be specified through the use of the **SPECIFICATIONS CONCENTRATED FLOW** command.

- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The *default* **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE STRAIN** idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.

- The **FLUID_DENSITY** keyword is used to specify the number of the mass density idealization for the fluid phase (ρ^f) (units of ML^{-3} or $\mathrm{FL}^{-4}\mathrm{t}^2$) associated with the element. The **SOLID_DENSITY** keyword is used to specify the number of the mass density idealization for the solid phase (ρ^s) associated with the element.

  Given the values of ρ^f and ρ^s (specified using the **DENSITY_FLUID** and **DENSITY_SOLID** commands, respectively, as described in Chapter 17, the total mass density of a saturated soil is computed from

  $$\rho = (1-n)\rho^s + n\rho^f$$

  where n is the porosity, which is related to the void ratio (e) by $n = e/(1+e)$.

- The **FLUID_BULK** keyword is used to specify the number of the bulk modulus idealization for the fluid phase (K_f) (units of FL^{-2}) associated with the element. The **SOLID_BULK** keyword is used to specify the number of the bulk modulus idealization for the solid phase (K_s) associated with the element.

  Given the values of K_f and K_s (specified using the **BULK_FLUID** and **BULK_SOLID** commands, respectively, as described in Chapter 16, the value of the "Biot" modulus [27] is computed as follows:

  $$\frac{1}{M} = \frac{n}{K_f} + \frac{1-n}{K_s}$$

  where n is again the porosity. An explicit value of M can be input using the **MATERIAL POROELASTIC ANISOTROPIC** command.

- If the **INCompressible_constituents** keyword is used, both the solid skeleton and the pore fluid are assumed to be *incompressible*. This supersedes any bulk modulus values specified in conjunction with the **MATERIAL FLUID** or **MATERIAL SOLID**

commands. If an anisotropic poroelastic material idealization is used (refer to the description of the **MATERIAL POROELASTIC ANISOTROPIC** command), this implies an *infinite* value for the "Biot modulus" M (specified via the **M_MODULUS** keyword) for the element.

- If the body being analyzed can be divided into a layer (or layers) of elements, and if the characteristics of the element (i.e., the **MATERIAL**, the **INITIAL** state, the incremental **CONSTRUCTION** and **EXCAVATION** numbers, and the **THICKNESS** value) are the *same* for several elements within a layer, and if the nodes are numbered in a *consistent* fashion, then an element data generation option can be employed. To generate a sequence of elements within a single layer, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**.

  If several layers of elements have the *same* attributes, the above generation option can be carried one step further by making use of the **2_ADDITIONAL**, and **2_INCREMENT** keywords.

- The purpose of the **PRINT** commands is to eliminate unnecessary output (printed and saved) generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the *default* condition both strains are printed.

- Specification of the **DONT_PRINT_STRESSES** keyword indicates that stresses are not to be printed. Under the *default* condition stresses are printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the *default* condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the *default* condition these quantities are not computed and printed.

- The **PRINT_AVG_STRAINS** keyword indicates that average strains (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the *default* condition average strains are not computed and printed.

- The **PRINT_AVG_STRESSES** keyword indicates that average stresses (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the *default* condition average stresses are not computed and printed.

- The **PRINT_VOLUMETRIC_STRAIN** keyword causes the volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the volumetric strain to the absolute value of the minimum non-zero normal strain in the element is printed. That is,

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33}) \neq 0$$

  This ratio is instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the *default* condition the volumetric strain and the aforementioned ratio are *not* computed and printed.

- The **PRINT_AVG_VOLUMETRIC_STRAIN** keyword causes the *average* volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the *average* volumetric strain to the absolute value of the minimum *average* non-zero normal strain in the element is printed. This ratio is also instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the *default* condition the average volumetric strain and the aforementioned ratio are *not* computed and printed.

- The **PRINT_AVG_PRESSURE** keyword causes the *average* pressure to be computed and printed for the element. This value represents the average of the approximate pressures at the four pressure (vertex) nodes.

- The **PRINT_VELOCITY** keyword causes the components of the Darcy velocity in the global x_1- and x_2-coordinate directions to be computed and printed at the element center.

28.2 The ELEMENT MIXED_FLOW T6P3c Command

Synopsis

The **ELEMENT MIXED_FLOW T6P3c** command is used to describe all mixed, 6-node, quadratic triangular continuum elements that are to be used in a coupled flow-mechanical analysis.

Remarks

- The **T6P3c** is a conforming 6-node mixed, isoparametric triangular element [52].
- The element
  - Contains three (3) vertex nodes.
  - Contains three (3) mid-side nodes.
  - Has two (2) displacements degrees of freedom at each node, for a total of twelve (12) displacement degrees of freedom.
  - Employs a *quadratic* approximation for the displacement field.
  - Employs a *linear*, continuous approximation for the pressure (Fig. 28.2).
  - Has a total of three (3) pressure degrees of freedom.
- The **T6P3c** element should be used for coupled displacement-flow analyses of porous media. The element is based on a generalized Biot formulation.

Syntax

The following syntax is used to describe a typical **T6P3c** mixed-flow quadrilateral continuum element:

ELEment MIXed_Flow TYPe T6P3c (DEScription "string")
NODes #:#:# (MATerial #) (HYDraulic #.#) (INItial #)
(CONstruction #) (EXCavation #) (THIckness #.#)
(FLUID_Density #) (SOLID_Density #) (FLUID_Bulk #) (SOLID_Bulk #)
(INCompressible_constituents)
(1_Additional #) (1_Increment #)
(2_Additional #) (2_Increment #)
(DONT_PRINT_Results)

(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLumetric_strain) (PRINT_AVG_PREssure) (PRINT_VELocity)

Explanatory Notes

- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENTS OFF** or the **ECHO ELEMENT_DESCRIPTIONS OFF** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with the **T6P3c** element is shown in Figure 28.2. For this element the nodes are numbered sequentially from 1 to 6.

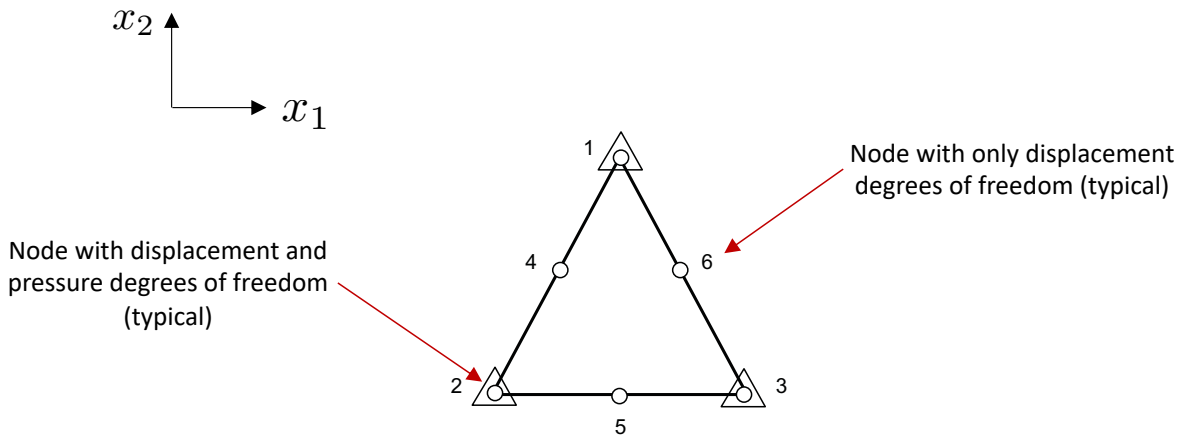

Figure 28.2: Node numbering associated with a typical mixed 6-node quadratic (T6P3c) triangular continuum element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default value for the **MATERIAL** number is one (1).
- The **HYDRAULIC** keyword is used to specify the number of the hydraulic conductivity (permeability) idealization associated with the element. In order to maximize flexibility, the **HYDRAULIC** number differs from the **MATERIAL** number. The default value for the **HYDRAULIC** conductivity number is one (1).
- The **INITIAL** keyword is used to specify the initial state number associated with the element. The *default* value for the **INITIAL** is zero (0).

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to, or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the *default* condition, no input is required in such a case. The condition of no excavation is likewise the default. In the current version of APES+ the hydraulic boundary condition at the temporary top of a soil mass that is being constructed is *zero flow*; i.e., an *impervious* surface. The conditions on top of the final surface can, however, be specified through the use of the **SPECIFICATIONS CONCENTRATED FLOW** command.

- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The *default* **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE STRAIN** idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.

- The **FLUID_DENSITY** keyword is used to specify the number of the mass density idealization for the fluid phase (ρ^f) (units of ML^{-3} or $\mathrm{FL}^{-4}\mathrm{t}^2$) associated with the element. The **SOLID_DENSITY** keyword is used to specify the number of the mass density idealization for the solid phase (ρ^s) associated with the element.

  Given the values of ρ^f and ρ^s (specified using the **DENSITY_FLUID** and **DENSITY_SOLID** commands, respectively, as described in Chapter 17, the total mass density of a saturated soil is computed from

  $$\rho = (1-n)\rho^s + n\rho^f$$

  where n is the porosity, which is related to the void ratio (e) by $n = e/(1+e)$.

- The **FLUID_BULK** keyword is used to specify the number of the bulk modulus idealization for the fluid phase (K_f) (units of FL^{-2}) associated with the element. The **SOLID_BULK** keyword is used to specify the number of the bulk modulus idealization for the solid phase (K_s) associated with the element.

  Given the values of K_f and K_s (specified using the **BULK_FLUID** and **BULK_SOLID** commands, respectively, as described in Chapter 16, the value of the "Biot" modulus [27] is computed as follows:

  $$\frac{1}{M} = \frac{n}{K_f} + \frac{1-n}{K_s}$$

  where n is again the porosity. An explicit value of M can be input using the **MATERIAL POROELASTIC ANISOTROPIC** command.

- If the **INCompressible_constituents** keyword is used, both the solid skeleton and the pore fluid are assumed to be *incompressible*. This supersedes any bulk modulus values specified in conjunction with the **MATERIAL FLUID** or **MATERIAL SOLID**

commands. If an anisotropic poroelastic material idealization is used (refer to the description of the **MATERIAL POROELASTIC ANISOTROPIC** command), this implies an *infinite* value for the "Biot modulus" M (specified via the **M_MODULUS** keyword) for the element.

- If the body being analyzed can be divided into a layer (or layers) of elements, and if the characteristics of the element (i.e., the **MATERIAL**, the **INITIAL** state, the incremental **CONSTRUCTION** and **EXCAVATION** numbers, and the **THICKNESS** value) are the *same* for several elements within a layer, and if the nodes are numbered in a *consistent* fashion, then an element data generation option can be employed. To generate a sequence of elements within a single layer, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**.

  If several layers of elements have the *same* attributes, the above generation option can be carried one step further by making use of the **2_ADDITIONAL**, and **2_INCREMENT** keywords.

- The purpose of the **PRINT** commands is to eliminate unnecessary output (printed and saved) generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the *default* condition both strains are printed.

- Specification of the **DONT_PRINT_STRESSES** keyword indicates that stresses are not to be printed. Under the *default* condition stresses are printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the *default* condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the *default* condition these quantities are not computed and printed.

- The **PRINT_AVG_STRAINS** keyword indicates that average strains (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the *default* condition average strains are not computed and printed.

- The **PRINT_AVG_STRESSES** keyword indicates that average stresses (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the *default* condition average stresses are not computed and printed.

- The **PRINT_VOLUMETRIC_STRAIN** keyword causes the volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the volumetric strain to the absolute value of the minimum non-zero normal strain in the element is printed. That is,

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33}) \neq 0$$

  This ratio is instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the *default* condition the volumetric strain and the aforementioned ratio are *not* computed and printed.

- The **PRINT_AVG_VOLUMETRIC_STRAIN** keyword causes the *average* volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the *average* volumetric strain to the absolute value of the minimum *average* non-zero normal strain in the element is printed. This ratio is also instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the *default* condition the average volumetric strain and the aforementioned ratio are *not* computed and printed.

- The **PRINT_AVG_PRESSURE** keyword causes the *average* pressure to be computed and printed for the element. This value represents the average of the approximate pressures at the four pressure (vertex) nodes.

- The **PRINT_VELOCITY** keyword causes the components of the Darcy velocity in the global x_1- and x_2-coordinate directions to be computed and printed at the element center.

28.3 The ELEMENT MIXED_FLOW T7P3c Command

Synopsis

The **ELEMENT MIXED_FLOW T7P3c** command is used to describe all mixed, 15-node, quartic triangular continuum elements that are to be used in a coupled flow-mechanical analysis.

Remarks

- The **T7P3c** is a conforming 7-node mixed, isoparametric triangular element [52].
- The element
  - Contains three (3) vertex nodes.
  - Contains three (3) mid-side nodes.
  - Contains one (1) interior node. Associated with this node is a *cubic* interpolation function that "enhances" the element.
  - Has two (2) displacements degrees of freedom at each node, for a total of fourteen (14) displacement degrees of freedom.
  - Employs a *enhanced qudratic* approximation for the displacement field.
  - Employs a *linear*, continuous approximation for the pressure (Figure 28.3).
  - Has a total of three (3) pressure degrees of freedom.
- The **T7P3c** element should be used for coupled displacement-flow analyses of porous media. The element is based on a generalized Biot formulation.

Syntax

The following syntax is used to describe a typical **T7P3c** mixed-flow triangular continuum element:

ELEment MIXed_Flow TYPe T7P3c (DEScription "string")
NODes #:#:# (MATerial #) (HYDraulic #.#) (INItial #)
(CONstruction #) (EXCavation #) (THIckness #.#)
(FLUID_Density #) (SOLID_Density #) (FLUID_Bulk #) (SOLID_Bulk #)
(INCompressible_constituents)
(1_Additional #) (1_Increment #)

(2_Additional #) (2_Increment #)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLumetric_strain) (PRINT_AVG_PREssure) (PRINT_VELocity)

Explanatory Notes

- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENTS OFF** or the **ECHO ELEMENT_DESCRIPTIONS OFF** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with the **T7P3c** element is shown in Figure 28.3. For this element the nodes are numbered sequentially from 1 to 7.

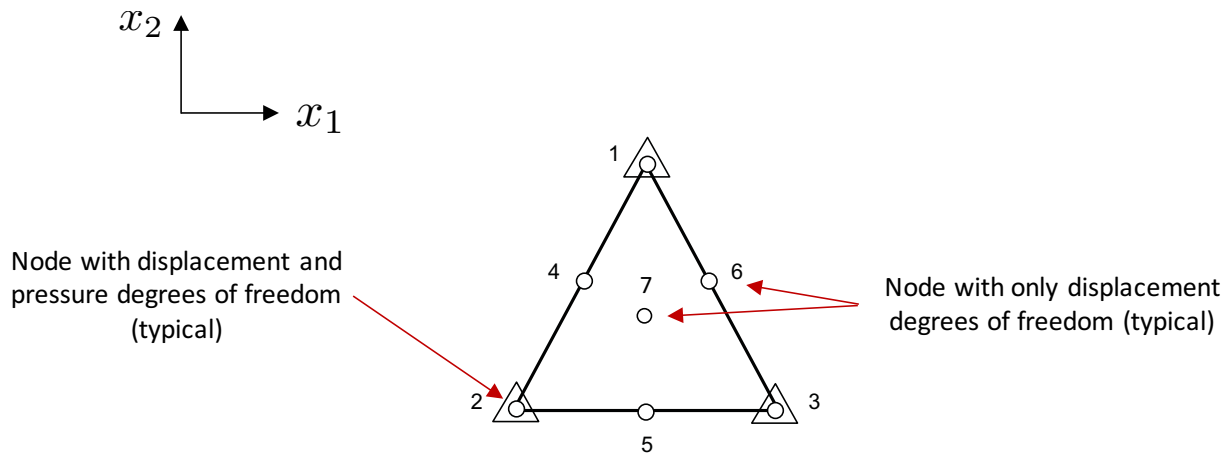

Figure 28.3: Node numbering associated with a typical mixed 7-node quadratic (T7P3c) triangular continuum element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default value for the **MATERIAL** number is one (1).
- The **HYDRAULIC** keyword is used to specify the number of the hydraulic conductivity (permeability) idealization associated with the element. In order to maximize flexibility, the **HYDRAULIC** number differs from the **MATERIAL** number. The default value for the **HYDRAULIC** conductivity number is one (1).
- The **INITIAL** keyword is used to specify the initial state number associated with the element. The *default* value for the **INITIAL** is zero (0).

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to, or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the *default* condition, no input is required in such a case. The condition of no excavation is likewise the default. In the current version of APES$^+$ the hydraulic boundary condition at the temporary top of a soil mass that is being constructed is *zero flow*; i.e., an *impervious* surface. The conditions on top of the final surface can, however, be specified through the use of the **SPECIFICATIONS CONCENTRATED FLOW** command.

- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The *default* **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE STRAIN** idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.

- The **FLUID_DENSITY** keyword is used to specify the number of the mass density idealization for the fluid phase (ρ^f) (units of ML^{-3} or $\mathrm{FL}^{-4}\mathrm{t}^2$) associated with the element. The **SOLID_DENSITY** keyword is used to specify the number of the mass density idealization for the solid phase (ρ^s) associated with the element.

  Given the values of ρ^f and ρ^s (specified using the **DENSITY_FLUID** and **DENSITY_SOLID** commands, respectively, as described in Chapter 17, the total mass density of a saturated soil is computed from

  $$\rho = (1-n)\rho^s + n\rho^f$$

  where n is the porosity, which is related to the void ratio (e) by $n = e/(1+e)$.

- The **FLUID_BULK** keyword is used to specify the number of the bulk modulus idealization for the fluid phase (K_f) (units of FL^{-2}) associated with the element. The **SOLID_BULK** keyword is used to specify the number of the bulk modulus idealization for the solid phase (K_s) associated with the element.

  Given the values of K_f and K_s (specified using the **BULK_FLUID** and **BULK_SOLID** commands, respectively, as described in Chapter 16, the value of the "Biot" modulus [27] is computed as follows:

  $$\frac{1}{M} = \frac{n}{K_f} + \frac{1-n}{K_s}$$

  where n is again the porosity. An explicit value of M can be input using the **MATERIAL POROELASTIC ANISOTROPIC** command.

- If the **INCompressible_constituents** keyword is used, both the solid skeleton and the pore fluid are assumed to be *incompressible*. This supersedes any bulk modulus values specified in conjunction with the **MATERIAL FLUID** or **MATERIAL SOLID**

commands. If an anisotropic poroelastic material idealization is used (refer to the description of the **MATERIAL POROELASTIC ANISOTROPIC** command), this implies an *infinite* value for the "Biot modulus" M (specified via the **M_MODULUS** keyword) for the element.

- If the body being analyzed can be divided into a layer (or layers) of elements, and if the characteristics of the element (i.e., the **MATERIAL**, the **INITIAL** state, the incremental **CONSTRUCTION** and **EXCAVATION** numbers, and the **THICKNESS** value) are the *same* for several elements within a layer, and if the nodes are numbered in a *consistent* fashion, then an element data generation option can be employed. To generate a sequence of elements within a single layer, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**.

  If several layers of elements have the *same* attributes, the above generation option can be carried one step further by making use of the **2_ADDITIONAL**, and **2_INCREMENT** keywords.

- The purpose of the **PRINT** commands is to eliminate unnecessary output (printed and saved) generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the *default* condition both strains are printed.

- Specification of the **DONT_PRINT_STRESSES** keyword indicates that stresses are not to be printed. Under the *default* condition stresses are printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the *default* condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the *default* condition these quantities are not computed and printed.

- The **PRINT_AVG_STRAINS** keyword indicates that average strains (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the *default* condition average strains are not computed and printed.

- The **PRINT_AVG_STRESSES** keyword indicates that average stresses (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the *default* condition average stresses are not computed and printed.

- The **PRINT_VOLUMETRIC_STRAIN** keyword causes the volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the volumetric strain to the absolute value of the minimum non-zero normal strain in the element is printed. That is,

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33}) \neq 0$$

  This ratio is instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the *default* condition the volumetric strain and the aforementioned ratio are *not* computed and printed.

- The **PRINT_AVG_VOLUMETRIC_STRAIN** keyword causes the *average* volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the *average* volumetric strain to the absolute value of the minimum *average* non-zero normal strain in the element is printed. This ratio is also instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the *default* condition the average volumetric strain and the aforementioned ratio are *not* computed and printed.

- The **PRINT_AVG_PRESSURE** keyword causes the *average* pressure to be computed and printed for the element. This value represents the average of the approximate pressures at the four pressure (vertex) nodes.

- The **PRINT_VELOCITY** keyword causes the components of the Darcy velocity in the global x_1- and x_2-coordinate directions to be computed and printed at the element center.

28.4 The ELEMENT MIXED_FLOW T15P6c Command

Synopsis

The **ELEMENT MIXED_FLOW T15P6c** command is used to describe all mixed, 15-node, quartic triangular continuum elements that are to be used in a coupled flow-mechanical analysis.

Remarks

- The **T15P6c** is a conforming 15-node mixed, isoparametric triangular element [52].
- The element
  - Contains three (3) vertex nodes.
  - Contains nine (9) mid-side nodes.
  - Has two (2) displacements degrees of freedom at each node, for a total of thirty (30) displacement degrees of freedom.
  - Employs a *quartic* approximation for the displacement field.
  - Employs a *quadratic*, continuous approximation for the pressure (Fig. 28.4).
  - Has a total of six (6) pressure degrees of freedom.
- The **T15P6c** element should be used for coupled displacement-flow analyses of porous media. The element is based on a generalized Biot formulation.

Syntax

The following syntax is used to describe a typical **T15P6c** mixed-flow triangular continuum element:

ELEment MIXed_Flow TYPe T15P6c (DEScription "string")
NODes #:#:# (MATerial #) (HYDraulic #.#) (INItial #)
(CONstruction #) (EXCavation #) (THIckness #.#)
(FLUID_Density #) (SOLID_Density #) (FLUID_Bulk #) (SOLID_Bulk #)
(INCompressible_constituents)
(1_Additional #) (1_Increment #)
(2_Additional #) (2_Increment #)

(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLumetric_strain) (PRINT_AVG_PREssure) (PRINT_VELocity)

Explanatory Notes

- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENTS OFF** or the **ECHO ELEMENT_DESCRIPTIONS OFF** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with the **T15P6c** element is shown in Figure 28.4. For this element the nodes are numbered sequentially from 1 to 15.

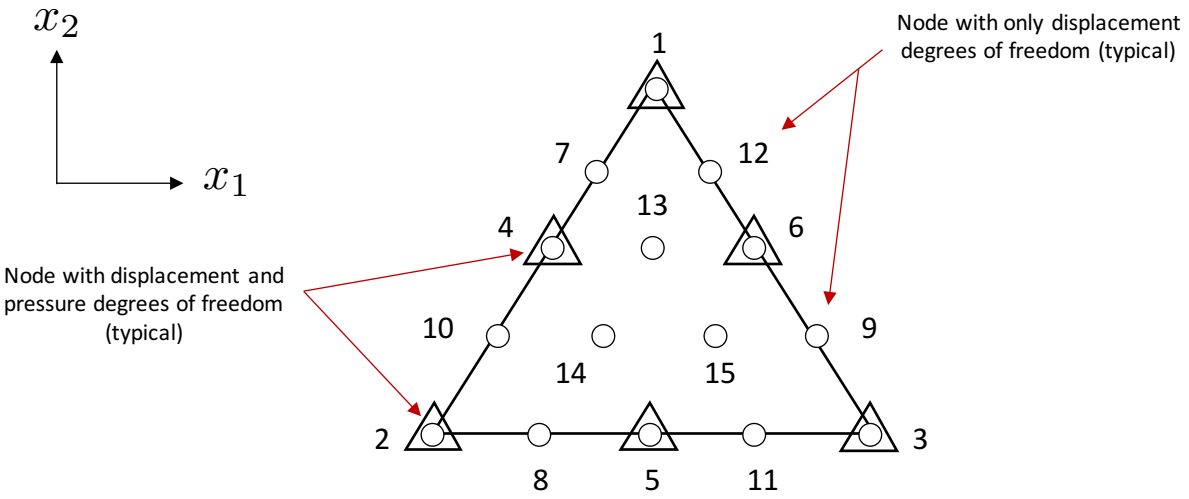

Figure 28.4: Node numbering associated with a typical mixed 15-node quartic (T15P6c) triangular continuum element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default value for the **MATERIAL** number is one (1).
- The **HYDRAULIC** keyword is used to specify the number of the hydraulic conductivity (permeability) idealization associated with the element. In order to maximize flexibility, the **HYDRAULIC** number differs from the **MATERIAL** number. The default value for the **HYDRAULIC** conductivity number is one (1).

- The **INITIAL** keyword is used to specify the initial state number associated with the element. The *default* value for the **INITIAL** is zero (0).

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to, or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the *default* condition, no input is required in such a case. The condition of no excavation is likewise the default. In the current version of APES$^+$ the hydraulic boundary condition at the temporary top of a soil mass that is being constructed is *zero flow*; i.e., an *impervious* surface. The conditions on top of the final surface can, however, be specified through the use of the **SPECIFICATIONS CONCENTRATED FLOW** command.

- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The *default* **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE STRAIN** idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.

- The **FLUID_DENSITY** keyword is used to specify the number of the mass density idealization for the fluid phase (ρ^f) (units of ML^{-3} or $\mathrm{FL}^{-4}\mathrm{t}^2$) associated with the element. The **SOLID_DENSITY** keyword is used to specify the number of the mass density idealization for the solid phase (ρ^s) associated with the element.

  Given the values of ρ^f and ρ^s (specified using the **DENSITY_FLUID** and **DENSITY_SOLID** commands, respectively, as described in Chapter 17, the total mass density of a saturated soil is computed from

  $$\rho = (1-n)\rho^s + n\rho^f$$

  where n is the porosity, which is related to the void ratio (e) by $n = e/(1+e)$.

- The **FLUID_BULK** keyword is used to specify the number of the bulk modulus idealization for the fluid phase (K_f) (units of FL^{-2}) associated with the element. The **SOLID_BULK** keyword is used to specify the number of the bulk modulus idealization for the solid phase (K_s) associated with the element.

  Given the values of K_f and K_s (specified using the **BULK_FLUID** and **BULK_SOLID** commands, respectively, as described in Chapter 16, the value of the "Biot" modulus [27] is computed as follows:

  $$\frac{1}{M} = \frac{n}{K_f} + \frac{1-n}{K_s}$$

  where n is again the porosity. An explicit value of M can be input using the **MATERIAL POROELASTIC ANISOTROPIC** command.

- If the **INCompressible_constituents** keyword is used, both the solid skeleton and the pore fluid are assumed to be *incompressible*. This supersedes any bulk modulus values specified in conjunction with the **MATERIAL FLUID** or **MATERIAL SOLID** commands. If an anisotropic poroelastic material idealization is used (refer to the description of the **MATERIAL POROELASTIC ANISOTROPIC** command), this implies an *infinite* value for the "Biot modulus" M (specified via the **M_MODULUS** keyword) for the element.

- If the body being analyzed can be divided into a layer (or layers) of elements, and if the characteristics of the element (i.e., the **MATERIAL**, the **INITIAL** state, the incremental **CONSTRUCTION** and **EXCAVATION** numbers, and the **THICKNESS** value) are the *same* for several elements within a layer, and if the nodes are numbered in a *consistent* fashion, then an element data generation option can be employed. To generate a sequence of elements within a single layer, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**.

  If several layers of elements have the *same* attributes, the above generation option can be carried one step further by making use of the **2_ADDITIONAL**, and **2_INCREMENT** keywords.

- The purpose of the **PRINT** commands is to eliminate unnecessary output (printed and saved) generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the *default* condition both strains are printed.

- Specification of the **DONT_PRINT_STRESSES** keyword indicates that stresses are not to be printed. Under the *default* condition stresses are printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the *default* condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the *default* condition these quantities are not computed and printed.

- The **PRINT_AVG_STRAINS** keyword indicates that average strains (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the *default* condition average strains are not computed and printed.

- The **PRINT_AVG_STRESSES** keyword indicates that average stresses (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the *default* condition average stresses are not computed and printed.

- The **PRINT_VOLUMETRIC_STRAIN** keyword causes the volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the volumetric strain to the absolute value of the minimum non-zero normal strain in the element is printed. That is,

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33}) \neq 0$$

  This ratio is instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the *default* condition the volumetric strain and the aforementioned ratio are *not* computed and printed.

- The **PRINT_AVG_VOLUMETRIC_STRAIN** keyword causes the *average* volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the *average* volumetric strain to the absolute value of the minimum *average* non-zero normal strain in the element is printed. This ratio is also instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the *default* condition the average volumetric strain and the aforementioned ratio are *not* computed and printed.

- The **PRINT_AVG_PRESSURE** keyword causes the *average* pressure to be computed and printed for the element. This value represents the average of the approximate pressures at the four pressure (vertex) nodes.

- The **PRINT_VELOCITY** keyword causes the components of the Darcy velocity in the global x_1- and x_2-coordinate directions to be computed and printed at the element center.

28.5 The ELEMENT MIXED_FLOW Q5P4c Command

Synopsis

The **ELEMENT MIXED_FLOW Q5P4c** command is used to describe all conforming 5-node mixed "enhanced" bilinear quadrilateral continuum elements with a bi-linear continuous pressure approximation.

Remarks

- The **Q5P4c** is a mixed, isoparametric quadrilateral element [52].
- The element
  - Contains four (4) vertex nodes.
  - Contains one (1) interior node. Associated with this node is a *quadratic* interpolation function that "enhances" the element.
  - Has two (2) displacements degrees of freedom at each node, for a total of ten (10) displacement degrees of freedom.
  - Employs a *bilinear*, continuous approximation for the pressure (Fig. 28.5).
  - Has a total of four (4) pressure degrees of freedom.
- The **Q5P4c** element should be used for coupled displacement-flow analyses of porous media. The element is based on a generalized Biot formulation and employs a $u - p$ form of the governing equations.

Syntax

The following syntax is used to describe a typical **Q5P4c** mixed-flow quadrilateral continuum element:

ELEment MIXed_Flow TYPe Q5P4c (DEScription "string")
NODes #:#:# (MATerial #) (HYDraulic #.#) (INItial #)
(CONstruction #) (EXCavation #) (THIckness #.#)
(FLUID_Density #) (SOLID_Density #) (FLUID_Bulk #) (SOLID_Bulk #)
(INCompressible_constituents)
(1_Additional #) (1_Increment #) (2_Additional #) (2_Increment #)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)

(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLumetric_strain) (PRINT_VELocity)

Explanatory Notes

- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENTS OFF** or the **ECHO ELEMENT_DESCRIPTIONS OFF** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with the **Q5P4c** element is shown in Figure 28.5. For this element the nodes are numbered sequentially from 1 to 5.

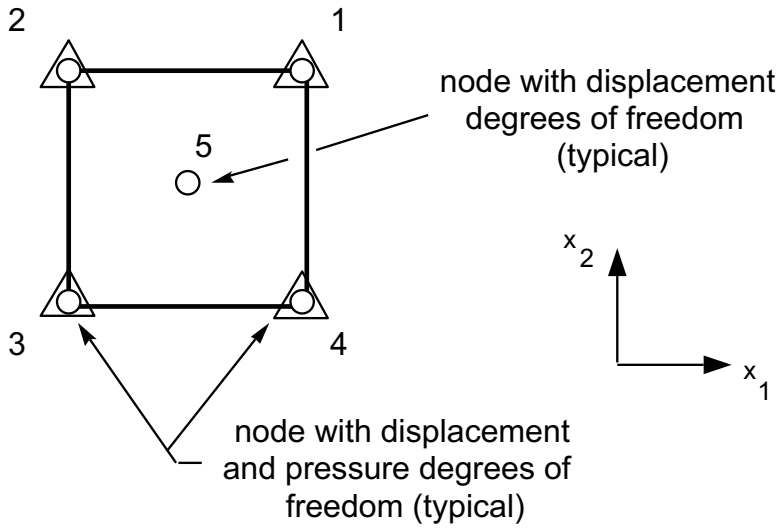

Figure 28.5: Node numbering associated with a typical mixed 5-node "enhanced" bilinear Q5P4c quadrilateral continuum element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default value for the **MATERIAL** number is one (1).
- The **HYDRAULIC** keyword is used to specify the number of the hydraulic conductivity (permeability) idealization associated with the element. In order to maximize flexibility, the **HYDRAULIC** number differs from the **MATERIAL** number. The default value for the **HYDRAULIC** conductivity number is one (1).
- The **INITIAL** keyword is used to specify the initial state number associated with the element. The *default* value for the **INITIAL** is zero (0).

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to, or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the *default* condition, no input is required in such a case. The condition of no excavation is likewise the default. In the current version of APES$^+$ the hydraulic boundary condition at the temporary top of a soil mass that is being constructed is *zero flow*; i.e., an *impervious* surface. The conditions on top of the final surface can, however, be specified through the use of the **SPECIFICATIONS CONCENTRATED FLOW** command.

- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The *default* **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE STRAIN** idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.

- The **FLUID_DENSITY** keyword is used to specify the number of the mass density idealization for the fluid phase (ρ^f) (units of ML^{-3} or $\mathrm{FL}^{-4}\mathrm{t}^2$) associated with the element. The **SOLID_DENSITY** keyword is used to specify the number of the mass density idealization for the solid phase (ρ^s) associated with the element.

  Given the values of ρ^f and ρ^s (specified using the **DENSITY_FLUID** and **DENSITY_SOLID** commands, respectively, as described in Chapter 17, the total mass density of a saturated soil is computed from

  $$\rho = (1-n)\rho^s + n\rho^f$$

  where n is the porosity, which is related to the void ratio (e) by $n = e/(1+e)$.

- The **FLUID_BULK** keyword is used to specify the number of the bulk modulus idealization for the fluid phase (K_f) (units of FL^{-2}) associated with the element. The **SOLID_BULK** keyword is used to specify the number of the bulk modulus idealization for the solid phase (K_s) associated with the element.

  Given the values of K_f and K_s (specified using the **BULK_FLUID** and **BULK_SOLID** commands, respectively, as described in Chapter 16, the value of the "Biot" modulus [27] is computed as follows:

  $$\frac{1}{M} = \frac{n}{K_f} + \frac{1-n}{K_s}$$

  where n is again the porosity. An explicit value of M can be input using the **MATERIAL POROELASTIC ANISOTROPIC** command.

- If the **INCompressible_constituents** keyword is used, both the solid skeleton and the pore fluid are assumed to be *incompressible*. This supersedes any bulk modulus values specified in conjunction with the **MATERIAL FLUID** or **MATERIAL SOLID**

commands. If an anisotropic poroelastic material idealization is used (refer to the description of the **MATERIAL POROELASTIC ANISOTROPIC** command), this implies an *infinite* value for the "Biot modulus" M (specified via the **M_MODULUS** keyword) for the element.

- If the body being analyzed can be divided into a layer (or layers) of elements, and if the characteristics of the element (i.e., the **MATERIAL**, the **INITIAL** state, the incremental **CONSTRUCTION** and **EXCAVATION** numbers, and the **THICKNESS** value) are the *same* for several elements within a layer, and if the nodes are numbered in a *consistent* fashion, then an element data generation option can be employed. To generate a sequence of elements within a single layer, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**.

  If several layers of elements have the *same* attributes, the above generation option can be carried one step further by making use of the **2_ADDITIONAL**, and **2_INCREMENT** keywords.

- The purpose of the **PRINT** commands is to eliminate unnecessary output (printed and saved) generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the *default* condition both strains are printed.

- Specification of the **DONT_PRINT_STRESSES** keyword indicates that stresses are not to be printed. Under the *default* condition stresses are printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the *default* condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the *default* condition these quantities are not computed and printed.

- The **PRINT_AVG_STRAINS** keyword indicates that average strains (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the *default* condition average strains are not computed and printed.

- The **PRINT_AVG_STRESSES** keyword indicates that average stresses (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the *default* condition average stresses are not computed and printed.

- The keyword **PRINT_VOLUMETRIC_STRAIN** causes the volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the volumetric strain to the absolute value of the minimum non-zero normal strain in the element is printed. That is,

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33}) \neq 0$$

  This ratio is instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the *default* condition the volumetric strain and the aforementioned ratio are *not* computed and printed.

- The **PRINT_AVG_VOLUMETRIC_STRAIN** keyword causes the *average* volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the *average* volumetric strain to the absolute value of the minimum *average* non-zero normal strain in the element is printed. This ratio is also instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the *default* condition the average volumetric strain and the aforementioned ratio are *not* computed and printed.

- The **PRINT_AVG_PRESSURE** keyword causes the *average* pressure to be computed and printed for the element. This value represents the average of the approximate pressures at the four pressure (vertex) nodes.

- The **PRINT_VELOCITY** keyword causes the components of the Darcy velocity in the global x_1- and x_2-coordinate directions to be computed and printed at the element center.

28.6 The ELEMENT MIXED_FLOW Q8P4c Command

Synopsis

The **ELEMENT MIXED_FLOW Q8P4c** command is used to describe all mixed, 8-node, quadratic quadrilateral continuum elements with a bi-linear *continuous* pressure approximation.

Remarks

- The **Q8P4c** is a conforming 8-node mixed, quadrilateral isoparametric "serendipity" continuum element [52].
- The element
  - Contains four (4) vertex nodes.
  - Contains four (4) mid-side nodes.
  - Has two (2) displacements degrees of freedom at each node, for a total of sixteen (16) displacement degrees of freedom.
  - Employs a *quadratic* approximation for the displacement field.
  - Employs a *linear*, continuous approximation for the pressure (Fig. 28.6).
  - Has a total of four (4) pressure degrees of freedom.
- The **Q8P4c** element should be used for coupled displacement-flow analyses of porous media. The element is based on a generalized Biot formulation.
- The **Q8P4c** element does *not* satisfy the Babuška-Brezzi condition [44].

Syntax

The following syntax is used to describe a typical **Q8P4c** mixed-flow quadrilateral continuum element:

ELEment MIXed_Flow TYPe Q8P4c (DEScription "string")
NODes #:#:# (MATerial #) (HYDraulic #.#) (INItial #)
(CONstruction #) (EXCavation #) (THIckness #.#)
(FLUID_Density #) (SOLID_Density #) (FLUID_Bulk #) (SOLID_Bulk #)

(INCompressible_constituents)
(1_Additional #) (1_Increment #) (2_Additional #) (2_Increment #)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_AVG_STRAins) (PRINT_AVG_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLumetric_strain) (PRINT_AVG_VOLumetric_strain)
(PRINT_AVG_PREssure) (PRINT_VELocity)

Explanatory Notes

- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENTS OFF** or the **ECHO ELEMENT_DESCRIPTIONS OFF** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with the **Q8P4c** element is shown in Figure 28.6. For this element the nodes are numbered sequentially from 1 to 8.

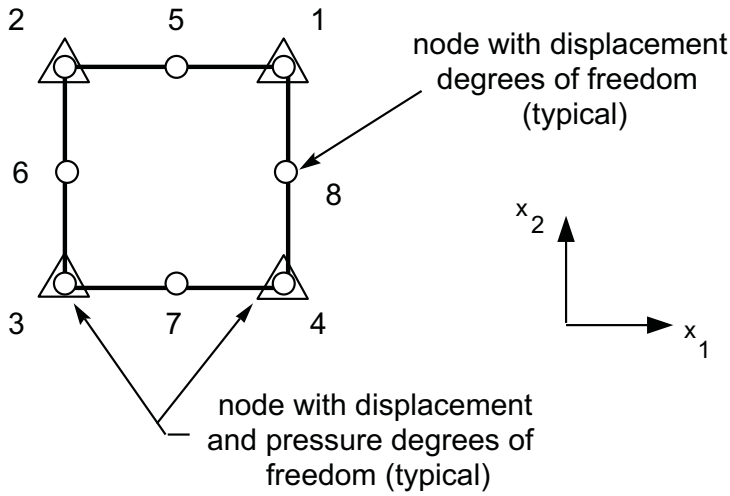

Figure 28.6: Node numbering associated with a typical mixed 8-node quadratic (Q8P4c) quadrilateral continuum element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default value for the **MATERIAL** number is one (1).
- The **HYDRAULIC** keyword is used to specify the number of the hydraulic conductivity (permeability) idealization associated with the element. In order to maximize

flexibility, the **HYDRAULIC** number differs from the **MATERIAL** number. The default value for the **HYDRAULIC** conductivity number is one (1).

- The **INITIAL** keyword is used to specify the initial state number associated with the element. The *default* value for the **INITIAL** is zero (0).

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to, or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the *default* condition, no input is required in such a case. The condition of no excavation is likewise the default. In the current version of APES$^+$ the hydraulic boundary condition at the temporary top of a soil mass that is being constructed is *zero flow*; i.e., an *impervious* surface. The conditions on top of the final surface can, however, be specified through the use of the **SPECIFICATIONS CONCENTRATED FLOW** command.

- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The *default* **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE STRAIN** idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.

- The **FLUID_DENSITY** keyword is used to specify the number of the mass density idealization for the fluid phase (ρ^f) (units of ML^{-3} or $\mathrm{FL}^{-4}\mathrm{t}^2$) associated with the element. The **SOLID_DENSITY** keyword is used to specify the number of the mass density idealization for the solid phase (ρ^s) associated with the element.

  Given the values of ρ^f and ρ^s (specified using the **DENSITY_FLUID** and **DENSITY_SOLID** commands, respectively, as described in Chapter 17, the total mass density of a saturated soil is computed from

$$\rho = (1-n)\rho^s + n\rho^f$$

  where n is the porosity, which is related to the void ratio (e) by $n = e/(1+e)$.

- The **FLUID_BULK** keyword is used to specify the number of the bulk modulus idealization for the fluid phase (K_f) (units of FL^{-2}) associated with the element. The **SOLID_BULK** keyword is used to specify the number of the bulk modulus idealization for the solid phase (K_s) associated with the element.

  Given the values of K_f and K_s (specified using the **BULK_FLUID** and **BULK_SOLID** commands, respectively, as described in Chapter 16, the value of the "Biot" modulus [27] is computed as follows:

$$\frac{1}{M} = \frac{n}{K_f} + \frac{1-n}{K_s}$$

where n is again the porosity. An explicit value of M can be input using the **MATERIAL POROELASTIC ANISOTROPIC** command.

- If the **INCompressible_constituents** keyword is used, both the solid skeleton and the pore fluid are assumed to be *incompressible*. This supersedes any bulk modulus values specified in conjunction with the **MATERIAL FLUID** or **MATERIAL SOLID** commands. If an anisotropic poroelastic material idealization is used (refer to the description of the **MATERIAL POROELASTIC ANISOTROPIC** command), this implies an *infinite* value for the "Biot modulus" M (specified via the **M_MODULUS** keyword) for the element.

- If the body being analyzed can be divided into a layer (or layers) of elements, and if the characteristics of the element (i.e., the **MATERIAL**, the **INITIAL** state, the incremental **CONSTRUCTION** and **EXCAVATION** numbers, the **THICKNESS**, and the **REDUCED** value) are the *same* for several elements within a layer, and if the nodes are numbered in a *consistent* fashion, then an element data generation option can be employed. To generate a sequence of elements within a single layer, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**.

  If several layers of elements have the *same* attributes, the above generation option can be carried one step further by making use of the **2_ADDITIONAL**, and **2_INCREMENT** keywords.

- The purpose of the **PRINT** commands is to eliminate unnecessary output (printed and saved) generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the *default* condition both strains are printed.

- Specification of the **DONT_PRINT_STRESSES** keyword indicates that stresses are not to be printed. Under the *default* condition stresses are printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the *default* condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the *default* condition these quantities are not computed and printed.

- The **PRINT_AVG_STRAINS** keyword indicates that average strains (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the *default* condition average strains are not computed and printed.

- The **PRINT_AVG_STRESSES** keyword indicates that average stresses (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the *default* condition average stresses are not computed and printed.

- The **PRINT_VOLUMETRIC_STRAIN** keyword causes the volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the volumetric strain to the absolute value of the minimum non-zero normal strain in the element is printed. That is,

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33}) \neq 0$$

  This ratio is instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the *default* condition the volumetric strain and the aforementioned ratio are *not* computed and printed.

- The **PRINT_AVG_VOLUMETRIC_STRAIN** keyword causes the *average* volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the *average* volumetric strain to the absolute value of the minimum *average* non-zero normal strain in the element is printed. This ratio is also instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the *default* condition the average volumetric strain and the aforementioned ratio are *not* computed and printed.

- The **PRINT_AVG_PRESSURE** keyword causes the *average* pressure to be computed and printed for the element. This value represents the average of the approximate pressures at the four pressure (vertex) nodes.

- The **PRINT_VELOCITY** keyword causes the components of the Darcy velocity in the global x_1- and x_2-coordinate directions to be computed and printed at the element center.

28.7 The ELEMENT MIXED_FLOW Q9P4c Command

Synopsis

The **ELEMENT MIXED_FLOW Q9P4c** command is used to describe all mixed, 9-node, quadratic quadrilateral continuum elements with a bilinear *continuous* pressure approximation.

Remarks

- The **Q9P4c** is a conforming 9-node mixed, isoparametric Lagrangian quadrilateral element [52].
- The element
  - Contains four (4) vertex nodes.
  - Contains four (4) mid-side nodes.
  - Contains one (1) interior node.
  - Has two (2) displacements degrees of freedom at each node, for a total of eighteen (18) displacement degrees of freedom.
  - Employs a *bi-quadratic* approximation for the displacement field.
  - Employs a *linear*, continuous approximation for the pressure (Fig. 28.7).
  - Has a total of four (4) pressure degrees of freedom.
- The **Q9P4c** element should be used for coupled displacement-flow analyses of porous media. The element is based on a generalized Biot formulation.
- The **Q9P4c** element satisfies the Babuška-Brezzi condition [44]. For this element $k = 2$ and $l = 1$, meaning that the accuracy is $O(h^2)$ (or the rate of convergence is 2). As such, this is an *optimal* element.

Syntax

The following syntax is used to describe a typical **Q9P4c** mixed-flow quadrilateral continuum element:

ELEment MIXed_Flow TYPe Q9P4c (DEScription "string")
NODes #:#:# (MATerial #) (HYDraulic #.#) (INItial #)
(CONstruction #) (EXCavation #) (THIckness #.#)
(FLUID_Density #) (SOLID_Density #) (FLUID_Bulk #) (SOLID_Bulk #)
(INCompressible_constituents)
(1_Additional #) (1_Increment #) (2_Additional #) (2_Increment #)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_AVG_STRAins) (PRINT_AVG_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLumetric_strain) (PRINT_AVG_VOLumetric_strain)
(PRINT_AVG_PREssure) (PRINT_VELocity)

Explanatory Notes

- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENTS OFF** or the **ECHO ELEMENT_DESCRIPTIONS OFF** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.

- The numbering order of **NODES** associated with the **Q9P4c** element is shown in Figure 28.7. For this element the nodes are numbered sequentially from 1 to 9.

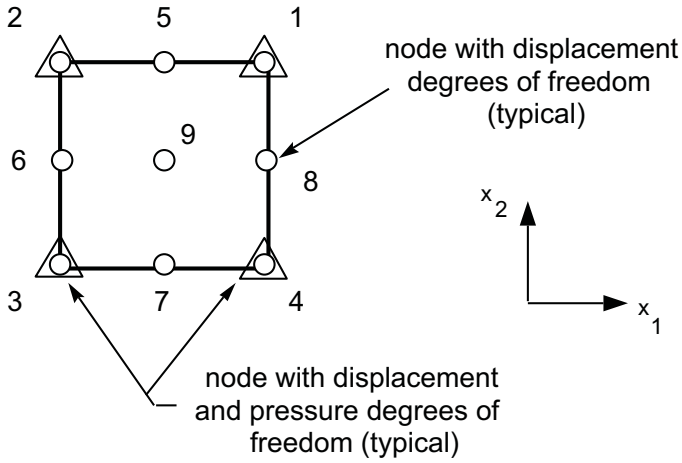

Figure 28.7: Node numbering associated with a typical mixed 9-node quadratic (Q9P4c) quadrilateral continuum element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default value for the **MATERIAL** number is one (1).

- The **HYDRAULIC** keyword is used to specify the number of the hydraulic conductivity (permeability) idealization associated with the element. In order to maximize flexibility, the **HYDRAULIC** number differs from the **MATERIAL** number. The default value for the **HYDRAULIC** conductivity number is one (1).

- The **INITIAL** keyword is used to specify the initial state number associated with the element. The *default* value for the **INITIAL** is zero (0).

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to, or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the *default* condition, no input is required in such a case. The condition of no excavation is likewise the default. In the current version of APES+ the hydraulic boundary condition at the temporary top of a soil mass that is being constructed is *zero flow*; i.e., an *impervious* surface. The conditions on top of the final surface can, however, be specified through the use of the **SPECIFICATIONS CONCENTRATED FLOW** command.

- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The *default* **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE STRAIN** idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.

- The **FLUID_DENSITY** keyword is used to specify the number of the mass density idealization for the fluid phase (ρ^f) (units of ML^{-3} or $\mathrm{FL}^{-4}\mathrm{t}^2$) associated with the element. The **SOLID_DENSITY** keyword is used to specify the number of the mass density idealization for the solid phase (ρ^s) associated with the element.

  Given the values of ρ^f and ρ^s (specified using the **DENSITY_FLUID** and **DENSITY_SOLID** commands, respectively, as described in Chapter 17, the total mass density of a saturated soil is computed from

  $$\rho = (1 - n)\rho^s + n\rho^f$$

  where n is the porosity, which is related to the void ratio (e) by $n = e/(1 + e)$.

- The **FLUID_BULK** keyword is used to specify the number of the bulk modulus idealization for the fluid phase (K_f) (units of FL^{-2}) associated with the element. The **SOLID_BULK** keyword is used to specify the number of the bulk modulus idealization for the solid phase (K_s) associated with the element.

  Given the values of K_f and K_s (specified using the **BULK_FLUID** and **BULK_SOLID** commands, respectively, as described in Chapter 16, the value of the "Biot" modulus [27] is computed as follows:

$$\frac{1}{M} = \frac{n}{K_f} + \frac{1-n}{K_s}$$

where n is again the porosity. An explicit value of M can be input using the **MATERIAL POROELASTIC ANISOTROPIC** command.

- If the **INCompressible_constituents** keyword is used, both the solid skeleton and the pore fluid are assumed to be *incompressible*. This supersedes any bulk modulus values specified in conjunction with the **MATERIAL FLUID** or **MATERIAL SOLID** commands. If an anisotropic poroelastic material idealization is used (refer to the description of the **MATERIAL POROELASTIC ANISOTROPIC** command), this implies an *infinite* value for the "Biot modulus" M (specified via the **M_MODULUS** keyword) for the element.

- If the body being analyzed can be divided into a layer (or layers) of elements, and if the characteristics of the element (i.e., the **MATERIAL**, the **INITIAL** state, the incremental **CONSTRUCTION** and **EXCAVATION** numbers, the **THICKNESS**, and the **REDUCED** value) are the *same* for several elements within a layer, and if the nodes are numbered in a *consistent* fashion, then an element data generation option can be employed. To generate a sequence of elements within a single layer, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**.

  If several layers of elements have the *same* attributes, the above generation option can be carried one step further by making use of the **2_ADDITIONAL**, and **2_INCREMENT** keywords.

- The purpose of the **PRINT** commands is to eliminate unnecessary output (printed and saved) generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the *default* condition both strains are printed.

- Specification of the **DONT_PRINT_STRESSES** keyword indicates that stresses are not to be printed. Under the *default* condition stresses are printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the *default* condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the *default* condition these quantities are not computed and printed.

- The **PRINT_AVG_STRAINS** keyword indicates that average strains (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the *default* condition average strains are not computed and printed.

- The **PRINT_AVG_STRESSES** keyword indicates that average stresses (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the *default* condition average stresses are not computed and printed.

- The **PRINT_VOLUMETRIC_STRAIN** keyword causes the volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the volumetric strain to the absolute value of the minimum non-zero normal strain in the element is printed. That is,

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33}) \neq 0$$

  This ratio is instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the *default* condition the volumetric strain and the aforementioned ratio are *not* computed and printed.

- The **PRINT_AVG_VOLUMETRIC_STRAIN** keyword causes the *average* volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the *average* volumetric strain to the absolute value of the minimum *average* non-zero normal strain in the element is printed. This ratio is also instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the *default* condition the average volumetric strain and the aforementioned ratio are *not* computed and printed.

- The **PRINT_AVG_PRESSURE** keyword causes the *average* pressure to be computed and printed for the element. This value represents the average of the approximate pressures at the four pressure (vertex) nodes.

- The **PRINT_VELOCITY** keyword causes the components of the Darcy velocity in the global x_1- and x_2-coordinate directions to be computed and printed at the element center.

28.8 The ELEMENT MIXED_FLOW Q17P8c Command

Synopsis

The **ELEMENT MIXED_FLOW Q17P8c** command is used to describe all mixed, 17-node, quartic quadrilateral continuum elements that are to be used in a coupled flow-mechanical analysis.

Remarks

- The **Q17P8c** is a conforming 17-node mixed, isoparametric quadrilateral element [52].
- The element
  - Contains four (4) vertex nodes.
  - Contains twelve (12) mid-side nodes.
  - Contains one (1) interior node. Associated with this node is a *cubic* interpolation function that "enhances" the element.
  - Has two (2) displacements degrees of freedom at each node, for a total of fourteen (34) displacement degrees of freedom.
  - Employs a *quartic* approximation for the displacement field.
  - Employs a *quadratic*, continuous approximation for the pressure (Figure 28.8).
  - Has a total of eight (8) pressure degrees of freedom.
- The **Q17P8c** element should be used for coupled displacement-flow analyses of porous media. The element is based on a generalized Biot formulation.

Syntax

The following syntax is used to describe a typical **Q17P8c** mixed-flow quadrilateral continuum element:

ELEment MIXed_Flow TYPe Q17P8c (DEScription "string")
NODes #:#:# (MATerial #) (HYDraulic #.#) (INItial #)
(CONstruction #) (EXCavation #) (THIckness #.#)
(FLUID_Density #) (SOLID_Density #) (FLUID_Bulk #) (SOLID_Bulk #)
(INCompressible_constituents)

(1_Additional #) (1_Increment #)
(2_Additional #) (2_Increment #)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLumetric_strain) (PRINT_AVG_PREssure) (PRINT_VELocity)

Explanatory Notes

- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENTS OFF** or the **ECHO ELEMENT_DESCRIPTIONS OFF** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.

- The numbering order of **NODES** associated with the **Q17P8c** element is shown in Figure 28.8. For this element the nodes are numbered sequentially from 1 to 17.

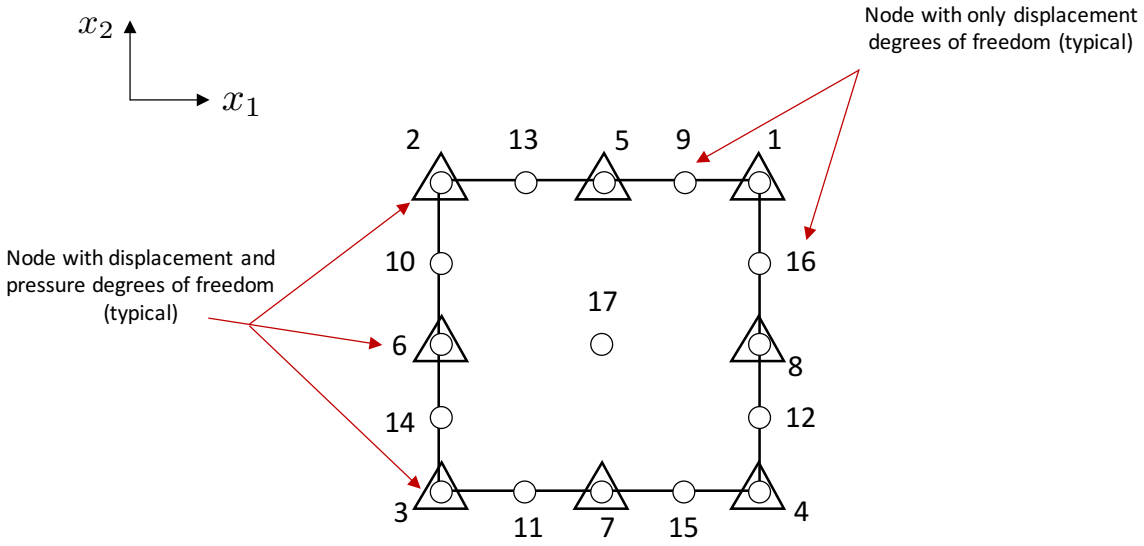

Figure 28.8: Node numbering associated with a typical mixed 17-node quadratic (Q17P8c) quadrilateral continuum element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default value for the **MATERIAL** number is one (1).

- The **HYDRAULIC** keyword is used to specify the number of the hydraulic conductivity (permeability) idealization associated with the element. In order to maximize flexibility, the **HYDRAULIC** number differs from the **MATERIAL** number. The default value for the **HYDRAULIC** conductivity number is one (1).

- The **INITIAL** keyword is used to specify the initial state number associated with the element. The *default* value for the **INITIAL** is zero (0).

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to, or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the *default* condition, no input is required in such a case. The condition of no excavation is likewise the default. In the current version of APES+ the hydraulic boundary condition at the temporary top of a soil mass that is being constructed is *zero flow*; i.e., an *impervious* surface. The conditions on top of the final surface can, however, be specified through the use of the **SPECIFICATIONS CONCENTRATED FLOW** command.

- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The *default* **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE STRAIN** idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.

- The **FLUID_DENSITY** keyword is used to specify the number of the mass density idealization for the fluid phase (ρ^f) (units of ML^{-3} or $\mathrm{FL}^{-4}\mathrm{t}^2$) associated with the element. The **SOLID_DENSITY** keyword is used to specify the number of the mass density idealization for the solid phase (ρ^s) associated with the element.

  Given the values of ρ^f and ρ^s (specified using the **DENSITY_FLUID** and **DENSITY_SOLID** commands, respectively, as described in Chapter 17, the total mass density of a saturated soil is computed from

  $$\rho = (1 - n)\rho^s + n\rho^f$$

  where n is the porosity, which is related to the void ratio (e) by $n = e/(1 + e)$.

- The **FLUID_BULK** keyword is used to specify the number of the bulk modulus idealization for the fluid phase (K_f) (units of FL^{-2}) associated with the element. The **SOLID_BULK** keyword is used to specify the number of the bulk modulus idealization for the solid phase (K_s) associated with the element.

  Given the values of K_f and K_s (specified using the **BULK_FLUID** and **BULK_SOLID** commands, respectively, as described in Chapter 16, the value of the "Biot" modulus [27] is computed as follows:

$$\frac{1}{M} = \frac{n}{K_f} + \frac{1-n}{K_s}$$

where n is again the porosity. An explicit value of M can be input using the **MATERIAL POROELASTIC ANISOTROPIC** command.

- If the **INCompressible_constituents** keyword is used, both the solid skeleton and the pore fluid are assumed to be *incompressible*. This supersedes any bulk modulus values specified in conjunction with the **MATERIAL FLUID** or **MATERIAL SOLID** commands. If an anisotropic poroelastic material idealization is used (refer to the description of the **MATERIAL POROELASTIC ANISOTROPIC** command), this implies an *infinite* value for the "Biot modulus" M (specified via the **M_MODULUS** keyword) for the element.

- If the body being analyzed can be divided into a layer (or layers) of elements, and if the characteristics of the element (i.e., the **MATERIAL**, the **INITIAL** state, the incremental **CONSTRUCTION** and **EXCAVATION** numbers, and the **THICKNESS** value) are the *same* for several elements within a layer, and if the nodes are numbered in a *consistent* fashion, then an element data generation option can be employed. To generate a sequence of elements within a single layer, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**.

  If several layers of elements have the *same* attributes, the above generation option can be carried one step further by making use of the **2_ADDITIONAL**, and **2_INCREMENT** keywords.

- The purpose of the **PRINT** commands is to eliminate unnecessary output (printed and saved) generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the *default* condition both strains are printed.

- Specification of the **DONT_PRINT_STRESSES** keyword indicates that stresses are not to be printed. Under the *default* condition stresses are printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the *default* condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the *default* condition these quantities are not computed and printed.

- The **PRINT_AVG_STRAINS** keyword indicates that average strains (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the *default* condition average strains are not computed and printed.

- The **PRINT_AVG_STRESSES** keyword indicates that average stresses (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the *default* condition average stresses are not computed and printed.

- The **PRINT_VOLUMETRIC_STRAIN** keyword causes the volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the volumetric strain to the absolute value of the minimum non-zero normal strain in the element is printed. That is,

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33}) \neq 0$$

  This ratio is instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the *default* condition the volumetric strain and the aforementioned ratio are *not* computed and printed.

- The **PRINT_AVG_VOLUMETRIC_STRAIN** keyword causes the *average* volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the *average* volumetric strain to the absolute value of the minimum *average* non-zero normal strain in the element is printed. This ratio is also instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the *default* condition the average volumetric strain and the aforementioned ratio are *not* computed and printed.

- The **PRINT_AVG_PRESSURE** keyword causes the *average* pressure to be computed and printed for the element. This value represents the average of the approximate pressures at the four pressure (vertex) nodes.

- The **PRINT_VELOCITY** keyword causes the components of the Darcy velocity in the global x_1- and x_2-coordinate directions to be computed and printed at the element center.

28.9 The ELEMENT MIXED_FLOW H9P8c Command

Synopsis

The **ELEMENT MIXED_FLOW H9P8c** command is used to describe all mixed, 9-node, "enhanced" trilinear hexahedral continuum elements with a trilinear *continuous* pressure approximation.

Remarks

- The **H9P8c** is a mixed, isoparametric hexahedral element [52].
- The element
  - Contains eight (8) vertex nodes.
  - Contains one (1) interior node. Associated with this node is a *quadratic* interpolation function that "enhances" the element.
  - Has three (3) displacements degrees of freedom at each node, for a total of twenty-seven (27) displacement degrees of freedom.
  - Employs a *trilinear* approximation for the displacement field.
  - Employs a *trilinear*, continuous approximation for the pressure (Fig. 28.9).
  - Has a total of eight (8) pressure degrees of freedom.
- The **H9P8c** element should be used for coupled displacement-flow analyses of porous media. The element is based on a generalized Biot formulation and employs a $u - p$ form of the governing equations.

Syntax

The following syntax is used to describe a typical **H9P8c** mixed-flow hexahedral continuum element:

ELEment MIXed_Flow TYPe H9P8c (DEScription "string")
NODes #:#:# (MATerial #) (HYDraulic #.#) (INItial #)
(CONstruction #) (EXCavation #)
(FLUID_Density #) (SOLID_Density #) (FLUID_Bulk #) (SOLID_Bulk #)
(INCompressible_constituents) (REDuced #.#)
(1_Additional #) (1_Increment #)
(2_Additional #) (2_Increment #)

(3_Additional #) (3_Increment #)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLumetric_strain) (PRINT_VELocity)

Explanatory Notes

- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with **H9P8c** elements, which must be specified sequentially from 1 to 9, is shown in Figure 28.9.

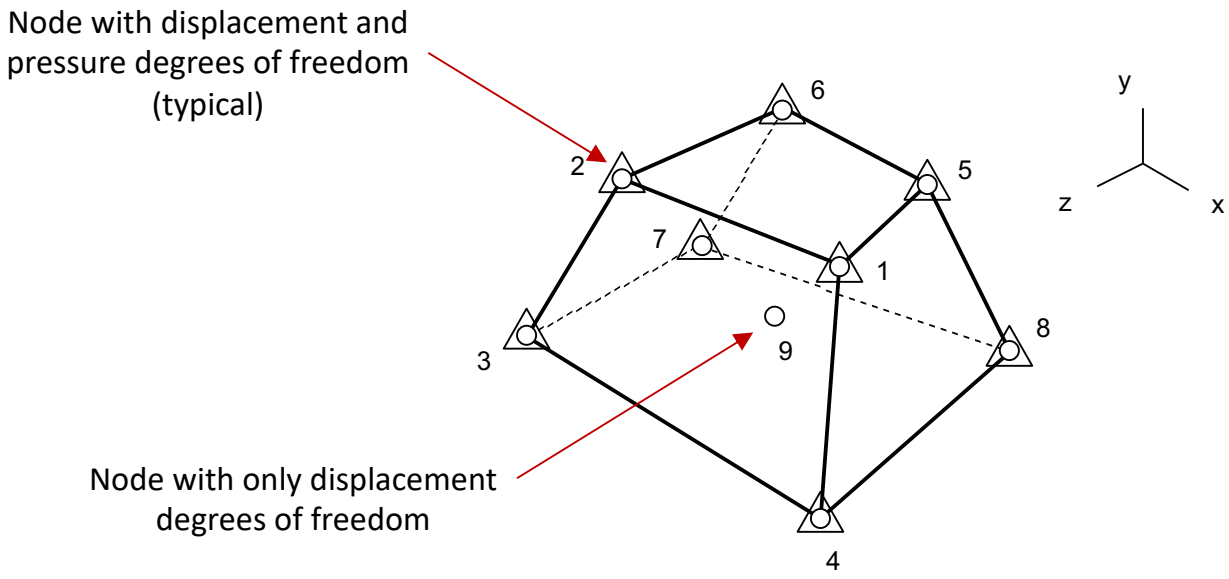

Figure 28.9: Node numbering associated with a typical mixed 9-node "Enhanced" trilinear (H9P8c) hexahedral continuum element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default value for the **MATERIAL** number is one (1).
- The **HYDRAULIC** keyword is used to specify the number of the hydraulic conductivity (permeability) idealization associated with the element. In order to maximize flexibility, the **HYDRAULIC** number differs from the **MATERIAL** number. The default value for the **HYDRAULIC** conductivity number is one (1).

- The **INITIAL** keyword is used to specify the initial state number associated with the element. The *default* value for the **INITIAL** is zero (0).

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the *default* condition, no input is required in such a case. The condition of no excavation is likewise the default. In the current version of APES+ the hydraulic boundary condition at the temporary top of a soil mass that is being constructed is *zero flow*; i.e., an *impervious* surface. The conditions on top of the final surface can, however, be specified through the use of the **SPECIFICATIONS CONCENTRATED FLOW** command.

- If the **INCompressible_constituents** keyword is used, both the solid skeleton and the pore fluid are assumed to be *incompressible*. This supersedes any bulk modulus values specified in conjunction with the **MATERIAL FLUID** or **MATERIAL SOLID** commands. If an anisotropic poroelastic material idealization is used (refer to the description of the **MATERIAL POROELASTIC ANISOTROPIC** command), this implies an *infinite* value for the "Biot modulus" M (specified via the **M_MODULUS** keyword) for the element.

- The value specified in conjunction with the **REDUCED** keyword determines whether *hybrid* selective reduced integration shall be used in the determination of the "coupling" submatrix associated with the mixed penalty elements. What this actually entails is replacing the interpolation functions approximating the element pressure field in the "coupling" submatrix with the following expression:

$$N_i^{(p)*} = \left[\alpha N_i^{(p)} + \frac{1}{8}(1-\alpha) \right] \; ; \; i = 1, N_{en}$$

  where α represents the value specified in conjunction with the **REDUCED** command, $N_i^{(p)}$ represents the bi-linear pressure interpolation function associated with node i, and N_{en} denotes the number of element nodes (= 8). For $\alpha = 0$, the above expression reduces to a value of 1/8, which corresponds to $N_i^{(p)}$ evaluated at the element center; i.e., at the *reduced* quadrature point for either the **H8P8c** of **H9P8c** elements. For $\alpha = 1.0$, the above expression simply yields $N_i^{(p)}$, with the quadrature points coinciding with the standard ones.

- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the *default* condition both strains are printed.

- Specification of the **DONT_PRINT_STRESSES** keyword indicates that stresses are not to be printed. Under the *default* condition stresses are printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the *default* condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the *default* condition these quantities are not computed and printed.

- The **PRINT_VOLUMETRIC_STRAIN** keyword causes the volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the volumetric strain to the absolute value of the minimum non-zero normal strain in the element is printed. That is,

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33}) \neq 0$$

  This ratio is instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the *default* condition the volumetric strain and the aforementioned ratio are not computed and printed.

- The **PRINT_VELOCITY** keyword causes the components of the Darcy velocity in the global x_1- and x_2-coordinate directions to be computed and printed at the element center.

28.10 The ELEMENT MIXED_FLOW H20P8c Command

Synopsis

The **ELEMENT MIXED_FLOW H20P8c** command is used to describe all mixed, 20-node, triquadratic isoparametric "serendipity" hexahedral continuum elements with a trilinear *continuous* pressure approximation.

Remarks

- The **H20P8c** is a mixed, isoparametric "serendipity" hexahedral element [52].
- The element
  - Contains eight (8) vertex nodes.
  - Contains twelve (12) midside nodes.
  - Has three (3) displacements degrees of freedom at each node, for a total of sixty (60) displacement degrees of freedom.
  - Employs a *triquadratic* approximation for the displacement field.
  - Employs a *trilinear*, continuous approximation for the pressure (Fig. 28.10).
  - Has a total of eight (8) pressure degrees of freedom.
- The **H20P8c** element should be used for coupled displacement-flow analyses of porous media. The element is based on a generalized Biot formulation and employs a $u - p$ form of the governing equations.

Syntax

The following syntax is used to describe a typical **H20P8c** mixed-flow hexahedral continuum element:

ELEment MIXed_Flow TYPe H20P8c (DEScription "string")
NODes #:#:# (MATerial #) (HYDraulic #.#) (INItial #)
(CONstruction #) (EXCavation #)
(FLUID_Density #) (SOLID_Density #) (FLUID_Bulk #) (SOLID_Bulk #)
(INCompressible_constituents)
(1_Additional #) (1_Increment #)

(2_Additional #) (2_Increment #)
(3_Additional #) (3_Increment #)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLumetric_strain) (PRINT_VELocity)

Explanatory Notes

- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with **H20P8c** elements, which must be specified sequentially from 1 to 20, is shown in Figure 28.10.

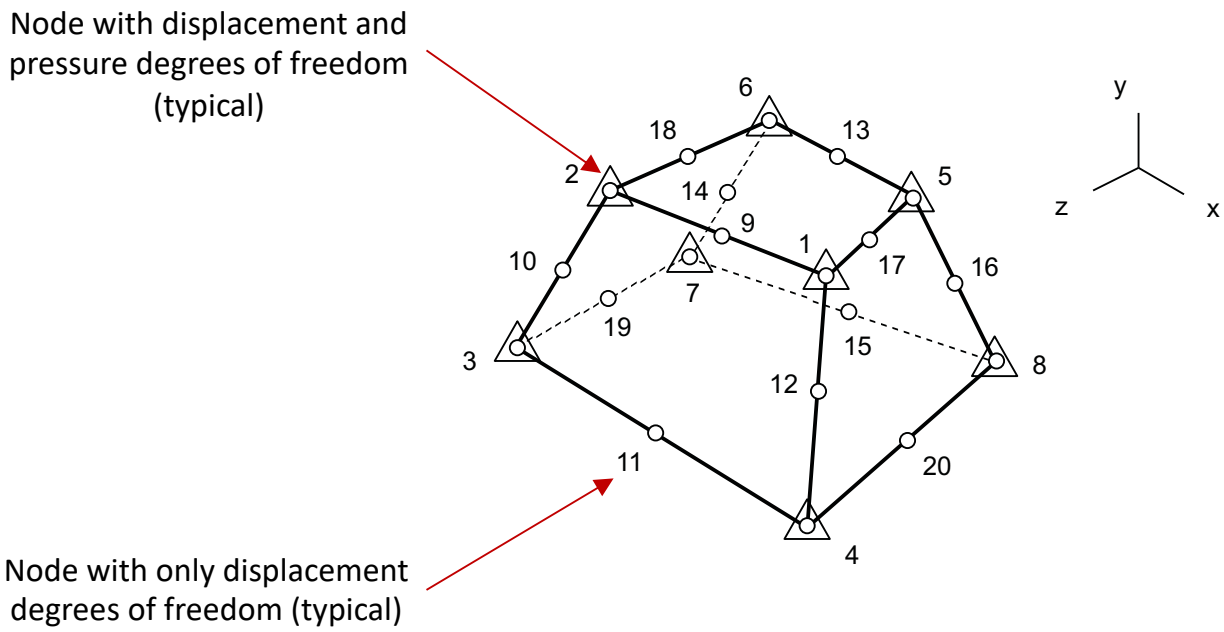

Figure 28.10: Node numbering associated with a typical mixed 20-node triquadratic (H20P8c) hexahedral continuum element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default value for the **MATERIAL** number is one (1).

- The **HYDRAULIC** keyword is used to specify the number of the hydraulic conductivity (permeability) idealization associated with the element. In order to maximize flexibility, the **HYDRAULIC** number differs from the **MATERIAL** number. The default value for the **HYDRAULIC** conductivity number is one (1).

- The **INITIAL** keyword is used to specify the initial state number associated with the element. The *default* value for the **INITIAL** is zero (0).

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the *default* condition, no input is required in such a case. The condition of no excavation is likewise the default. In the current version of APES+ the hydraulic boundary condition at the temporary top of a soil mass that is being constructed is *zero flow*; i.e., an *impervious* surface. The conditions on top of the final surface can, however, be specified through the use of the **SPECIFICATIONS CONCENTRATED FLOW** command.

- If the **INCompressible_constituents** keyword is used, both the solid skeleton and the pore fluid are assumed to be *incompressible*. This supersedes any bulk modulus values specified in conjunction with the **MATERIAL FLUID** or **MATERIAL SOLID** commands. If an anisotropic poroelastic material idealization is used (refer to the description of the **MATERIAL POROELASTIC ANISOTROPIC** command), this implies an *infinite* value for the "Biot modulus" M (specified via the **M_MODULUS** keyword) for the element.

- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the *default* condition both strains are printed.

- Specification of the **DONT_PRINT_STRESSES** keyword indicates that stresses are not to be printed. Under the *default* condition stresses are printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the *default* condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the *default* condition these quantities are not computed and printed.

- The **PRINT_VOLUMETRIC_STRAIN** keyword causes the volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the volumetric strain to the absolute value of the minimum non-zero normal strain in the element is printed. That is,

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33}) \neq 0$$

  This ratio is instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the *default* condition the volumetric strain and the aforementioned ratio are not computed and printed.

- The **PRINT_VELOCITY** keyword causes the components of the Darcy velocity in the global x_1- and x_2-coordinate directions to be computed and printed at the element center.

28.11 The ELEMENT MIXED_FLOW H27P8c Command

Synopsis

The **ELEMENT MIXED_FLOW H27P8c** command is used to describe all mixed, 27-node, triquadratic isoparametric "Lagrangian" hexahedral continuum elements with a trilinear *continuous* pressure approximation.

Remarks

- The **H27P8c** is a mixed, isoparametric "Lagrangian" hexahedral element [52].
- The element
  - Contains eight (8) vertex nodes.
  - Contains twelve (12) midside nodes.
  - Contains nine (9) interior nodes.
  - Has three (3) displacements degrees of freedom at each node, for a total of eighty-one (81) displacement degrees of freedom.
  - Employs a *triquadratic* approximation for the displacement field.
  - Employs a *trilinear*, continuous approximation for the pressure (Fig. 28.11).
  - Has a total of eight (8) pressure degrees of freedom.
- The **H27P8c** element should be used for coupled displacement-flow analyses of porous media. The element is based on a generalized Biot formulation and employs a $u - p$ form of the governing equations.

Syntax

The following syntax is used to describe a typical **H27P8c** mixed-flow hexahedral continuum element:

ELEment MIXed_Flow TYPe H27P8c (DEScription "string")
NODes #:#:# (MATerial #) (HYDraulic #.#) (INItial #)
(CONstruction #) (EXCavation #)
(FLUID_Density #) (SOLID_Density #) (FLUID_Bulk #) (SOLID_Bulk #)
(INCompressible_constituents)

(1_Additional #) (1_Increment #)
(2_Additional #) (2_Increment #)
(3_Additional #) (3_Increment #)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLumetric_strain) (PRINT_VELocity)

Explanatory Notes

- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with **H20P8c** elements, which must be specified sequentially from 1 to 27, is shown in Figure 28.11.

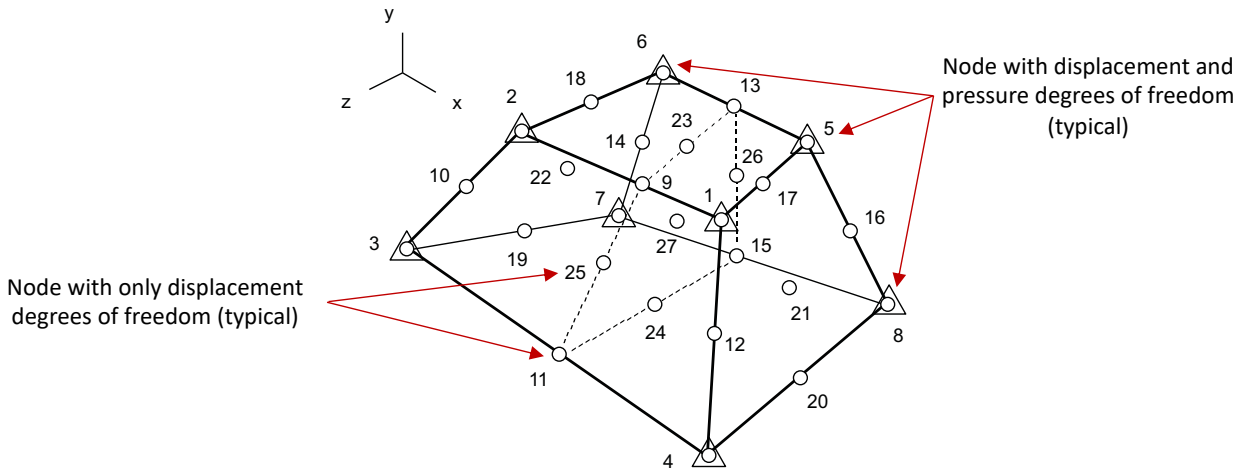

Figure 28.11: Node numbering associated with a typical mixed 27-node triquadratic (H27P8c) hexahedral continuum element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default value for the **MATERIAL** number is one (1).
- The **HYDRAULIC** keyword is used to specify the number of the hydraulic conductivity (permeability) idealization associated with the element. In order to maximize flexibility, the **HYDRAULIC** number differs from the **MATERIAL** number. The default value for the **HYDRAULIC** conductivity number is one (1).

- The **INITIAL** keyword is used to specify the initial state number associated with the element. The *default* value for the **INITIAL** is zero (0).

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the *default* condition, no input is required in such a case. The condition of no excavation is likewise the default. In the current version of APES+ the hydraulic boundary condition at the temporary top of a soil mass that is being constructed is *zero flow*; i.e., an *impervious* surface. The conditions on top of the final surface can, however, be specified through the use of the **SPECIFICATIONS CONCENTRATED FLOW** command.

- If the **INCompressible_constituents** keyword is used, both the solid skeleton and the pore fluid are assumed to be *incompressible*. This supersedes any bulk modulus values specified in conjunction with the **MATERIAL FLUID** or **MATERIAL SOLID** commands. If an anisotropic poroelastic material idealization is used (refer to the description of the **MATERIAL POROELASTIC ANISOTROPIC** command), this implies an *infinite* value for the "Biot modulus" M (specified via the **M_MODULUS** keyword) for the element.

- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the *default* condition both strains are printed.

- Specification of the **DONT_PRINT_STRESSES** keyword indicates that stresses are not to be printed. Under the *default* condition stresses are printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the *default* condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the *default* condition these quantities are not computed and printed.

- The **PRINT_VOLUMETRIC_STRAIN** keyword causes the volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the volumetric strain to the absolute value of the minimum non-zero normal strain in the element is printed. That is,

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33}) \neq 0$$

  This ratio is instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the *default* condition the volumetric strain and the aforementioned ratio are not computed and printed.

- The **PRINT_VELOCITY** keyword causes the components of the Darcy velocity in the global x_1- and x_2-coordinate directions to be computed and printed at the element center.

28.12 The ELEMENT MIXED_FLOW Q4P4c Command

Synopsis

The **ELEMENT MIXED_FLOW Q4P4c** command is used to describe all mixed, 4-node, bilinear quadrilateral continuum elements with a bilinear *continuous* pressure approximation.

Remarks

- The **Q4P4c** is a mixed, isoparametric hexahedral element [52].
- The element
  - Contains four (4) vertex nodes.
  - Has two (2) displacements degrees of freedom at each node, for a total of eight (8) displacement degrees of freedom.
  - Employs a *bilinear* approximation for the displacement field.
  - Employs a *bilinear*, continuous approximation for the pressure (Fig. 28.12).
  - Has a total of four (4) pressure degrees of freedom.
- The **Q4P4c** element should be used for coupled displacement-flow analyses of porous media. The element is based on a generalized Biot formulation and employs a $u - p$ form of the governing equations.

Syntax

The following syntax is used to describe a typical **Q4P4c** mixed-flow quadrilateral continuum element:

ELEment MIXed_Flow TYPe Q4P4c (DEScription "string")
NODes #:#:# (MATerial #) (HYDraulic #.#) (INItial #)
(CONstruction #) (EXCavation #)
(FLUID_Density #) (SOLID_Density #) (FLUID_Bulk #) (SOLID_Bulk #)
(INCompressible_constituents) (REDuced #.#)
(1_Additional #) (1_Increment #) (2_Additional #) (2_Increment #)
(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLumetric_strain) (PRINT_VELocity)

Explanatory Notes

- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENTS OFF** or the **ECHO ELEMENT_DESCRIPTIONS OFF** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with **Q4P4c** elements, which must be specified sequentially from $n1$ to $n4$, is shown in Figure 28.12.

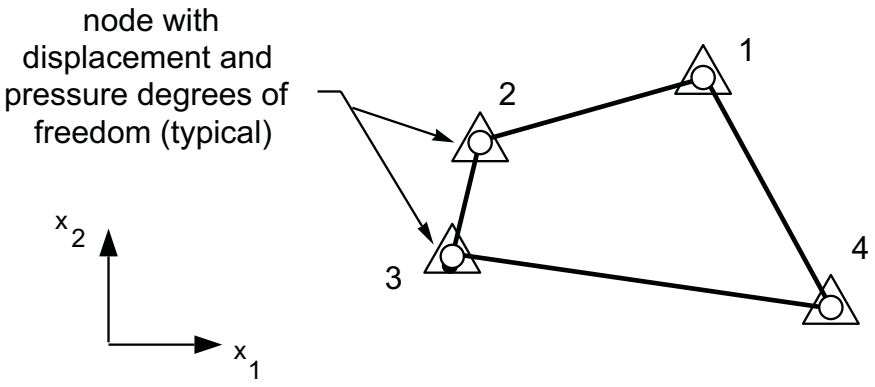

Figure 28.12: Node numbering associated with a typical mixed 4-node bilinear Q4P4c quadrilateral continuum element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default value for the **MATERIAL** number is one (1).
- The **HYDRAULIC** keyword is used to specify the number of the hydraulic conductivity (permeability) idealization associated with the element. In order to maximize flexibility, the **HYDRAULIC** number differs from the **MATERIAL** number. The default value for the **HYDRAULIC** conductivity number is one (1).
- The **INITIAL** keyword is used to specify the initial state number associated with the element. The *default* value for the **INITIAL** is zero (0).
- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to, or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the *default* condition, no input is required in such a case. The condition of no excavation is likewise the default. In the current version of APES+ the hydraulic boundary condition at the temporary top of a soil mass that is being constructed is *zero flow*; i.e., an *impervious* surface. The

conditions on top of the final surface can, however, be specified through the use of the **SPECIFICATIONS CONCENTRATED FLOW** command.

- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The *default* **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE STRAIN** idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.

- The **FLUID_DENSITY** keyword is used to specify the number of the mass density idealization for the fluid phase (ρ^f) (units of ML^{-3} or $\mathrm{FL}^{-4}\mathrm{t}^2$) associated with the element. The **SOLID_DENSITY** keyword is used to specify the number of the mass density idealization for the solid phase (ρ^s) associated with the element.

  Given the values of ρ^f and ρ^s (specified using the **DENSITY_FLUID** and **DENSITY_SOLID** commands, respectively, as described in Chapter 17, the total mass density of a saturated soil is computed from

$$\rho = (1-n)\rho^s + n\rho^f$$

  where n is the porosity, which is related to the void ratio (e) by $n = e/(1+e)$.

- The **FLUID_BULK** keyword is used to specify the number of the bulk modulus idealization for the fluid phase (K_f) (units of FL^{-2}) associated with the element. The **SOLID_BULK** keyword is used to specify the number of the bulk modulus idealization for the solid phase (K_s) associated with the element.

  Given the values of K_f and K_s (specified using the **BULK_FLUID** and **BULK_SOLID** commands, respectively, as described in Chapter 16, the value of the "Biot" modulus [27] is computed as follows:

$$\frac{1}{M} = \frac{n}{K_f} + \frac{1-n}{K_s}$$

  where n is again the porosity. An explicit value of M can be input using the **MATERIAL POROELASTIC ANISOTROPIC** command.

- If the **INCompressible_constituents** keyword is used, both the solid skeleton and the pore fluid are assumed to be *incompressible*. This supersedes any bulk modulus values specified in conjunction with the **MATERIAL FLUID** or **MATERIAL SOLID** commands. If an anisotropic poroelastic material idealization is used (refer to the description of the **MATERIAL POROELASTIC ANISOTROPIC** command), this implies an *infinite* value for the "Biot modulus" M (specified via the **M_MODULUS** keyword) for the element.

- The value specified in conjunction with the **REDUCED** keyword determines whether *hybrid* selective reduced integration shall be used in the determination of the "coupling"

submatrix associated with the mixed penalty elements. What this actually entails is replacing the interpolation functions approximating the element pressure field in the "coupling" submatrix with the following expression:

$$N_i^{(p)*} = \left[\varpi N_i^{(p)} + \frac{1}{4}(1-\varpi)\right] \quad ; \quad i = 1, N_{en}$$

where $0 \leq \varpi \leq 1$ is the value of the reduced integration parameter specified in conjunction with the **REDUCED** command, $N_i^{(p)}$ represents the bi-linear pressure interpolation function associated with node i, and N_{en} denotes the number of element nodes $(= 4)$. If $\varpi = 0.0$, the above expression reduces to a value of $1/4$, which corresponds to $N_i^{(p)}$ evaluated at the element center; i.e., at the *reduced* (1×1) quadrature point for the **Q4P4c** element. If $\varpi = 1.0$, the above expression simply yields $N_i^{(p)}$, with the quadrature points coinciding with the standard (full integration) 2×2 Gauss-Legendre quadrature points.

- If the body being analyzed can be divided into a layer (or layers) of elements, and if the characteristics of the element (i.e., the **MATERIAL**, the **INITIAL** state, the incremental **CONSTRUCTION** and **EXCAVATION** numbers, and the **THICKNESS** value) are the *same* for several elements within a layer, and if the nodes are numbered in a *consistent* fashion, then an element data generation option can be employed. To generate a sequence of elements within a single layer, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**.

  If several layers of elements have the *same* attributes, the above generation option can be carried one step further by making use of the **2_ADDITIONAL**, and **2_INCREMENT** keywords.

- The purpose of the **PRINT** commands is to eliminate unnecessary output (printed and saved) generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the *default* condition both strains are printed.

- Specification of the **DONT_PRINT_STRESSES** keyword indicates that stresses are not to be printed. Under the *default* condition stresses are printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the *default* condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the *default* condition these quantities are not computed and printed.

- The **PRINT_AVG_STRAINS** keyword indicates that average strains (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the *default* condition average strains are not computed and printed.

- The **PRINT_AVG_STRESSES** keyword indicates that average stresses (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the *default* condition average stresses are not computed and printed.

- The **PRINT_VOLUMETRIC_STRAIN** keyword causes the volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the volumetric strain to the absolute value of the minimum non-zero normal strain in the element is printed. That is,

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33}) \neq 0$$

  This ratio is instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the *default* condition the volumetric strain and the aforementioned ratio are *not* computed and printed.

- The **PRINT_AVG_VOLUMETRIC_STRAIN** keyword causes the *average* volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the *average* volumetric strain to the absolute value of the minimum *average* non-zero normal strain in the element is printed. This ratio is also instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the *default* condition the average volumetric strain and the aforementioned ratio are *not* computed and printed.

- The **PRINT_AVG_PRESSURE** keyword causes the *average* pressure to be computed and printed for the element. This value represents the average of the approximate pressures at the four pressure (vertex) nodes.

- The **PRINT_VELOCITY** keyword causes the components of the Darcy velocity in the global x_1- and x_2-coordinate directions to be computed and printed at the element center.

28.13 The ELEMENT MIXED_FLOW H8P8c Command

Synopsis

The **ELEMENT MIXED_FLOW H8P8c** command is used to describe all mixed, 8-node, trilinear hexahedral continuum elements with a trilinear *continuous* pressure approximation.

Remarks

- The **H8P8c** is a mixed, isoparametric hexahedral element [52].
- The element
  - Contains eight (8) vertex nodes.
  - Has three (3) displacements degrees of freedom at each node, for a total of twenty-four (24) displacement degrees of freedom.
  - Employs a *trilinear* approximation for the displacement field.
  - Employs a *trilinear*, continuous approximation for the pressure (Fig. 28.13).
  - Has a total of eight (8) pressure degrees of freedom.
- The **H8P8c** element should be used for coupled displacement-flow analyses of porous media. The element is based on a generalized Biot formulation and employs a $u - p$ form of the governing equations.

Syntax

The following syntax is used to describe a typical **H8P8c** mixed-flow quadrilateral continuum element:

ELEment MIXed_Flow TYPe H8P8c (DEScription "string")
NODes #:#:# (MATerial #) (HYDraulic #.#) (INItial #)
(CONstruction #) (EXCavation #)
(FLUID_Density #) (SOLID_Density #) (FLUID_Bulk #) (SOLID_Bulk #)
(INCompressible_constituents) (REDuced #.#)
(1_Additional #) (1_Increment #)
(2_Additional #) (2_Increment #)
(3_Additional #) (3_Increment #)

(DONT_PRINT_Results)
(DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
(PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
(PRINT_VOLumetric_strain) (PRINT_VELocity)

Explanatory Notes

- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENTS OFF** or the **ECHO ELEMENT_DESCRIPTIONS OFF** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with **H8P8c** elements, which must be specified sequentially from $n1$ to $n8$, is shown in Figure 28.13.

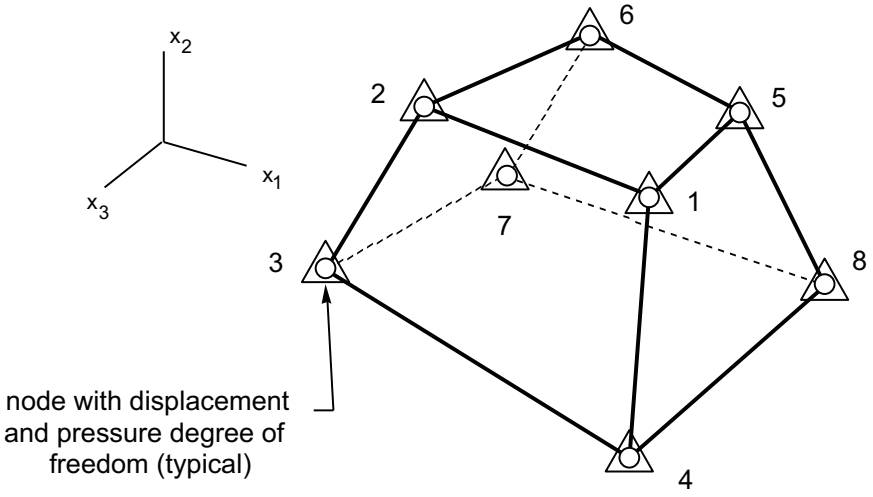

Figure 28.13: Node numbering associated with a typical mixed 8-node trilinear (H8P8c) hexahedral continuum element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default value for the **MATERIAL** number is one (1).
- The **HYDRAULIC** keyword is used to specify the number of the hydraulic conductivity (permeability) idealization associated with the element. In order to maximize flexibility, the **HYDRAULIC** number differs from the **MATERIAL** number. The default value for the **HYDRAULIC** conductivity number is one (1).
- The **INITIAL** keyword is used to specify the initial state number associated with the element. The *default* value for the **INITIAL** is zero (0).

- The incremental **CONSTRUCTION** and **EXCAVATION** numbers represent the time increment in which the material in this element(s) is added to, or removed from the model. A **CONSTRUCTION** number equal to *zero* corresponds to a material in existence at the beginning of the analysis. Since this is the *default* condition, no input is required in such a case. The condition of no excavation is likewise the default. In the current version of APES+ the hydraulic boundary condition at the temporary top of a soil mass that is being constructed is *zero flow*; i.e., an *impervious* surface. The conditions on top of the final surface can, however, be specified through the use of the **SPECIFICATIONS CONCENTRATED FLOW** command.

- The **FLUID_DENSITY** keyword is used to specify the number of the mass density idealization for the fluid phase (ρ^f) (units of ML^{-3} or $\mathrm{FL}^{-4}\mathrm{t}^2$) associated with the element. The **SOLID_DENSITY** keyword is used to specify the number of the mass density idealization for the solid phase (ρ^s) associated with the element.

  Given the values of ρ^f and ρ^s (specified using the **DENSITY_FLUID** and **DENSITY_SOLID** commands, respectively, as described in Chapter 17, the total mass density of a saturated soil is computed from

$$\rho = (1-n)\rho^s + n\rho^f$$

  where n is the porosity, which is related to the void ratio (e) by $n = e/(1+e)$.

- The **FLUID_BULK** keyword is used to specify the number of the bulk modulus idealization for the fluid phase (K_f) (units of FL^{-2}) associated with the element. The **SOLID_BULK** keyword is used to specify the number of the bulk modulus idealization for the solid phase (K_s) associated with the element.

  Given the values of K_f and K_s (specified using the **BULK_FLUID** and **BULK_SOLID** commands, respectively, as described in Chapter 16, the value of the "Biot" modulus [27] is computed as follows:

$$\frac{1}{M} = \frac{n}{K_f} + \frac{1-n}{K_s}$$

  where n is again the porosity. An explicit value of M can be input using the **MATERIAL POROELASTIC ANISOTROPIC** command.

- If the **INCompressible_constituents** keyword is used, both the solid skeleton and the pore fluid are assumed to be *incompressible*. This supersedes any bulk modulus values specified in conjunction with the **MATERIAL FLUID** or **MATERIAL SOLID** commands. If an anisotropic poroelastic material idealization is used (refer to the description of the **MATERIAL POROELASTIC ANISOTROPIC** command), this implies an *infinite* value for the "Biot modulus" M (specified via the **M_MODULUS** keyword) for the element.

- The value specified in conjunction with the **REDUCED** keyword determines whether *hybrid* selective reduced integration shall be used in the determination of the "coupling"

submatrix associated with the mixed penalty elements. What this actually entails is replacing the interpolation functions approximating the element pressure field in the "coupling" submatrix with the following expression:

$$N_i^{(p)*} = \left[\alpha N_i^{(p)} + \frac{1}{8}(1-\alpha)\right] \; ; \; i = 1, N_{en}$$

where α represents the value specified in conjunction with the **REDUCED** command, $N_i^{(p)}$ represents the bi-linear pressure interpolation function associated with node i, and N_{en} denotes the number of element nodes (= 8). For $\alpha = 0$, the above expression reduces to a value of 1/8, which corresponds to $N_i^{(p)}$ evaluated at the element center; i.e., at the *reduced* quadrature point for the **H8P8c** element. For $\alpha = 1.0$, the above expression simply yields $N_i^{(p)}$, with the quadrature points coinciding with the standard ones.

- If the body being analyzed can be divided into a layer (or layers) of elements, and if the characteristics of the element (i.e., the **MATERIAL**, the **INITIAL** state, the incremental **CONSTRUCTION** and **EXCAVATION** numbers, and the **THICKNESS** value) are the *same* for several elements within a layer, and if the nodes are numbered in a *consistent* fashion, then an element data generation option can be employed. To generate a sequence of elements within a single layer, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**.

  If several layers of elements have the *same* attributes, the above generation option can be carried one step further by making use of the **2_ADDITIONAL**, and **2_INCREMENT** keywords.

- The purpose of the **PRINT** commands is to eliminate unnecessary output (printed and saved) generated by APES$^+$. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., strains and stresses) for this element.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are not to be printed. Under the *default* condition both strains are printed.

- Specification of the **DONT_PRINT_STRESSES** keyword indicates that stresses are not to be printed. Under the *default* condition stresses are printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed and printed for the element. Under the *default* condition these quantities are not computed and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be computed and printed for the element. Under the *default* condition these quantities are not computed and printed.

- The **PRINT_AVG_STRAINS** keyword indicates that average strains (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the *default* condition average strains are not computed and printed.

- The **PRINT_AVG_STRESSES** keyword indicates that average stresses (averaged over the secondary quadrature points) are to be computed and printed for the element. Under the *default* condition average stresses are not computed and printed.

- The **PRINT_VOLUMETRIC_STRAIN** keyword causes the volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the volumetric strain to the absolute value of the minimum non-zero normal strain in the element is printed. That is,

$$\frac{|\varepsilon_{vol}|}{|\min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33})|} \quad ; \quad \min(\varepsilon_{11},\varepsilon_{22},\varepsilon_{33}) \neq 0$$

  This ratio is instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the *default* condition the volumetric strain and the aforementioned ratio are *not* computed and printed.

- The **PRINT_AVG_VOLUMETRIC_STRAIN** keyword causes the *average* volumetric strain to be computed and printed for the element. In addition, the ratio of the absolute value of the *average* volumetric strain to the absolute value of the minimum *average* non-zero normal strain in the element is printed. This ratio is also instructive in the assessment of mixed and mixed/penalty elements used to simulate material response in the incompressible limit. Under the *default* condition the average volumetric strain and the aforementioned ratio are *not* computed and printed.

- The **PRINT_AVG_PRESSURE** keyword causes the *average* pressure to be computed and printed for the element. This value represents the average of the approximate pressures at the four pressure (vertex) nodes.

- The **PRINT_VELOCITY** keyword causes the components of the Darcy velocity in the global x_1-, x_2- and x_3-coordinate directions to be computed and printed at the element center.

Chapter 29

ELEMENT SCALAR Commands

The available irreducible elements available for the analysis of scalar field problems are listed below. These elements are entered using the **ELEMENT SCALAR** commands.

29.1 The ELEMENT SCALAR TYPE T3S Command

Synopsis

The **ELEMENT SCALAR TYPE T3S** command is used to describe all irreducible, 3-node linear triangular continuum elements that are to be used in the analysis of scalar field problems.

Syntax

The following syntax is used to describe a typical **T3S** irreducible quadrilateral continuum element:

ELEment SCAlar TYPe T3S (DEScription "string")
NODes #:#:#
(SCAlar #) (SOUrce #) (INItial #) (INTcode #)
(1_Additional #) (1_Increment #)
(2_Additional #) (2_Increment #)
(DONT_PRINT_Results)

Explanatory Notes

- The **T3S** is an irreducible, linear, isoparametric triangular continuum element. The element
  - Contains three (3) vertex nodes.
  - Has one (1) scalar degree of freedom at each node.
  - Possesses a total of six (6) displacement degrees of freedom.
- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with **T3S** elements, which must be specified sequentially from 1 to 3, is shown in Figure 29.1.

  NOTE: Presently APES$^+$ does *not* possess the ability to generate **T3P0** elements. It is assumed that the analyst will use some stand alone pre-processing software to accomplish this task. The resulting element and node data will then be translated to the format expected by APES$^+$.

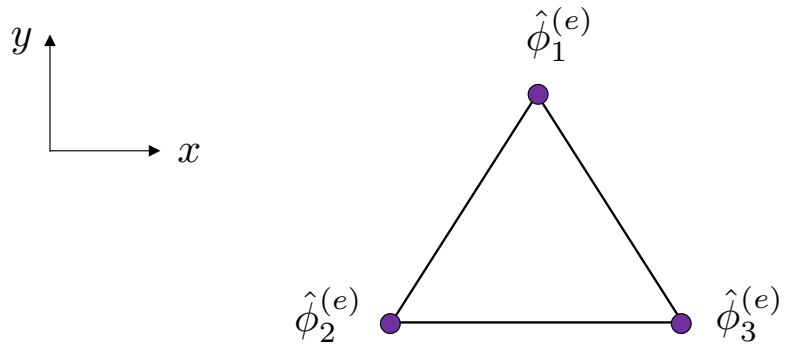

Figure 29.1: Node numbering and degrees of freedom associated with a typical irreducible, 3-node linear (**T3S**) triangular continuum element.

- The **MATERIAL** keyword is used to specify the number of the material idealization associated with the element. The default values for the **MATERIAL** number is one (1).
- The **INITIAL** keyword is used to specify the initial state number associated with the element. The default value for the **INITIAL** is zero (0).
- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The default **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE**

STRAIN idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.

- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., fluxes) for this element.

29.2 The ELEMENT SCALAR TYPE Q4S Command

Synopsis

The **ELEMENT SCALAR TYPE Q4S** command is used to describe all irreducible, 4-node bilinear quadrilateral continuum elements that are to be used in the analysis of scalar field problems.

Syntax

The following syntax is used to describe a typical **Q4S** irreducible quadrilateral continuum element:

ELEment SCAlar TYPe Q4S (DEScription "string")
NODes #:#:#
(SCAlar #) (SOUrce #) (INItial #) (INTcode #)
(1_Additional #) (1_Increment #)
(2_Additional #) (2_Increment #)
(DONT_PRINT_Results)

Explanatory Notes

- The **Q4S** is an irreducible, bilinear, isoparametric Lagrangian continuum element [52]. The element
  - Contains four (4) vertex nodes.
  - Has one (1) scalar degree of freedom at each node.
  - Possesses a total of four (4) degrees of freedom.
- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with a **Q4S** element, which must be specified sequentially from 1 to 4, is shown in Figure 29.2.

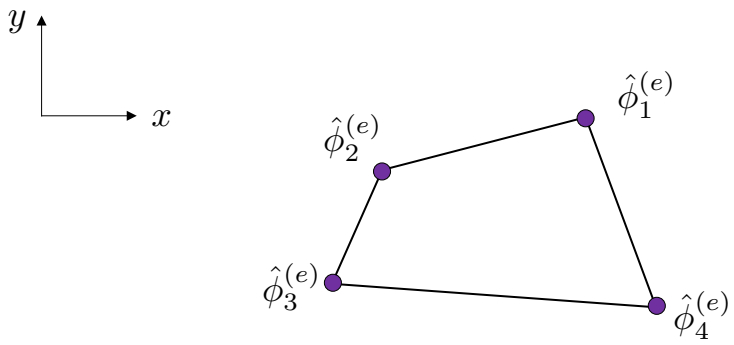

Figure 29.2: Node numbering and degrees of freedom associated with a typical irreducible, 4-node bilinear (**Q4S**) quadrilateral continuum element.

- A linear (3-node) triangular elements for mechanical analyses can be developed by *degenerating* a four-node (**Q4S**) quadrilateral. This is realized by repeating the first node number as the fourth; i.e., for a typical **Q4S** element, the nodes would be specified as follows:

```
nodes  n1  n2  n3  n1
```

  The results obtained using degenerated quadrilaterals are *identical* to those from true 3-node triangular (**T3S**) elements [5; 44]. Since the **T3S** element is implemented in the APES+ computer program (see Section 29.1), if use of a 3-node triangular element is desired, it is simper to use the **T3S** directly.

- The **SCALAR** keyword is used to specify the number of the scalar material parameter idealization associated with the element. The *default* values for the **SCALAR** number is one (1).

- The optional **SOURCE** keyword is used to specify the number of the scalar source associated with the element. The *default* values for the **SCALAR** number is zero (0).

- The **INITIAL** keyword is used to specify the initial state number associated with the element. The *default* value for the **INITIAL** is zero (0).

- The **THICKNESS** keyword is used to specify the material thickness assumed for the element. Over a given element, the thickness is assumed to be constant. The default **THICKNESS** value is equal to one (1.0). For **AXISYMMETRIC** and **PLANE STRAIN** idealizations (see discussion of the **ANALYSIS IDEALIZATION** command), the **THICKNESS** must be equal to 1.0. For such idealizations, specified values different from 1.0 are ignored and the proper value is used.

- The value specified in conjunction with the **INTCODE** keyword describes the order of numerical integration scheme to be used in developing the element equations for the element.

  The "commonly" used numerical integration rule for **Q4S** elements corresponds to a 2 by 2 Gauss-Legendre quadrature scheme (degree of precision equal to 3) for the primary dependent variables (i.e., nodal displacements) and a 1-point Gauss-Legendre scheme (degree of precision equal to 1) for the secondary dependent variables (i.e., strains and stresses). This is the default condition and requires no input using the **INTCODE** keyword. If a quadrature order different from the default condition is desired, the following integer values are associated with this keyword:

  INTCODE = 11: A 1-point Gauss-Legendre quadrature scheme (degree of precision equal to 1) is used to compute *both* the primary dependent variables (i.e., nodal displacements) and for the secondary dependent variables (i.e., strains and stresses).

  This serves to uniformly *under integrate* the element. The disadvantage of uniform reduced integration is that the rank of the element stiffness matrix $\mathbf{K}^{(e)}$ may be reduced, resulting in the singularity or near singularity of the global $\mathbf{K}$ [52]. The associated element instabilities manifest themselves in the form of *mechanisms* or *hourglass modes* [44]. Consequently, the specification of **INTCODE** = 11 should only be used to investigate the possible pathologies associated with uniform reduced integration and *not* in standard analyses.

  INTCODE = 21: a 2 by 2 Gauss-Legendre quadrature scheme (degree of precision equal to 3) is used to compute the primary dependent variables (i.e., nodal displacements) and a 1-point Gauss-Legendre scheme (degree of precision equal to 1) is used to compute the secondary dependent variables (i.e., strains and stresses). This is equivalent to the aforementioned default setting.

- If the body being analyzed can be divided into a layer (or layers) of elements, and if the characteristics of the element (i.e., the **SCALAR**, **SOURCE** and **INITIAL** state, numbers, the **THICKNESS**, and the **INTCODE**) are the *same* for several elements within a layer, and if the nodes are numbered in a *consistent* fashion, then an element data generation option can be employed. To generate a sequence of elements within a single layer, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**.

  For example, assuming that the coordinates of all boundary nodes have been specified, the bottom row of elements in the mesh shown in Figure 29.3 could be established by entering the node numbers associated with element *a* and the keywords/values **1_ADDITIONAL** 4 and **1_INCREMENT** 5. Assuming a **MATERIAL** number equal to one (1) and default values for all of the other element parameters, the input record corresponding to this specification would be:

```
element irreducible type "q4s" nodes  1  6  7  2  scalar  1 &
```

```
   1_add  4   1_incr 5
```

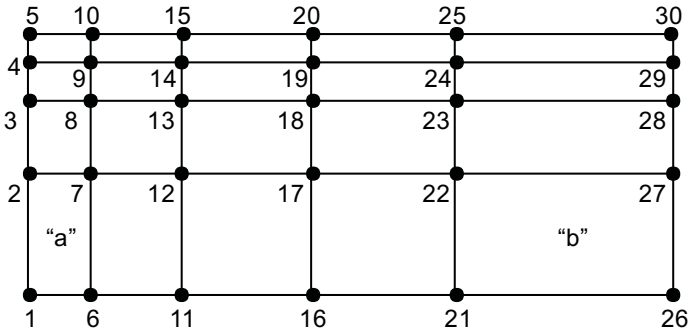

Figure 29.3: Hypothetical mesh consisting of **Q4S** elements.

Alternately, this bottom row of elements could be established by entering the node numbers associated with element *b* and the keywords/values **1_ADDITIONAL** 4 and **1_INCREMENT** -5 (i.e., a decrement). Assuming once again a **SCALAR** number equal to one (1) and default values for all of the other element parameters, the input record corresponding to this specification would be as follows:

```
element irreducible type "q4s" nodes  21  26  27  22 scalar 1 &
   1_add  4   1_incr -5
```

In a similar manner, the left most column of elements in the mesh could be established by entering the node numbers associated with element *a* and the keywords/values **1_ADDITIONAL** 2 and **1_INCREMENT** 1. The corresponding input record would be as follows:

```
element irreducible type "q4s" nodes  1  6  7  2  scalar  1 &
   1_add  2   1_incr 1
```

If several layers of elements have the *same* attributes, the above generation option can be carried one step further. For example, the entire mesh shown in Figure **??** could be established by entering the node numbers of element *a* and the keywords/values **1_ADDITIONAL** 4, **1_INCREMENT** 5, **2_ADDITIONAL** 3, and **2_INCREMENT** 1. The associated input record would thus be as follows:

```
element irreducible type "q4s" nodes  1  6  7  2  material  1 &
   1_add  4   1_incr 5  2_add  3   2_incr 1
```

Note that the 1- and 2-directions need not be orthogonal, nor be aligned with the global coordinate directions. Alternately, the mesh shown in Figure **??** could be generated by specifying the node points associated with element *a* and the keywords/values **1_ADDITIONAL** 3, **1_INCREMENT** 1, **2_ADDITIONAL** 4, and **2_INCREMENT** 5. The associated input record would thus be as follows:

```
element irreducible type "q4s" nodes  1  6  7  2  material  1 &
   1_add  2   1_incr 1  2_add  4   2_incr 5
```

The mesh generated with these two specifications would differ only in the order in which the elements were numbered by the APES+ program.

- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT SCALAR** command, then all the elements generated will be affected in a like manner by the above print control commands.
- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., fluxes) for this element.

29.3 The ELEMENT SCALAR TYPE Q8S Command

Synopsis

The **ELEMENT SCALAR TYPE Q8S** command is used to describe all irreducible, 8-node quadratic quadrilateral continuum elements that are to be used in the analysis of scalar field problems.

Syntax

The following syntax is used to describe a typical **Q8S** irreducible quadrilateral continuum element:

ELEment SCAlar TYPe Q8S (DEScription "string")
NODes #:#:#
(SCAlar #) (SOUrce #) (INItial #) (INTcode #)
(1_Additional #) (1_Increment #)
(2_Additional #) (2_Increment #)
(DONT_PRINT_Results)

Explanatory Notes

- The **Q8S** is an irreducible, quadratic, isoparametric Lagrangian continuum element [52]. The element
  - Contains eight (8) vertex nodes.
  - Has one (1) scalar degree of freedom at each node.
  - Possesses a total of eight (8) degrees of freedom.
- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with a **Q8S** element, which must be specified sequentially from 1 to 8, is shown in Figure 29.4.

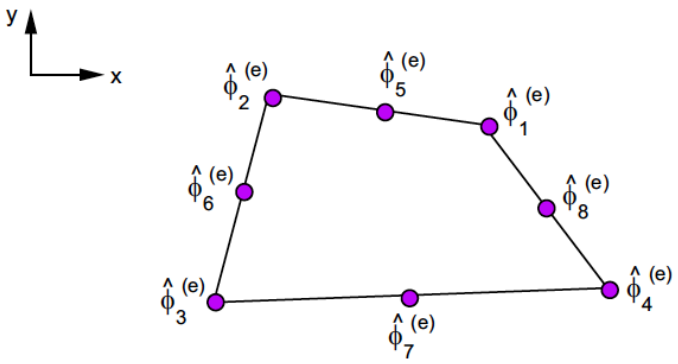

Figure 29.4: Node numbering and degrees of freedom associated with a typical irreducible, 8-node quadratic (**Q8S**) quadrilateral continuum element.

The "commonly" used numerical integration rule for **Q8S** elements corresponds to a 3 by 3 Gauss-Legendre quadrature scheme (degree of precision equal to 5) for the primary dependent variables (i.e., nodal displacements) and a 2 by 2 Gauss-Legendre scheme (degree of precision equal to 3) for the secondary dependent variables (i.e., strains and stresses). This is the default condition and requires no input using the **INTCODE** keyword. If a quadrature order different from the default condition is desired, the following integer values are associated with this keyword:

INTCODE = 32: a 3 by 3 Gauss-Legendre quadrature scheme (degree of precision equal to 5) is used to compute the primary dependent variables (i.e., nodal displacements) and a 2 by 2 Gauss-Legendre scheme (degree of precision equal to 3) is used to compute the secondary dependent variables (i.e., strains and stresses). This is equivalent to the aforementioned default setting.

INTCODE = 21: a 2 by 2 Gauss-Legendre quadrature scheme (degree of precision equal to 3) is used to compute the primary dependent variables (i.e., nodal displacements) and a 1-point Gauss-Legendre scheme (degree of precision equal to 1) is used to compute the secondary dependent variables (i.e., strains and stresses).

- If the body being analyzed can be divided into a layer (or layers) of elements, and if the characteristics of the element (i.e., the **SCALAR**, **SOURCE** and **INITIAL** state, numbers, the **THICKNESS**, and the **INTCODE**) are the *same* for several elements within a layer, and if the nodes are numbered in a *consistent* fashion, then an element data generation option can be employed. To generate a sequence of elements within a single layer, node numbers are specified only for the first element, together with appropriate values for **1_ADDITIONAL** and **1_INCREMENT**.

- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates

inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT SCALAR** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., fluxes) for this element.

29.4 The ELEMENT SCALAR TYPE TET4S Command

Synopsis

The **ELEMENT SCALAR TYPE TET4S** command is used to describe all irreducible, 4-node linear isoparametric tetrahedral continuum elements that are to be used in the analysis of scalar field problems.

Syntax

The following syntax is used to describe a typical **TET4S** irreducible tetrahedral continuum element:

ELEment SCAlar TYPe TET4S (DEScription "string")
NODes #:#:#
(SCAlar #) (SOUrce #) (INItial #)
(1_Additional #) (1_Increment #)
(2_Additional #) (2_Increment #)
(3_Additional #) (3_Increment #)
(DONT_PRINT_Results)

Explanatory Notes

- The **H8S** is an irreducible, trilinear, isoparametric hexahedral continuum element [52]. The element
  - Contains four (4) vertex nodes.
  - Has one (1) scalar degree of freedom at each node.
  - Possesses a total of four (4) degrees of freedom.
- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.

- The numbering order of **NODES** associated with **TET4S** elements, which must be specified sequentially from 1 to 4, is shown in Figure 29.5.

  NOTE: Presently APES+ does *not* possess the ability to generate **TET4S** elements. It is assumed that the analyst will thus use some stand-alone pre-processing software to accomplish this task. The resulting element and node data will then be translated to the format expected by APES+.

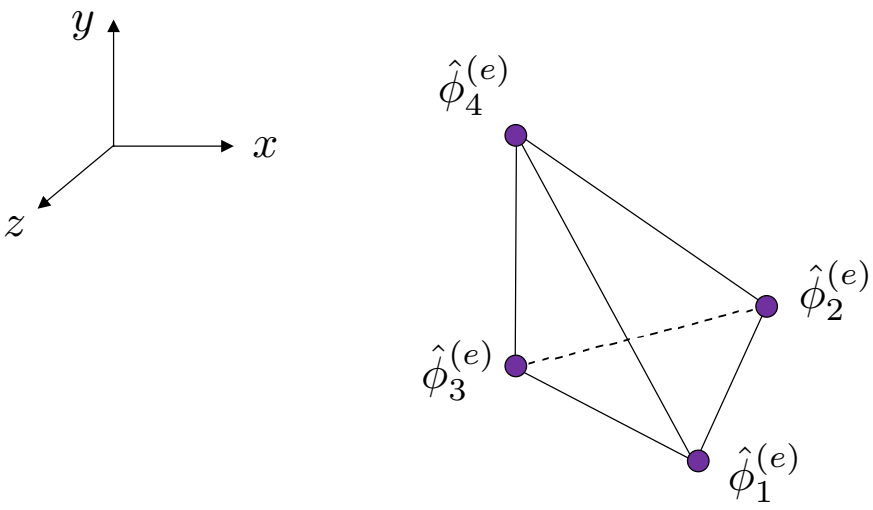

Figure 29.5: Node numbering and degrees of freedom associated with a typical irreducible, 4-node tetrahedral linear (**TET4S**) continuum element for scalar field analyses.

- The **SCALAR** keyword is used to specify the number of the scalar material parameter idealization associated with the element. The *default* values for the **SCALAR** number is one (1).

- The optional **SOURCE** keyword is used to specify the number of the scalar source associated with the element. The *default* values for the **SCALAR** number is zero (0).

- The **INITIAL** keyword is used to specify the initial state number associated with the element. The *default* value for the **INITIAL** is zero (0).

- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT SCALAR** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., fluxes) for this element.

29.5 The ELEMENT SCALAR TYPE H8S Command

Synopsis

The **ELEMENT SCALAR TYPE H8S** command is used to describe all irreducible, 8-node trilinear isoparametric hexahedral continuum elements that are to be used in the analysis of scalar field problems.

Syntax

The following syntax is used to describe a typical **H8S** irreducible hexahedral continuum element:

ELEment SCAlar TYPe H8S (DEScription "string")
NODes #:#:#
(SCAlar #) (SOUrce #) (INItial #) (INTcode #)
(1_Additional #) (1_Increment #)
(2_Additional #) (2_Increment #)
(3_Additional #) (3_Increment #)
(DONT_PRINT_Results)

Explanatory Notes

- The **H8S** is an irreducible, trilinear, isoparametric hexahedral continuum element [52]. The element
  - Contains eight (8) vertex nodes.
  - Has one (1) scalar degree of freedom at each node.
  - Possesses a total of eight (8) degrees of freedom.
- The optional string associated with the **DESCRIPTION** keyword is included to facilitate the description of groups of elements present in a particular mesh. Provided that the **ECHO ELEMENT** or the **ECHO ELEMENT_DESCRIPTIONS** commands have not been given, this optional string will be printed along with the remaining **ELEMENT** information.
- The numbering order of **NODES** associated with **H8S** elements, which must be specified sequentially from 1 to 8, is shown in Figure 29.6.

NOTE: Presently APES+ does *not* possess the ability to generate **H8S** elements. It is assumed that the analyst will thus use some stand-alone pre-processing software to accomplish this task. The resulting element and node data will then be translated to the format expected by APES+.

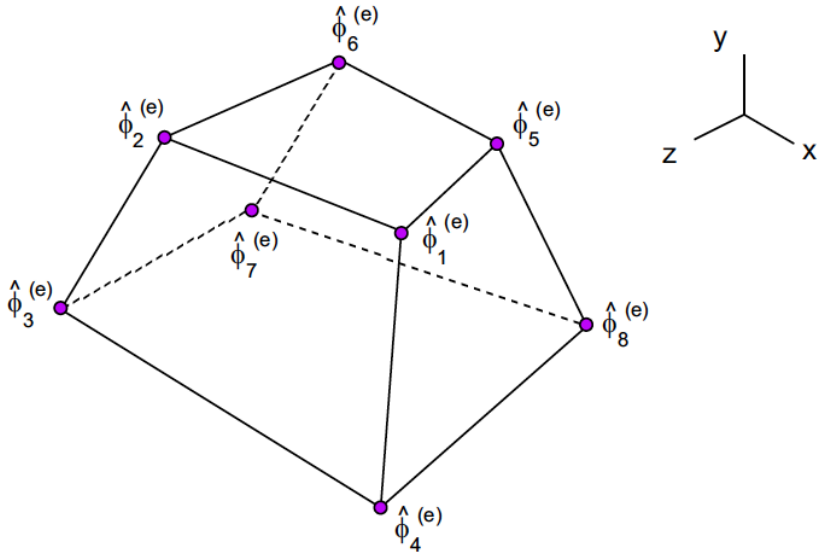

Figure 29.6: Node numbering and degrees of freedom associated with a typical irreducible 8-node hexahedral trilinear (**H8S**) continuum element for scalar field analyses.

- The **SCALAR** keyword is used to specify the number of the scalar material parameter idealization associated with the element. The *default* values for the **SCALAR** number is one (1).
- The optional **SOURCE** keyword is used to specify the number of the scalar source associated with the element. The *default* values for the **SCALAR** number is zero (0).
- The **INITIAL** keyword is used to specify the initial state number associated with the element. The *default* value for the **INITIAL** is zero (0).
- The value specified in conjunction with the **INTCODE** keyword describes the order of numerical integration scheme to be used in developing the element equations for the element.

  The "commonly" used numerical integration rule for **H8S** elements corresponds to a 2 by 2 by 2 Gauss-Legendre quadrature scheme (degree of precision equal to 3) for the primary dependent variables (i.e., nodal displacements) and a 1-point Gauss-Legendre scheme (degree of precision equal to 1) for the secondary dependent variables (i.e., strains and stresses). This is the *default* condition and requires no input using the **INTCODE** keyword. If a quadrature order different from the default condition is desired, the following integer values are associated with this keyword:

INTCODE = 11: A 1-point Gauss-Legendre quadrature scheme (degree of precision equal to 1) is used to compute *both* the primary dependent variables (i.e., nodal displacements) and for the secondary dependent variables (i.e., strains and stresses).

This serves to uniformly *under integrate* the element. The disadvantage of uniform reduced integration is that the rank of the element stiffness matrix $\mathbf{K}^{(e)}$ may be reduced, resulting in the singularity or near singularity of the global $\mathbf{K}$ [52]. The associated element instabilities manifest themselves in the form of *mechanisms* or *hourglass modes* [44]. Consequently, the specification of **INTCODE** = 11 should only be used to investigate the possible pathologies associated with uniform reduced integration and *not* in standard analyses.

INTCODE = 21: a 2 by 2 by 2 Gauss-Legendre quadrature scheme (degree of precision equal to 3) is used to compute the primary dependent variables (i.e., nodal displacements) and a 1-point Gauss-Legendre scheme (degree of precision equal to 1) is used to compute the secondary dependent variables (i.e., strains and stresses). This is equivalent to the aforementioned *default* setting.

If several layers of elements have the *same* attributes, the above generation option can be carried one step further by making use of the **2_ADDITIONAL**, and **2_INCREMENT** keywords, and possibly the **3_ADDITIONAL**, and **3_INCREMENT** keywords.

- The purpose of the **PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. If *generation* is performed using this **ELEMENT SCALAR** command, then all the elements generated will be affected in a like manner by the above print control commands.

- Specification of the **DONT_PRINT_Results** keyword indicates that the analyst does not desire to see output of secondary dependent variables (i.e., fluxes) for this element.

Chapter 30

The MATERIAL Commands

This chapter contains information associated with the description of material idealizations for finite element analyses using the APES$^+$ computer program. To facilitate the development, the following variables are defined [52]:

- N_{sdim} = spatial dimension of the problem (= 1, 2 or 3).
- N_{rowb} = number of stress and strain components.

In light of the above definitions, for three-dimensional analyses ($N_{sdim} = 3$), $N_{rowb} = 6$. For torsionless axisymmetry $N_{sdim} = 2$ and $N_{rowb} = 4$. Finally, for plane strain analyses $N_{sdim} = 2$ and $N_{rowb} = 3$.

30.1 General MATERIAL Information

Synopsis

All material idealizations available in the APES$^+$ computer program are described using the **MATERIAL** commands. At least one material idealization is required in the following types of analyses[1]:

- **ANALYSIS TYPE MECHANICAL**
- **ANALYSIS TYPE MECHANICAL_THERMAL**
- **ANALYSIS TYPE FLOW_MECHANICAL**
- **ANALYSIS TYPE FLOW_MECHANICAL_THERMAL**

The following remarks apply to all material idealizations used in the APES$^+$ computer program:

[1] Section 5.9 gives additional details pertaining to the types of analyses available in APES$^+$.

Remark: The units associated with the material property information must be consistent with those used to describe the geometry of the body being analyzed and with the units of the applied loads and/or displacements.

Remark: For all **MATERIAL** idealizations, the type of material is indicated by the appropriate keywords (e.g., **ELASTIC ISOTROPIC**, **INTERFACE STANDARD**, etc.).

Remark: The material NUMBER serves as an identifier in the assignment of particular material descriptions to groups of elements described using the **ELEMENT** commands.

Remark: Associated with the **DESCRIPTION** keyword is a string that is used to describe the material. This is included solely for the benefit of the analyst; the associated string is printed in the "echo" of the **MATERIAL** data.

30.2 Material Idealizations Available in APES+

The following **MATERIAL** idealizations are available in the APES+ computer program:

30.2.1 Linear Elastic Material Idealizations

The linear elastic material idealizations available for use in the APES+ computer program are listed below. Chapter 31 gives additional details pertaining to these idealizations.

- General **ANISOTROPIC** linear elastic material idealization for solids (see Section 31.3).
- General **ORTHOTROPIC** linear elastic material idealization for solids (see Section 31.4).
- General **ISOTROPIC** linear elastic material idealization for solids (see Section 31.5).
- **POROELASTIC ANISOTROPIC** (generalized "Biot") linear elastic material idealization for porous media (see Section 31.6).
- Isotropic linear elastic ("Herrmann") material idealization for **INCOMPRESSIBLE** and nearly-incompressible solids (see Section 31.7).
- An **ISOTROPIC** linear elastic material idealization for *axial-flexural* (i.e., beam and beam-column type) elements (see Section 31.8).

30.2.2 Quasi-Linear and Variable Modulus Elastic Material Idealizations

The quasi-linear and variable modulus elastic material idealizations available for use in the APES+ computer program are listed below. Chapter 32 gives additional details pertaining to these idealizations.

- Quasi-linear **HYPERBOLIC_ELASTIC** material idealizations for axisymmetric triaxial conditions (see Section 32.5).
- **VARIABLE_MODULUS JANBU** nonlinear hypoelastic material idealization (see Section 32.6).
- **VARIABLE_MODULUS LADE_NELSON** nonlinear hypoelastic material idealization (see Section 32.7).
- **VARIABLE_MODULUS QUBAIN** nonlinear hypoelastic material idealization (see Section 32.8).

30.2.3 Material Idealizations Suitable for Polymeric Inclusions

The material idealizations suitable for simulating the behavior of polymeric inclusions (e.g., geosynthetics) available for use in the APES+ computer program are listed below. Chapter 33 gives additional details pertaining to these idealizations.

- Time-dependent material idealization, after Dechasakulsom and Kaliakin [56], suitable for geosynthetics (see Section 33.2).
- Time-independent material idealization (after Ling and Tatsuoka [82]) suitable for geosynthetics (see Section 33.3).
- Time-independent Ramberg-Osgood [101] material idealization (see Section 33.4).
- Viscoelastic material idealization (see Section 33.5).

30.2.4 Material Idealizations Suitable for Cohesive Soils

The material idealizations suitable for simulating the behavior of cohesive soils that are available for use in the APES+ computer program are listed below.

- ELASTOPLASTIC-VISCOPLASTIC BOUNDING SURFACE material idealization for *isotropic* cohesive soils.

30.2.5 Material Idealizations Suitable for Interface Elements

The material idealizations suitable for use in conjunction with interface elements that are available for use in the APES+ computer program are listed below. Chapter 34 gives additional details pertaining to these idealizations.

- "Standard" material idealization for use with zero-thickness interface elements (see Section 34.2).

Chapter 31

Linear Elastic MATERIAL Idealizations

This chapter contains information associated with the description of linear elastic material idealizations for finite element analyses using the APES$^+$ computer program.

31.1 General Remarks

In a linear elastic material the current value of stress is a function only of the current strain, and is independent of the previous loading/straining history. For a general homogeneous, anisotropic, linear elastic material, the constitutive relations (in "inverse" form) are given by generalized Hooke's law[1]; viz.,

$$\sigma_{ij} = D_{ijkl}\left(\varepsilon_{kl} - \varepsilon^0_{ij}\right) + \sigma^0_{ij} \tag{31.1}$$

where D_{ijkl} represents the fourth-order tensor of elastic moduli, ε^0_{ij} is a second-rank tensor of known initial strains, and σ^0_{ij} is a second-rank tensor of known initial stresses.

Due to the symmetry of the strain and stress tensors, D_{ijkl} possesses *minor* symmetries. In particular, due to the symmetry of σ_{kl}, $D_{ijkl} = D_{jilk}$. In a similar manner, $D_{ijkl} = D_{jikl}$ and $D_{ijkl} = D_{jilk}$. In addition, since the constitutive relation for elasticity can be derived from a strain energy density function, D_{ijkl} possesses *major* symmetries; i.e., $D_{ijkl} = D_{klij}$. As a result, the 81 coefficients associated with the rank four tensor D_{ijkl} reduces to 36, of which only 21 are independent.

The analysis of bodies made of anisotropic materials is thus quite complicated. Fortunately most of the important engineering materials possess some internal structure that exhibits certain symmetries. Such symmetries reduce the number of required coefficients.

[1]Named in honor of Robert Hooke (1635-1703). Hooke was born on the Isle of Wight in England. Following primary education, he was sent to Christ Church, Oxford. In 1662 he took the degree of Master of Arts and, at the recommendation of the famous scientist Robert Boyle, became the curator of experiments for the Royal Society. His excellent knowledge of mechanics and inventive abilities were put to good use in devising apparatus to demonstrate his own ideas, or to illustrate and clarify the points of others. From his experiments, Hooke established the linear relation between the magnitude of forces and the resulting deformations. This linear force-deformation relationship, which is referred to as Hooke's Law, was later used as the foundation upon which further development of the mechanics of elastic bodies was built [110].

Table 31.1: Relation Between Matrix and Tensor Indices

Matrix	Tensor	Tensor	Matrix	Tensor	Tensor
α	i	j	β	k	l
1	1	1	1	1	1
2	2	2	2	2	2
3	3	3	3	3	3
4	1	2	4	1	2
5	1	3	5	1	3
6	2	3	6	2	3

Appendix **??** gives additional details pertaining to the aforementioned material symmetries. Books by Love [83] and Lekhnitskii [81] give additional details pertaining to this subject.

Although the constitutive relations for an elastic material are often written using tensor notation, for purposes of numerical analysis it is more convenient to express these relations in vector/matrix form. As such, it is advantageous to *contract* the indices of tensor quantities appearing in the previous equations. This is achieved by noting that D_{ijkl} has the aforementioned major and minor symmetries. Consequently, the indices associated with the $(6 * 6)$ matrix $D_{\alpha\beta}$ are related to indices of the tensor D_{ijkl} using the Voigt notation[2]. This is realized by the mapping of tensor indices to matrix indices given in Table 31.1.

For a general homogeneous, anisotropic linear elastic (Hookian) material, the constitutive relations, in "inverse" vector-matrix form form, are thus given by

$$\boldsymbol{\sigma} = \mathbf{D}\left(\boldsymbol{\varepsilon} - \boldsymbol{\varepsilon}^{\mathbf{0}}\right) + \boldsymbol{\sigma}^{\mathbf{0}} \tag{31.2}$$

where $\mathbf{D}$ is a symmetric $(N_{rowb} * N_{rowb})$ matrix of elastic coefficients that characterize the material.

Writing Equation (31.2) in expanded form, the inverse form of the constitutive relation for the most general case of a three-dimensional *anisotropic* linear elastic material is

$$\begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{12} \\ \sigma_{13} \\ \sigma_{23} \end{Bmatrix} = \begin{bmatrix} D_{11} & D_{12} & D_{13} & D_{14} & D_{15} & D_{16} \\ D_{12} & D_{22} & D_{23} & D_{24} & D_{25} & D_{26} \\ D_{13} & D_{23} & D_{33} & D_{34} & D_{35} & D_{36} \\ D_{14} & D_{24} & D_{34} & D_{44} & D_{45} & D_{46} \\ D_{15} & D_{25} & D_{35} & D_{45} & D_{55} & D_{56} \\ D_{16} & D_{26} & D_{36} & D_{46} & D_{56} & D_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_{11} - \varepsilon^0_{11} \\ \varepsilon_{22} - \varepsilon^0_{22} \\ \varepsilon_{33} - \varepsilon^0_{33} \\ \gamma_{12} - \gamma^0_{12} \\ \gamma_{13} - \gamma^0_{13} \\ \gamma_{23} - \gamma^0_{23} \end{Bmatrix} + \begin{Bmatrix} \sigma^0_{11} \\ \sigma^0_{22} \\ \sigma^0_{33} \\ \sigma^0_{12} \\ \sigma^0_{13} \\ \sigma^0_{23} \end{Bmatrix} \tag{31.3}$$

In the above expressions, the engineering measure of shearing strains, which is twice the tensorial values, has been employed; i.e., $\gamma_{12} = 2\varepsilon_{12}$, $\gamma_{13} = 2\varepsilon_{13}$, and $\gamma_{23} = 2\varepsilon_{23}$.

For conditions of torsionless axisymmetry, the constitutive relation reduces to

[2]Voigt notation is named after the German physicist Woldemar Voigt (1850-1919). It is a way to represent a symmetric tensor by reducing its order.

$$\begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{12} \end{Bmatrix} = \begin{bmatrix} D_{11} & D_{12} & D_{13} & D_{14} \\ D_{12} & D_{22} & D_{23} & D_{24} \\ D_{13} & D_{23} & D_{33} & D_{34} \\ D_{14} & D_{24} & D_{34} & D_{44} \end{bmatrix} \begin{Bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ \gamma_{12} \end{Bmatrix} \tag{31.4}$$

where the "1", "2", and "3" directions are associated with the radial (r), axial (z), and circumferential (θ) coordinate directions, respectively.

Finally, for conditions of plane stress and plane strain, the constitutive relation further reduce to

$$\begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{12} \end{Bmatrix} = \begin{bmatrix} D_{11} & D_{12} & D_{14} \\ D_{12} & D_{22} & D_{24} \\ D_{14} & D_{24} & D_{44} \end{bmatrix} \begin{Bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \gamma_{12} \end{Bmatrix} \tag{31.5}$$

31.2 Available Linear Elastic Material Idealizations

The following linear elastic material idealizations are currently available in the APES+ computer program:

- General **ANISOTROPIC** linear elastic material idealization for solids (see Section 31.3).
- General **ORTHOTROPIC** linear elastic material idealization for solids (see Section 31.4).
- General **ISOTROPIC** linear elastic material idealization for solids (see Section 31.5).
- An **ISOTROPIC** linear elastic material idealization for *axial-flexural* (i.e., beam and beam-column type) elements (see Section 31.8).
- **POROELASTIC ANISOTROPIC** (generalized "Biot") linear elastic material idealization for porous media (see Section 31.6).
- Isotropic linear elastic ("Herrmann") material idealization for **INCOMPRESSIBLE** and nearly-incompressible solids (see Section 31.7).

Appendix **??** gives additional details pertaining to the aforementioned material symmetries. Books by Love [83] and Lekhnitskii [81] give additional details pertaining to this subject.

31.3 The MATERIAL ELASTIC ANISOTROPIC command

Synopsis

The **MATERIAL ELASTIC ANISOTROPIC** command is used to specify the parameters associated with an anisotropic linear elastic material idealization.

Syntax

The following syntax is associated with the **MATERIAL ELASTIC ANISOTROPIC** command:

MATerial ELAstic ANIsotropic NUMber #
(DEScription "string")
(D11 #.#) (D12 #.#) (D13 #.#) (D14 #.#) (D15 #.#) (D16 #.#)
(D22 #.#) (D23 #.#) (D24 #.#) (D25 #.#) (D26 #.#)
(D33 #.#) (D34 #.#) (D35 #.#) (D36 #.#)
(D44 #.#) (D45 #.#) (D46 #.#)
(D55 #.#) (D56 #.#) (D66 #.#)

Explanatory Notes

- The **NUMBER** keyword is used to specify the (global) number of the material associated with the incompressible isotropic elastic idealization. The default material number is one (1).
- The optional alphanumeric string associated with the **DESCRIPTION** keyword must be enclosed in double quotes ("). It is used solely to describe the material being idealized to the analyst. The **DESCRIPTION** string is printed as part of the "echo" of the material.

Orthotropic Elastic Material Idealization: For a material through each point of which pass *three* mutually perpendicular planes of elastic symmetry. If similar planes are parallel at all points in the material, then taking (x_1, x_2, x_3) coordinate axes normal to these planes (i.e., along the principal directions) it follows that there should be no interaction between the various shear components or between the shear and normal components. The material is described by twelve elastic constants, only *nine* of which are independent. The entries in the three-dimensional constitutive matrix now become

$$D_{11} = \frac{E_1(1-\nu_{23}\nu_{32})}{\mathcal{D}} \quad ; \quad D_{12} = D_{21} = \frac{E_1(\nu_{21}+\nu_{31}\nu_{23})}{\mathcal{D}}$$

$$D_{13} = C_{31} = \frac{E_1(\nu_{31}+\nu_{21}\nu_{32})}{\mathcal{D}} \quad ; \quad D_{22} = \frac{E_2(1-\nu_{13}\nu_{31})}{\mathcal{D}}$$

$$D_{23} = C_{32} = \frac{E_2(\nu_{32}+\nu_{12}\nu_{31})}{\mathcal{D}} \quad ; \quad D_{33} = \frac{E_3(1-\nu_{12}\nu_{21})}{\mathcal{D}}$$

$$D_{44} = G_{12} \quad ; \quad D_{55} = G_{13} \quad ; \quad D_{66} = G_{23}$$

where $\mathcal{D} = 1 - \nu_{12}\nu_{21} - \nu_{13}\nu_{31} - \nu_{23}\nu_{32} - 2\nu_{12}\nu_{31}\nu_{23}$.

Remark: Although the **MATERIAL ELASTIC ANISOTROPIC** command can be used to directly specify suitable values for the D_{11}, D_{12}, etc., the **MATERIAL ELASTIC ORTHOTROPIC** command is easier to use (see Section 31.4). That is, sub-commands associated with the latter command involve specification of nine (or seven) elastic constants directly - the requisite calculations of the D_{11}, D_{12}, etc. are then performed by the APES$^+$ computer program. ◀

Transversely Isotropic Elastic Material Idealization: Such materials possess the following properties: through all points there pass parallel planes of elastic symmetry in which all directions are elastically equivalent (i.e., planes of isotropy). Thus at each point there exists one principal direction and an infinite number of principal directions in a plane normal to the first direction. Five parameter values characterize a transversely isotropic elastic material. Taking the x_3-axis normal to the plane of isotropy, with the x_1 and x_2 axes directed arbitrarily in this plane, these are

- E_1 is the elastic modulus for compression or tension in the plane of isotropy.
- E_2 is the elastic modulus for compression or tension in a direction *normal* to the plane of isotropy.
- ν_{11} is Poisson's ratio[3] characterizing transverse contraction in the plane of isotropy when tension is applied in the plane of isotropy.
- ν_{21} is Poisson's ratio characterizing *transverse* contraction when tension is applied in the plane of isotropy. Similarly, ν_{12} is Poisson's ratio characterizing contraction *normal* to the plane of isotropy when tension is applied in this plane.
- G_2 is the shear modulus associated with the plane of isotropy and any plane perpendicular to it.

[3]Named in honor of S. D. Poisson (1781-1840).

The non-zero entries in the three-dimensional constitutive matrix are thus

$$D_{11} = D_{22} = \frac{E_1\,[E_2 - E_1(\nu_{21})^2]}{(1+\nu_{11})\,[E_2(1-\nu_{11}) - 2E_1(\nu_{21})^2]}$$

$$D_{12} = \frac{E_1\,[E_1(\nu_{21})^2 + E_2\nu_{11}]}{(1+\nu_{11})\,[E_2(1-\nu_{11}) - 2E_1(\nu_{21})^2]}$$

$$D_{13} = D_{23} = \frac{E_1 E_2 \nu_{21}}{E_2(1-\nu_{11}) - 2E_1(\nu_{21})^2}$$

$$D_{33} = \frac{(E_2)^2\,(1-\nu_{11})}{E_2(1-\nu_{11}) - 2E_1(\nu_{21})^2}$$

$$D_{44} = \frac{E_1}{2(1+\nu_{11})} \quad ; \quad D_{55} = D_{66} = G_2$$

Isotropic Elastic Material Idealization: The simplest linearly elastic material, for which the elastic behavior is *independent* of the orientation of the coordinate axes, is called *isotropic*. For such a material the constitutive relations simplify in that only *two* material constants are required to completely describe the material behavior; e.g., the elastic (Young's) modulus E and Poisson's ratio ν, the Lamé constants λ and μ, or the bulk modulus K and the shear modulus G. The entries in the three-dimensional constitutive matrix now become

$$D_{11} = D_{22} = D_{33} = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} = \lambda + 2\mu = K + \frac{4}{3}G$$

$$D_{12} = D_{13} = D_{23} = \frac{E\nu}{(1+\nu)(1-2\nu)} = \lambda = K - \frac{2}{3}G$$

$$D_{44} = D_{55} = D_{66} = \frac{E}{2(1+\nu)} = \mu = G$$

Remark: Although the **MATERIAL ELASTIC ANISOTROPIC** command can be used to directly specify suitable values for the D_{11}, D_{12}, etc., the **MATERIAL ELASTIC ISOTROPIC** command is easier to use (see Section 31.5). That is, sub-commands associated with the latter command involve specification of nine (or seven) elastic constants directly - the requisite calculations of the D_{11}, D_{12}, etc. are then performed by the APES+ computer program. ◀

Example of Command Usage

Idealization of an Anisotropic Elastic Material

The following commands are sufficient to describe the anisotropic elastic material idealization

```
material elastic aniso num 2 desc ‘‘hypothetical hybrid material’’  &
   c11 4.038e+07  c12 1.731e+07  c13 1.731e+07  c14 0.0  &
   c22 4.038e+07  c23 1.731e+07  &
   c24 0.0   c33 4.038e+07  c34 0.0   c44 1.154e+07
```

Idealization of Wood (an Orthotropic Elastic Material)

In this example the linear anisotropic material idealization is used to describe an *orthotropic* material (Douglas Fir). Twelve parameters, nine of which are independent, characterize an orthotropic material. The three principal axes of material symmetry are oriented with respect to the grain directions and growth rings [91]. Denoting the longitudinal, tangential and radial directions by the letters "L", "T", and "R", respectively, the following values are used to characterize Douglas Fir at approximately 12% moisture content: [91]

- $E_T/E_L = 0.050$
- $E_R/E_L = 0.068$
- $\nu_{LR} = 0.292 \quad \nu_{RL} = 0.036$
- $\nu_{LT} = 0.292 \quad \nu_{TL} = 0.029$
- $\nu_{RT} = 0.390 \quad \nu_{TR} = 0.374$
- $G_{LR}/E_L = 0.064$
- $G_{LT}/E_L = 0.078$
- $G_{RT}/E_L = 0.007$

where $E_L = 12,600\,MPa$ (an average value, obtained from bending tests). Denoting the longitudinal, transverse and radial directions by "1", "2", and "3", respectively, it follows that

$$
\begin{aligned}
E_1 &= 12,600\,MPa \\
E_2 &= 0.050(12,600) = 630.0\,MPa \\
E_3 &= 0.068(12,600) = 856.8\,MPa
\end{aligned}
$$

and

$$G_{12} = 0.078(12,600) = 982.8\,MPa$$
$$G_{13} = 0.064(12,600) = 806.4\,MPa$$
$$G_{23} = 0.007(12,600) = 88.2\,MPa$$

Finally, since $\nu_{12} = 0.449$, $\nu_{13} = 0.292$, and $\nu_{23} = 0.390$, it follows that

$$\nu_{21} = \frac{E_2\nu_{12}}{E_1} = (0.050)(0.449) = 0.02245$$
$$\nu_{31} = \frac{E_3\nu_{13}}{E_1} = 0.068(0.292) = 0.01986$$
$$\nu_{32} = \frac{E_3\nu_{23}}{E_2} = \left(\frac{0.068}{0.050}\right)(0.390) = 0.5304$$

For an orthotropic material the entries in $\mathbf{D}$ are computed as follows:

$$\mathcal{D} = 1 - \nu_{12}\nu_{21} - \nu_{13}\nu_{31} - \nu_{23}\nu_{32} - \nu_{12}\nu_{31}\nu_{23} - \nu_{21}\nu_{13}\nu_{32}$$
$$D_{11} = \frac{E_1(1-\nu_{23}\nu_{32})}{\mathcal{D}}, \quad D_{12} = \frac{E_1(\nu_{21}+\nu_{31}\nu_{23})}{\mathcal{D}}, \quad D_{13} = \frac{E_1(\nu_{31}+\nu_{21}\nu_{32})}{\mathcal{D}}$$
$$D_{21} = \frac{E_2(\nu_{12}+\nu_{13}\nu_{32})}{\mathcal{D}}, \quad D_{22} = \frac{E_2(1-\nu_{13}\nu_{31})}{\mathcal{D}}, \quad D_{23} = \frac{E_2(\nu_{32}+\nu_{12}\nu_{31})}{\mathcal{D}}$$
$$D_{31} = \frac{E_3(\nu_{13}+\nu_{12}\nu_{23})}{\mathcal{D}}, \quad D_{32} = \frac{E_3(\nu_{23}+\nu_{21}\nu_{13})}{\mathcal{D}}, \quad D_{33} = \frac{E_3(1-\nu_{12}\nu_{21})}{\mathcal{D}}$$
$$D_{44} = G_{12} \quad ; \quad D_{55} = G_{13} \quad ; \quad D_{66} = G_{23}$$

Thus, for the present material idealization

$$\mathcal{D} = 1 - (0.449)(0.02245) - (0.292)(0.01986) - (0.390)(0.5304) - 2(0.449)(0.01986)(0.390) = 0.7703$$

and

$$C_{11} = 12972.150\,MPa$$
$$C_{12} = 493.908\,MPa$$
$$C_{13} = 519.623\,MPa$$
$$C_{22} = 813.111\,MPa$$
$$C_{23} = 441.082\,MPa$$
$$C_{33} = 1101.069\,MPa$$
$$C_{44} = G_{12} = 0.078(12,600) = 982.8\,MPa$$
$$C_{55} = G_{13} = 0.064(12,600) = 806.4\,MPa$$
$$C_{66} = G_{23} = 0.007(12,600) = 88.2\,MPa$$

The following commands are sufficient to describe the orthotropic elastic material idealization:

```
material elastic aniso num 1 &
   desc ‘‘orthotropic material entered using anisotropic command’’  &
   c11 12972.150  c12 493.908  c13 519.623   &
   c22 813.111   c23 441.082  c33  1101.069 &
   c44 982.8   c55 806.4   c66 88.2
```

Idealization of an Isotropic Elastic Material

In this example the linear anisotropic material idealization is used to describe an *isotropic* material. Two parameters characterize such a material; in this case an elastic modulus $E = 20,000$ and a Poisson's ratio of $\nu = 0.25$. The entries in $\mathbf{C}$ for an isotropic material are computed as follows:

$$C_{11} = C_{22} = C_{33} = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} = 24,000$$
$$C_{12} = C_{13} = C_{23} = \frac{E\nu}{(1+\nu)(1-2\nu)} = 8,000$$
$$C_{44} = C_{55} = C_{66} = \frac{E}{2(1+\nu)} = 8,000$$

The following commands are sufficient to describe the isotropic elastic material idealization:

```
material elastic aniso num 1 &
   desc ‘‘isotropic material entered using anisotropic command’’  &
   c11 24000.0   c12 8000.0   c13  8000.0  &
   c22 24000.0   c23 8000.0   c33  24000.0 &
   c44 8000.0    c55 8000.0   c66  8000.0
```

31.4 The MATERIAL ELASTIC ORTHOTROPIC command

Synopsis

The **MATERIAL ELASTIC ORTHOTROPIC** command is used to specify the parameters associated with an orthotropic linear elastic material idealization. Although the same parameters can be specified via the **MATERIAL ELASTIC ANISOTROPIC** command, the **MATERIAL ELASTIC ORTHOTROPIC** command simplifies the description of an orthotropic elastic material.

Syntax

The following syntax is associated with the **MATERIAL ELASTIC ORTHOTROPIC** command:

MATerial ELAstic ORThotropic NUMber #
(DEScription "string")
(1_Modulus #.#) (2_Modulus #.#) (3_Modulus #.#)
(12_Ratio #.#) (13_Ratio #.#) (23_Ratio #.#)
(12_Shear #.#) (13_Shear #.#) (23_Shear #.#)

Explanatory Notes

For the special case of an anisotropic material possessing three planes of elastic symmetry, the material constants can be specified using the **MATERIAL ELASTIC ORTHOTROPIC** command in lieu of the **MATERIAL ELASTIC ANISOTROPIC** command.

- The **NUMBER** keyword is used to specify the (global) number of the material associated with the incompressible isotropic elastic idealization. The default material number is one (1).
- The optional alphanumeric string associated with the **DESCRIPTION** keyword must be enclosed in double quotes ("). It is used solely to describe the material being idealized to the analyst. The **DESCRIPTION** string is printed as part of the "echo" of the material.
- The default values for **1_MODULUS**, for **2_MODULUS**, **3_MODULUS**, **12_SHEAR**, **13_SHEAR**, and for **23_SHEAR** are 1.0. The default values for **12_RATIO**, **13_RATIO**,

and for **23_RATIO** are zero (0.0) If the principal directions of the anisotropic material differ from the global ones, suitable transformations must be performed using the **TRANSFORM** command.

- For a three-dimensional state of stress the constitutive relation associated with an orthotropic elastic material is as follows:

$$\begin{Bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ \gamma_{12} \\ \gamma_{13} \\ \gamma_{23} \end{Bmatrix} = \begin{bmatrix} A_{11} & A_{12} & A_{13} & 0 & 0 & 0 \\ A_{21} & A_{22} & A_{23} & 0 & 0 & 0 \\ A_{31} & A_{32} & A_{33} & 0 & 0 & 0 \\ 0 & 0 & 0 & A_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & A_{55} & 0 \\ 0 & 0 & 0 & 0 & 0 & A_{66} \end{bmatrix} \begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{12} \\ \sigma_{13} \\ \sigma_{23} \end{Bmatrix}$$

where the non-zero compliance coefficients are

$$A_{11} = \frac{1}{E_1}, \quad A_{12} = -\frac{\nu_{21}}{E_2}, \quad A_{13} = -\frac{\nu_{31}}{E_3}$$
$$A_{21} = -\frac{\nu_{12}}{E_1}, \quad A_{22} = \frac{1}{E_2}, \quad A_{23} = -\frac{\nu_{32}}{E_3}$$
$$A_{31} = -\frac{\nu_{13}}{E_1}, \quad A_{32} = -\frac{\nu_{23}}{E_2}, \quad A_{33} = \frac{1}{E_3}$$
$$A_{44} = \frac{1}{G_{12}}, \quad A_{55} = \frac{1}{G_{13}}, \quad A_{66} = \frac{1}{G_{23}}$$

- For conditions of plane stress, plane strain and torsionless axisymmetry, the constitutive law reduces to

$$\begin{Bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ \gamma_{12} \end{Bmatrix} = \begin{bmatrix} A_{11} & A_{12} & A_{13} & 0 \\ A_{12} & A_{22} & A_{23} & 0 \\ A_{13} & A_{23} & A_{33} & 0 \\ 0 & 0 & 0 & A_{44} \end{bmatrix} \begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{12} \end{Bmatrix}$$

where loading has been assumed to be applied only in the 1-2 plane. Note that in the above expressions, the engineering measure of shearing strains, which is twice the tensorial values, has been employed; i.e., $\gamma_{12} = 2\varepsilon_{12}$, $\gamma_{13} = 2\varepsilon_{13}$, and $\gamma_{23} = 2\varepsilon_{23}$.

Symmetry of **A** implies that

$$A_{12} = A_{21} \quad \Rightarrow \quad \frac{\nu_{21}}{E_2} = \frac{\nu_{12}}{E_1} \quad \text{or} \quad E_1\nu_{21} = E_2\nu_{12}$$
$$A_{13} = A_{31} \quad \Rightarrow \quad \frac{\nu_{31}}{E_3} = \frac{\nu_{13}}{E_1} \quad \text{or} \quad E_1\nu_{31} = E_3\nu_{13}$$
$$A_{23} = A_{32} \quad \Rightarrow \quad \frac{\nu_{32}}{E_3} = \frac{\nu_{23}}{E_2} \quad \text{or} \quad E_2\nu_{32} = E_3\nu_{23}$$

Thus, only *nine* of the twelve elastic constants entering the above equations are independent.

- Inverting **A** gives the following non-zero entries in **D**:

$$D_{11} = \frac{E_1(1-\nu_{23}\nu_{32})}{\mathcal{D}}, \quad D_{12} = \frac{E_1(\nu_{21}+\nu_{31}\nu_{23})}{\mathcal{D}}, \quad D_{13} = \frac{E_1(\nu_{31}+\nu_{21}\nu_{32})}{\mathcal{D}}$$

$$D_{21} = \frac{E_2(\nu_{12}+\nu_{13}\nu_{32})}{\mathcal{D}}, \quad D_{22} = \frac{E_2(1-\nu_{13}\nu_{31})}{\mathcal{D}}, \quad D_{23} = \frac{E_2(\nu_{32}+\nu_{12}\nu_{31})}{\mathcal{D}}$$

$$D_{31} = \frac{E_3(\nu_{13}+\nu_{12}\nu_{23})}{\mathcal{D}}, \quad D_{32} = \frac{E_3(\nu_{23}+\nu_{21}\nu_{13})}{\mathcal{D}}, \quad D_{33} = \frac{E_3(1-\nu_{12}\nu_{21})}{\mathcal{D}}$$

$$D_{44} = G_{12} \quad ; \quad D_{55} = G_{13} \quad ; \quad D_{66} = G_{23}$$

where $\mathcal{D} = 1 - \nu_{12}\nu_{21} - \nu_{13}\nu_{31} - \nu_{23}\nu_{32} - \nu_{12}\nu_{31}\nu_{23} - \nu_{21}\nu_{13}\nu_{32}$. Finally, noting that

$$\nu_{21}\nu_{13}\nu_{32} = \left(\frac{E_2}{E_1}\nu_{12}\right)\left(\frac{E_1}{E_3}\nu_{31}\right)\left(\frac{E_3}{E_2}\nu_{23}\right) = \nu_{12}\nu_{31}\nu_{23}$$

gives $\mathcal{D} = 1 - \nu_{12}\nu_{21} - \nu_{13}\nu_{31} - \nu_{23}\nu_{32} - 2\nu_{12}\nu_{31}\nu_{23}$. Symmetry of **D** implies that

$$E_1(\nu_{21}+\nu_{31}\nu_{23}) = E_2(\nu_{12}+\nu_{13}\nu_{32})$$

$$E_1(\nu_{31}+\nu_{21}\nu_{32}) = E_3(\nu_{13}+\nu_{12}\nu_{23})$$

$$E_2(\nu_{32}+\nu_{12}\nu_{31}) = E_3(\nu_{23}+\nu_{21}\nu_{13})$$

Additional details pertaining to the finite element analysis of elastic bodies are given in Chapter 12 of [52].

Example of Command Usage

Idealization of a Single Orthotropic Elastic Material

Figure 31.1 schematically shows a sample of engineered wood (i.e., laminated veneer lumber). The following orthotropic elastic material idealization was used in simulating the behavior of this material [14]:

```
mat elastic orthotropic num 1  desc "laminated wood material (LVL)" &
     1_mod  1.40e+07  2_mod 6.087e+05  3_mod  6.176e+05   &
    12_ratio  0.460   13_ratio  0.340   23_ratio  0.424   &
    12_shear  8.60e+05
```

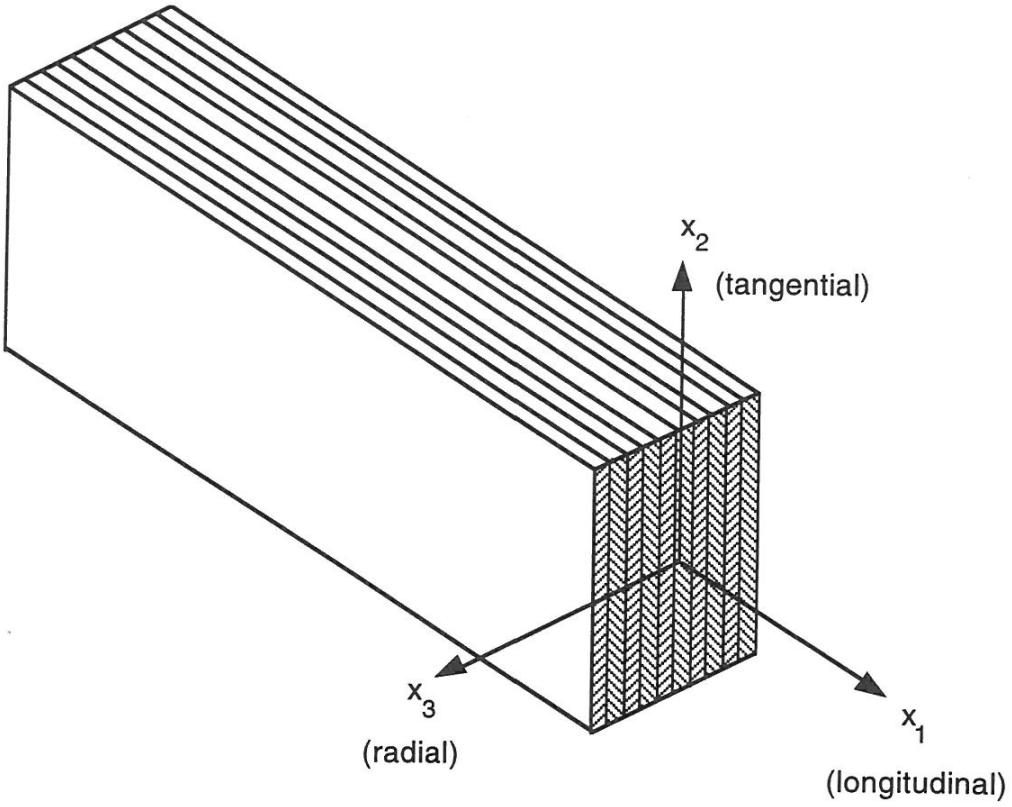

Figure 31.1: Schematic illustration of Laminated Veneer Lumber.

31.5 The MATERIAL ELASTIC ISOTROPIC command

Synopsis

The simplest linearly elastic material, for which the elastic behavior is independent of the orientation of the coordinate axes, is called an *isotropic* material (a more precise definition would be "a material whose constitutive relations are unaltered under orthogonal transformations of coordinates"). For an isotropic linear elastic material the constitutive relations simplify in that only *two* constants are required to completely describe the material behavior; i.e., either the Lamé constants λ and μ, or the elastic (Young's) modulus E and Poisson's ratio ν.

The **MATERIAL ELASTIC ISOTROPIC** command is used to specify the parameters associated with an isotropic linear elastic material idealization.

Syntax

The following syntax is associated with the **MATERIAL ELASTIC ISOTROPIC** command:

MATerial ELAstic ISOtropic NUMber #
(DEScription "string")
(MODulus #.#) (POIssons_ratio #.#)
(BULk_modulus #.#) (SHEar_modulus #.#)
(MU #.#) (LAMbda #.#)

Explanatory Notes

As noted above, isotropic linear elastic material idealizations require the values of *two* material parameters. In "engineering" applications, these are typically the elastic modulus E and Poisson's ratio ν. Alternatively, the elastic bulk modulus K and shear modulus G can be used to characterize the material. Finally, one may likewise use the Lamé parameters[4] λ and μ, where the latter is identical to G.

- The **NUMBER** keyword is used to specify the (global) number of the material associated with the isotropic linear elastic idealization. The default material number is one (1).

[4]Named in honor of Gabriel Lamé (1795-1870).

- The optional alphanumeric string associated with the **DESCRIPTION** keyword must be enclosed in double quotes ("). It is used solely to describe the material being idealized to the analyst. The **DESCRIPTION** string is printed as part of the "echo" of the material.

- The keyword **MODULUS** is used to specify the value of the elastic modulus E. The default value is equal to 3.0e+05.

- The keyword **POISSONS_RATIO** is used to specify the value of the Poisson's ratio ν. The default value is equal to 0.30.

- The keyword **BULK_MODULUS** is used to specify the value of the elastic bulk modulus K. The default value is equal to 2.50e+05.

- The keyword **SHEAR_MODULUS** is used to specify the value of the elastic shear modulus $G = \mu$. The default value is equal to 1.150e+05.

- The keyword **LAMBDA** is used to specify the value of the Lamé parameter λ. The default value is equal to 1.730e+05.

- The keyword **MU** is used to specify the value of the Lamé parameter $\mu = G$. The default value is equal to 1.150e+05.

Example of Command Usage

Idealization of a Single Isotropic Elastic Material

The following commands are sufficient to specify an isotropic elastic material with $E = 20.6 \times 10^5$ and $\nu = 0.25$:

```
mat elas isotropic num 1  des "compressible material idealization" &
     modulus 20.6e+05  poissons  0.25
```

Idealization of Two Isotropic Elastic Materials

```
mat elastic isotropic number 2  desc "CWF concrete"  &
        modulus  2.57e+07  poissons  0.20
!
mat elas iso num 3  des "FRP plate material"  mod  1.240e+08
```

Material number 2 can likewise be described using the following equivalent commands

```
mat elastic isotropic number 2  desc "CWF concrete"  &
        lambda  7.139e+06  mu  1.071e+07
```

or

```
mat elastic isotropic number 2  desc "CWF concrete"  &
        bulk_mod  1.428e+07  shear_mod  1.071e+07
```

Theoretical Considerations

Written in "inverse" form, the general form of the isotropic linear elastic constitutive relation is

$$\boldsymbol{\sigma} = \mathbf{D}\left(\boldsymbol{\varepsilon} - \boldsymbol{\varepsilon}^{\mathbf{0}}\right) + \boldsymbol{\sigma}^{\mathbf{0}} \tag{31.6}$$

where $\mathbf{D}$ is a symmetric $(N_{rowb} * N_{rowb})$ matrix of elastic coefficients that characterize the material, $\boldsymbol{\varepsilon}^{\mathbf{0}}$ is an $(N_{rowb} * 1)$ vector of initial strain components, and $\boldsymbol{\sigma}^{\mathbf{0}}$ is an $(N_{rowb} * 1)$ vector of initial stress components.

For a three-dimensional state of stress and strain,

$$\boldsymbol{\sigma} = \left\{\sigma_{11} \quad \sigma_{22} \quad \sigma_{33} \quad \sigma_{12} \quad \sigma_{13} \quad \sigma_{23}\right\}^T \tag{31.7}$$

and

$$\boldsymbol{\varepsilon} = \left\{\varepsilon_{11} \quad \varepsilon_{22} \quad \varepsilon_{33} \quad \gamma_{12} \quad \gamma_{13} \quad \gamma_{23}\right\}^T \tag{31.8}$$

The associated matrix of constitutive (stress-strain) parameters has the following non-zero entries:

$$D_{11} = D_{22} = D_{33} = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} = \lambda + 2\mu = K + \frac{4}{3}G \tag{31.9}$$

$$D_{12} = D_{13} = D_{23} = \frac{E\nu}{(1+\nu)(1-2\nu)} = \lambda = K - \frac{2}{3}G \tag{31.10}$$

$$D_{44} = D_{55} = D_{66} = \frac{E}{2(1+\nu)} = \mu = G \tag{31.11}$$

For *isotropic* elastic idealizations, the various material constants are related in the following manner:

$$\lambda = \frac{2\mu\nu}{1-2\nu} = \frac{\mu(E-2\mu)}{3-E} = K - \frac{2}{3}\mu = \frac{E\nu}{(1+\nu)(1-2\nu)}$$
$$= \frac{3K\nu}{1+\nu} = \frac{3K(3K-E)}{9K-E}$$

$$\mu = G = \frac{\lambda(1-2\nu)}{2\nu} = \frac{3}{2}(K-\lambda) = \frac{E}{2(1+\nu)}$$
$$= \frac{3K(1-2\nu)}{2(1+\nu)} = \frac{3KE}{9K-E}$$

$$\nu = \frac{\lambda}{2(\lambda+\mu)} = \frac{\lambda}{(3K-\lambda)} = \frac{E}{2\mu} - 1 = \frac{3K-2\mu}{2(3K+\mu)} = \frac{3K-E}{6K}$$
$$= -\frac{1}{4}\left(\frac{E}{\lambda}+1\right) \pm \frac{1}{4}\sqrt{\left(\frac{E}{\lambda}+1\right)^2 + 8}$$

$$E = \frac{\mu(3\lambda + 2\mu)}{\lambda + \mu} = \frac{\lambda(1+\nu)(1-2\nu)}{\nu} = \frac{9K(K-\lambda)}{3K-\lambda} = 2\mu(1+\nu)$$
$$= \frac{9K\mu}{3K+\mu} = 3K(1-2\nu)$$

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

Written in terms of E and ν, the matrix of elastic moduli is

$$\mathbf{D} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & \nu & 0 & 0 & 0 \\ \nu & 1-\nu & \nu & 0 & 0 & 0 \\ \nu & \nu & 1-\nu & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{2}(1-2\nu) & 0 & \\ 0 & 0 & 0 & 0 & \frac{1}{2}(1-2\nu) & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{2}(1-2\nu) \end{bmatrix} \tag{31.12}$$

Written in terms the Lamé constants, this matrix becomes

$$\mathbf{D} = \begin{bmatrix} \lambda+2\mu & \lambda & \lambda & 0 & 0 & 0 \\ \lambda & \lambda+2\mu & \lambda & 0 & 0 & 0 \\ \lambda & \lambda & \lambda+2\mu & 0 & 0 & 0 \\ 0 & 0 & 0 & \mu & 0 & 0 \\ 0 & 0 & 0 & 0 & \mu & 0 \\ 0 & 0 & 0 & 0 & 0 & \mu \end{bmatrix} \tag{31.13}$$

Finally, written in terms the elastic bulk (K) and shear (G) moduli, this matrix becomes

$$\mathbf{D} = \begin{bmatrix} K+\frac{4G}{3} & K-\frac{2G}{3} & K-\frac{2G}{3} & 0 & 0 & 0 \\ K-\frac{2G}{3} & K+\frac{4G}{3} & K-\frac{2G}{3} & 0 & 0 & 0 \\ K-\frac{2G}{3} & K-\frac{2G}{3} & K+\frac{4G}{3} & 0 & 0 & 0 \\ 0 & 0 & 0 & G & 0 & 0 \\ 0 & 0 & 0 & 0 & G & 0 \\ 0 & 0 & 0 & 0 & 0 & G \end{bmatrix} \tag{31.14}$$

where it is noted that now

$$D_{44} = D_{55} = D_{66} = \frac{1}{2}\left(D_{11} - D_{12}\right) \tag{31.15}$$

Additional details pertaining to the finite element analysis of elastic bodies under static or quasi-static conditions are given in Chapter 12 of [52].

31.6 The MATERIAL POROELASTIC ANISOTROPIC command

Synopsis

The **MATERIAL POROELASTIC ANISOTROPIC** command is used to specify the parameters associated with an anisotropic linear poroelastic material idealization based on the theory of Biot [9; 10; 11]. Additional details about such a material characterization can be found in, for example, Detournay and Cheng [27].

Syntax

The following syntax is associated with the **MATERIAL POROELASTIC ANISOTROPIC** command:

MATerial PORoelastic ANIsotropic NUMber ## (DEScription "string")
(C11 #.#) (C12 #.#) (C13 #.#) (C14 #.#) (C15 #.#) (C16 #.#)
(C22 #.#) (C23 #.#) (C24 #.#) (C25 #.#) (C26 #.#)
(C33 #.#) (C34 #.#) (C35 #.#) (C36 #.#)
(C44 #.#) (C45 #.#) (C46 #.#)
(C55 #.#) (C56 #.#) (C66 #.#)
(ALPHA_11 #.#) (ALPHA_22 #.#) (ALPHA_33 #.#)
(ALPHA_12 #.#) (ALPHA_13 #.#) (ALPHA_23 #.#)
(M_Modulus #.#)

Explanatory Notes

The keywords associated with the **MATERIAL POROELASTIC ANISOTROPIC** command have the following meaning:

NUMBER: The integer associated with this keyword is used to specify the (global) number assigned to the material for which the present idealization shall be applied. The default material number is one (1).

DESCRIPTION: The optional alphanumeric string associated with the **DESCRIPTION** keyword must be enclosed in double quotes ("). It is used solely to describe the material being idealized to the analyst. The **DESCRIPTION** string is printed as part of the "echo" of the material.

C11 $\cdots$ C66: The real numbers associated with these keyword represents the values of the entries in the constitutive matrix $\mathbf{C}$ (units of FL^{-2}) that relates the vector of approximate incremental effective stress components $\boldsymbol{\Delta\hat{\sigma}'}$ to the vector of approximate incremental strain components $\boldsymbol{\Delta\hat{\varepsilon}}$ according to

$$\boldsymbol{\Delta\hat{\sigma}'} = \mathbf{C}\boldsymbol{\Delta\hat{\varepsilon}}$$

For three-dimensional idealizations, the vectors of element effective stress and strain increments are given by

$$\boldsymbol{\Delta\sigma} = \left\{ \Delta\sigma_{11}^{(e)\prime} \quad \Delta\sigma_{22}^{(e)\prime} \quad \Delta\sigma_{33}^{(e)\prime} \quad \Delta\sigma_{12}^{(e)} \quad \Delta\sigma_{13}^{(e)\prime} \quad \Delta\sigma_{23}^{(e)} \right\}^T$$

$$\boldsymbol{\Delta\hat{\varepsilon}^{(e)}} = \left\{ \Delta\varepsilon_{11}^{(e)} \quad \Delta\varepsilon_{22}^{(e)} \quad \Delta\varepsilon_{33}^{(e)} \quad \Delta\gamma_{12}^{(e)} \quad \Delta\gamma_{13}^{(e)} \quad \Delta\gamma_{23}^{(e)} \right\}^T$$

where $\gamma_{12}^{(e)} = 2\varepsilon_{12}^{(e)}$, $\gamma_{13}^{(e)} = 2\varepsilon_{13}^{(e)}$ and $\gamma_{23}^{(e)} = 2\varepsilon_{23}^{(e)}$ are engineering measures of shear strain. It follows that

$$\mathbf{C} = \begin{bmatrix} C_{11} & C_{12} & C_{13} & C_{14} & C_{15} & C_{16} \\ C_{12} & C_{22} & C_{23} & C_{24} & C_{25} & C_{26} \\ C_{13} & C_{23} & C_{33} & C_{34} & C_{35} & C_{36} \\ C_{14} & C_{24} & C_{34} & C_{44} & C_{45} & C_{46} \\ C_{15} & C_{25} & C_{35} & C_{45} & C_{55} & C_{56} \\ C_{16} & C_{26} & C_{36} & C_{46} & C_{56} & C_{66} \end{bmatrix}$$

For two-dimensional *plane strain* or *plane stress* idealizations, the vectors of element effective stress and strain increments are given by

$$\boldsymbol{\Delta\sigma^{(e)\prime}} = \left\{ \Delta\sigma_{11}^{(e)\prime} \quad \Delta\sigma_{22}^{(e)\prime} \quad \Delta\sigma_{12}^{(e)} \right\}^T$$

$$\boldsymbol{\Delta\hat{\varepsilon}^{(e)}} = \left\{ \Delta\varepsilon_{11}^{(e)} \quad \Delta\varepsilon_{22}^{(e)} \quad \Delta\gamma_{12}^{(e)} \right\}^T$$

It follows that

$$\mathbf{C} = \begin{bmatrix} C_{11} & C_{12} & C_{13} \\ C_{12} & C_{22} & C_{23} \\ C_{13} & C_{23} & C_{33} \end{bmatrix}$$

For *torsionless axisymmetric* idealizations, the vectors of element stress and strain increments are given by

$$\boldsymbol{\Delta\hat{\sigma}}^{(e)\prime} = \left\{ \Delta\sigma_{11}^{(e)\prime} \quad \Delta\sigma_{22}^{(e)\prime} \quad \Delta\sigma_{33}^{(e)\prime} \quad \Delta\sigma_{12}^{(e)} \right\}^T$$

$$\boldsymbol{\Delta\hat{\varepsilon}}^{(e)} = \left\{ \Delta\varepsilon_{11}^{(e)} \quad \Delta\varepsilon_{22}^{(e)} \quad \Delta\varepsilon_{33}^{(e)} \quad \Delta\gamma_{12}^{(e)} \right\}^T$$

where $\Delta\sigma_{11}^{(e)\prime}$, $\Delta\sigma_{22}^{(e)\prime}$ and $\Delta\sigma_{33}^{(e)\prime}$ correspond to the *radial*, *axial* and *circumferential* directions, respectively. It follows that

$$\mathbf{C} = \begin{bmatrix} C_{11} & C_{12} & C_{13} & C_{14} \\ C_{12} & C_{22} & C_{23} & C_{24} \\ C_{13} & C_{23} & C_{33} & C_{34} \\ C_{14} & C_{24} & C_{34} & C_{44} \end{bmatrix}$$

The default value of C_{11}, C_{22}, C_{33}, C_{44}, C_{55} and C_{66} is one (1.0). For all other entries in $\mathbf{C}$ the default value of C_{ij} is zero (0.0).

ALPHA_11 $\cdots$ ALPHA_23: The real numbers associated with these keyword represent the values of entries in the vector $\boldsymbol{\alpha}^{(e)}$ of generalized *Biot moduli* (dimensionless) [107].

For three-dimensional idealizations,

$$\boldsymbol{\alpha}^{(e)} = \left\{ \alpha_{11} \quad \alpha_{22} \quad \alpha_{33} \quad \alpha_{12} \quad \alpha_{13} \quad \alpha_{23} \right\}^T$$

For two-dimensional *plane strain* or *plane stress* idealizations,

$$\boldsymbol{\alpha}^{(e)} = \left\{ \alpha_{11} \quad \alpha_{22} \quad \alpha_{12} \right\}^T$$

For *torsionless axisymmetric* idealizations,

$$\boldsymbol{\alpha}^{(e)} = \left\{ \alpha_{11} \quad \alpha_{22} \quad \alpha_{33} \quad \alpha_{12} \right\}^T$$

The default values of α_{11}, α_{22} and α_{33} are 1.0. The default values of α_{12}, α_{13} and α_{23} are 0.0. It follows that the default condition is thus an *isotropic* material with scalar α equal to unity.

M_Modulus: The real number associated with this keyword represents the value of the "Biot modulus" M (units of FL^{-2}) [27] that characterizes the volume compressibility of the soil/fluid mixture. The default value of M is 10^{10}. For mixed-penalty elements, M represents the *penalty number* that is used to enforce the constraint of incompressibility on a saturated soil/fluid mixture.

Example of Command Usage

A specific transversely isotropic material is characterized by the following poroelastic constants: $c_{11} = c_{22} = 7943$ MPa, $c_{12} = 3282$ MPa, $c_{13} = c_{23} = 3367$ MPa, $c_{33} = 5020$ MPa, $c_{44} = 2331$ MPa, $c_{55} = c_{66} = 1500$ MPa, $\alpha_{11} = \alpha_{22} = 0.460$, $\alpha_{33} = 0.565$, and $M = 10.1 \times 10^3$ MPa [22]. All remaining constants are taken equal to zero. The material is assigned the number two (2). The associated material input data consists of the following information (recall that "=" constitutes a valid delimiter in the APES$^+$ Problem Oriented Language).

```
material poroelastic aniso  num  2  &
 desc ‘‘transversely isotropic idealization of the Mandel problem’’ &
  c11 = 7943.0  c12 = 3282.0  c13 = 3367.0  c22 = 7943.0  &
  c23 = 3367.0  c33 = 5020.0  c44 = 2331.0  c55 = 1500.0  &
  c66 = 1500.0  alpha_11 = 0.46 alpha_22 = 0.46 &
  alpha_33 = 0.565   m_mod = 10.1e+03
```

31.7 The MATERIAL ELASTIC HERRMANN command

Synopsis

The **MATERIAL ELASTIC HERRMANN_INCOMPRESSIBLE** command is used to specify the parameters associated with an *isotropic*, linear elastic material idealization valid for incompressible as well as compressible materials. The associated constitutive relation was first proposed in 1965 by L. R. Herrmann [36].

The **MATERIAL ELASTIC HERRMANN_INCOMPRESSIBLE** idealization is intended for use with *mixed* elements for the analysis of incompressible or nearly incompressible solids. Consequently, this material idealization should be used in conjunction with mixed elements possessing *discontinuous* pressure approximations. Presently APES+ contains the following elements of this type:

- **Q4P1d** quadrilateral elements.
- **Q9P3d** quadrilateral elements.
- **T6P1d** triangular elements.
- **T7P3d** triangular elements.
- **H8P1d** hexahedral elements.
- **H20P4d** hexahedral elements.
- **H27P4d** hexahedral elements.

Syntax

The following syntax is associated with the **MATERIAL ELASTIC HERRMANN_INCOMPRES** command:

MATerial ELAstic HERrmann_incompressible NUMber #
(DEScription "string")
(DENsity #.#)
(LAMbda #.#) (SHEar_modulus #.#)

Explanatory Notes

- The **NUMBER** keyword is used to specify the (global) number of the material associated with the incompressible isotropic elastic idealization. The default material number is one (1).

- The optional alphanumeric string associated with the **DESCRIPTION** keyword must be enclosed in double quotes ("). It is used solely to describe the material being idealized to the analyst. The **DESCRIPTION** string is printed as part of the "echo" of the material.

- For mixed (as opposed to mixed/penalty) versions the aforementioned elements, the keyword **LAMBDA** is used to input an appropriate value for the Lamé parameter λ. Denoting the elastic modulus by E and Poisson's ratio by ν,

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

  The default value of **LAMBDA** is 2.50e+05. The inverse of λ is used in the element pressure submatrix; viz.,

$$\mathbf{L}^{(\mathbf{e})} = -\int_{\Omega^e} \left(\frac{1}{\lambda^{(e)}} \mathbf{N}^{(\mathbf{p})^T} \mathbf{N}^{(\mathbf{p})} \right) d\Omega$$

  where $\mathbf{N}^{(\mathbf{p})}$ is a vector of pressure interpolation functions used to approximate the pressure in the element domain Ω^e.

- For the case of $\nu = 0.0$, one cannot input $\lambda \equiv$ **LAMBDA** $= 0.0$, for this would cause a "divide-by-zero" runtime error. In such cases, the analyst should simply input a small number for λ (e.g., 1.0). Past experience shows that the element will then yield results that are identical to those associated with the corresponding mixed elements with the incompressibility constraint *exactly* enforced (but with $\mathbf{L}^{(\mathbf{e})} = \mathbf{0}$ but with a computationally more complex solution).

- As a material approaches the incompressible limit λ approaches infinity. For slightly incompressible materials, experience shows that the *mixed* versions of the aforementioned elements give accurate results.

- If the *mixed/penalty* version of the aforementioned elements is used, the submatrix $\mathbf{L}^{(\mathbf{e})}$ is replaced by $(1/\lambda)\,\mathbf{I}$, where λ now represents a large but finite *penalty number* that approximately enforces the incompressibility constraint and $\mathbf{I}$ is an identity matrix of appropriate size. In this case, **LAMBDA** should be taken equal to a value that is approximately one order of magnitude greater than the actual value of λ associated with the material idealization. As the incompressible limit is approached, the check on the suitability of the value chosen for **LAMBDA** is the magnitude of the element volumetric strain, which should be zero (at least to the significant digits displayed in the result).

- The keyword **SHEAR_MODULUS** is used to specify the elastic shear modulus $G = \mu$, where

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

The *default* **SHEAR_MODULUS** is 1.150e+05.

Example of Command Usage

Assuming that $E = 20\,\text{x}\,10^5$ and $\nu = 0.4999$, and noting that

$$\lambda = \frac{E\nu}{(1+\nu)(1-2\nu)} = \frac{(20\,\text{x}\,10^5)(0.4999)}{(1+0.4999)(1-2(0.4999))} = 3.333 \times 10^9$$

and that

$$G = \frac{E}{2(1+\nu)} = \frac{(20\,\text{x}\,10^5)}{2(1+0.4999)} = 6.667 \times 10^5$$

the following commands are sufficient to specify an incompressible "Herrmann" isotropic elastic material for these parameter values:

```
mat elas Herrmann_incomp num 1  des "nearly incompressible idealization" &
     lambda 3.333e+09  shear  6.667e+05
```

Theoretical Considerations

The isotropic, linear elastic material idealization valid for incompressible as well as compressible materials was first proposed in 1965 by L. R. Herrmann [36]. This formulation is based on the observation that the main difficulty in applying standard displacement-based formulations to incompressible or nearly incompressible media lies in the determination of the mean stress or pressure, which is related to the volumetric part of the strain. For this reason, it is convenient to separate this pressure from the total stress field and to treat it as a primary dependent variable *independent* from the displacement.

Utilyzing a special case of the Hellinger-Reissner mixed variational principle in which displacements and the average hydrostatic pressure were selected as primary dependent variables, Herrmann [36] proposed a mixed formulation consisting of the constitutive relation

$$\sigma_{ij} = -p\delta_{ij} + 2\mu\varepsilon_{ij} \tag{31.16}$$

and the following constraint equation for the additional unknown p:

$$u_{i,i} + \frac{p}{\lambda} = 0 \tag{31.17}$$

In equations (31.16) and (31.17) compressive pressure is taken as being negative.

For the incompressible case, $\nu = 1/2 \Leftrightarrow \lambda = \infty$. Equation (31.17) thus correctly enforces the kinematic constraint of $u_{i,i} = 0$. For compressible materials $\nu < 1/2$ and λ is finite. Solving equation (31.17) gives $p = -\lambda u_{i,i}$. Substituting this relation into equation (31.16), and noting that $\delta_{ij}u_{k,k} = \varepsilon_{11} + \varepsilon_{22} + \varepsilon_{33} = \varepsilon_{kk}$ gives

$$\sigma_{ij} = \lambda\delta_{ij}\varepsilon_{kk} + 2\mu\varepsilon_{ij} \tag{31.18}$$

which is the standard constitutive relation for isotropic linear elasticity. Appendix **??** gives additional details pertaining to the "Herrmann" formulation.

Remark: The pressure p appearing in equations (31.16) and (31.17) does not necessarily equal the mean pressure. From equation (31.18) it follows that

$$\sigma_{ii} = 3\lambda\varepsilon_{kk} + 2\mu\varepsilon_{ii} = (3\lambda + 2\mu)\,\varepsilon_{kk}$$

From the definition of the mean or hydrostatic pressure $\bar{p}$, it follows that

$$\bar{p} = -\frac{\sigma_{ii}}{3} = -\left(\lambda + \frac{2}{3}\mu\right)\varepsilon_{kk} = -K\varepsilon_{kk}$$

Now from equation (31.17) $p = -\lambda u_{i,i} = -\lambda\varepsilon_{kk}$. Since $K \neq \lambda$, it follows that for the *compressible* case, p does *not* represent the hydrostatic pressure. Indeed,

$$\frac{p}{\lambda} = \frac{\bar{p}}{K}$$

For the *nearly incompressible* case $\mu << \lambda$, $K \approx \lambda$, and thus $p \approx \bar{p}$. Finally, for the *incompressible* case, the standard constitutive relations from isotropic elasticity do not apply, thus no meaningful relationship can be made. However, for the mixed method, for $\lambda = \infty$, equation (31.17) correctly gives $u_{i,i} = \varepsilon_{kk} = 0$. Thus, from equation (31.16)

$$\sigma_{kk} = -3p + 2\mu\varepsilon_{kk} = -3p$$

showing that now $p = -\sigma_{kk}/3 = \bar{p}$.

31.8 The MATERIAL FLEXURAL ELASTIC command

Synopsis

The **MATERIAL FLEXURAL ELASTIC** command is used to specify the parameters associated with an isotropic linear elastic material idealization for flexural (i.e., beam-column type) elements.

Syntax

The following syntax is associated with the **MATERIAL FLEXURAL ELASTIC** command:

MATerial FLExural ELAstic NUMber #
(DEScription "string")
(MODulus #.#) (POIssons_ratio #.#)
(WINKLER_Rotational #.#) (WINKLER_Translational #.#)

Explanatory Notes

- The **NUMBER** keyword is used to specify the (global) number of the material associated with the incompressible isotropic elastic idealization. The default material number is one (1).
- The optional alphanumeric string associated with the **DESCRIPTION** keyword must be enclosed in double quotes ("). It is used solely to describe the material being idealized to the analyst. The **DESCRIPTION** string is printed as part of the "echo" of the material.
- The **MODULUS** keyword corresponds to the elastic (Young's) modulus; the default elastic **MODULUS** is 3.00e+05.
- The keyword **POISSONS_RATIO** corresponds to Poisson's ratio; the default **POISSONS_RATIO** is 0.0.
- The **WINKLER_ROTATIONAL** keyword corresponds to a *rotational* Winkler spring stiffness that will be associated with the element; the default **WINKLER_ROTATIONAL** is 0.0.

- The **WINKLER_TRANSLATIONAL** keyword corresponds to a translational Winkler spring stiffness that will be associated with the element; the default **WINKLER_TRANSLATIONAL** is 0.0.

Example of Command Usage

The following command is used to specify a typical isotropic elastic material for flexural elements

```
mat flex elastic  number 1 &
     desc ‘‘exotic material for beam elements’’ &
        modulus  20.0e+05  winkler_trans  1.0e+04
```

Theoretical Considerations

The model of Winkler [113] assumes a continuous elastic foundation consisting of closely spaced, independent linear springs. Further details pertaining to basic elastic and viscoelastic foundation models are given by Kerr [63]. The analysis of beams resting on Winkler foundations using the finite difference method is discussed in Section 3.4 of [52]. Cook et al. [20] discuss the finite element solution to this problem.

Chapter 32

Quasi-Linear and Variable Modulus Elastic MATERIAL Idealizations

This chapter contains information associated with the description of quasi-linear and variable modulus elastic material idealizations for finite element analyses using the APES$^+$ computer program. The majority of these idealizations have been developed to simulate the behavior of geomaterials.

32.1 General Remarks

Both quasi-linear and variable modulus elastic material idealizations are classified as hypoelastic[1] formulations. Hypoelastic models is characterized by an incremental or *hypoelastic* formulation.[2] In such a formulation the one-to-one relation between stress and strain is only defined *incrementally*. Hypoelasticity is used to model materials that exhibit nonlinear, but reversible, stress strain behavior even at small strains. Its most common application is in the so-called "deformation theory of plasticity," which is a crude approximation of the behavior loaded beyond the elastic limit.

Analytically hypoelastic materials are described by

$$\dot{\sigma}_{ij} = D_{ijkl}\dot{\varepsilon}_{kl} \tag{32.1}$$

where the fourth-order tensor C_{ijkl} can be a linear or nonlinear function of either stress or strain. The formulation is thus path-dependent, resulting in the possibility of energy generation or dissipation (in the form of residual strains or stresses) in closed cycles. This is seen by integrating equation (32.1), giving

$$\sigma_{ij} = \int_0^t D_{ijkl}\frac{\partial \varepsilon_{kl}}{\partial \tau}\, d\tau + \sigma_{ij}^0$$

Perhaps the major inconsistency of hypoelastic models is, however, the fact that under multiaxial stress conditions such models cannot consistently distinguish between loading and

[1] Nonlinear elastic constitutive models typically fall into one of three general classes: a) Cauchy elastic, b) Hyperelastic, and c) Hypoelastic.

[2] Truesdell [111] gave the name "hypoelastic" to this incremental theory.

unloading. Additional advantages and limitations of hypoelastic formulations are summarized below. The seminal papers by Green [35] and Truesdell [111] give additional details pertaining to hypoelastic material idealizations.

Remark: Similar to Cauchy elastic formulations, hypoelastic ones are conceptually and mathematically simple and are generally easy to implement numerically. ◀

Remark: The values for parameters associated with hypoelastic formulations are relatively easy to determine. Over time, a relatively extensive database of such parameter values has been established. ◀

Remark: However, the application of this type of hypoelastic models should be confined to loading situations that do not substantially differ from the experimental tests that were used to determine the model parameter values. Thus, the isotropic models should not be used in cases such as non-proportional loading or cyclic loading paths. ◀

Remark: Similar to Cauchy elastic formulations, hypoelastic ones cannot account for shear-dilatancy; i.e., volume changes resulting from shearing. ◀

32.2 Overview of Quasilinear and Variable Modulus Elastic Models for Geomaterials

A quasilinear model approximates actual nonlinear material behavior by a series of linear segments. The majority of such models employ a *piecewise linear* approximation to the actual non-linear response. This is shown schematically in Figures 32.1 and 32.2.

Often the piecewise linear increments are generated assuming *isotropic* elastic response.[3] Consequently, the material is characterized by the instantaneous value of *two* material constants. It follows that for each iteration of each solution increment, the elastic modulus E and Poisson's ratio ν, or the elastic bulk modulus K and elastic shear modulus G or the Lamé parameters λ and μ are revised by making them functions of stress or strain. The general incremental response thus has the following form:

$$ {}^{(m;i)}d\boldsymbol{\sigma} = {}^{(m;i)}\mathbf{C}\ {}^{(m;i)}d\boldsymbol{\varepsilon} \tag{32.2} $$

[3]Implicit in this approach is the assumption that the soil does not experience stress or strain induced anisotropy.

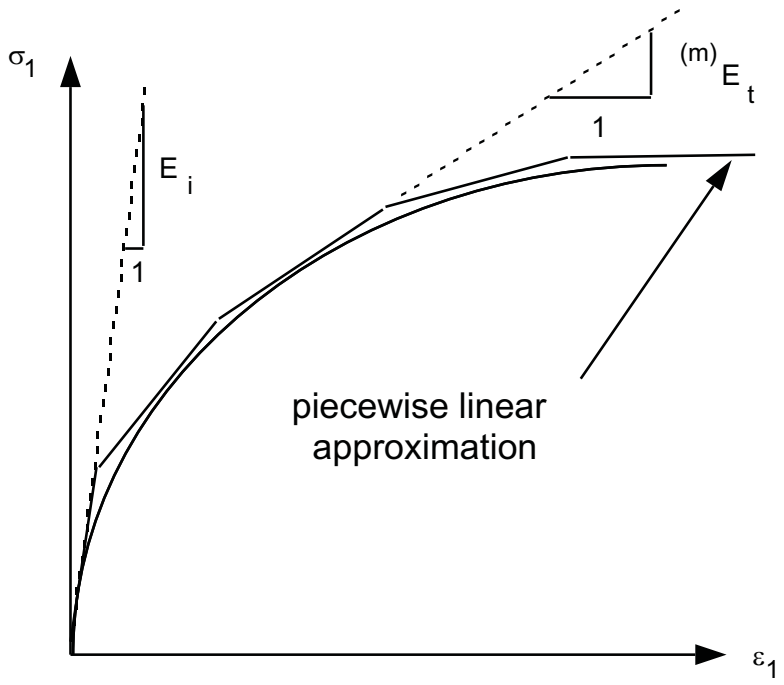

Figure 32.1: Piecewise Linear Approximation of Uniaxial Stress-Strain Curve

where m is the increment number, i is the iteration number, ${}^{(m)}d\boldsymbol{\sigma}$ and ${}^{(m)}d\boldsymbol{\varepsilon}$ are the incremental stress and strain vectors, respectively, and ${}^{(m)}\mathbf{C}$ is the (tangent) constitutive matrix associated with the ith iteration of the mth increment. In light of the incremental form given by equation (32.2), quasilinear elastic models are classified as being *hypoelastic*.

Due to the assumption of isotropic elasticity it follows that

$$ {}^{(m;i)}\mathbf{C} = \begin{bmatrix} {}^{(m)}\mathbf{C_{11}} & \mathbf{0} \\ \mathbf{0} & {}^{(m)}\mathbf{C_{22}} \end{bmatrix} \tag{32.3} $$

where

$$ {}^{(m;i)}\mathbf{C_{11}} = \frac{E_t}{(1+\nu_t)(1-2\nu_t)} \begin{bmatrix} 1-\nu_t & -\nu_t & -\nu_t \\ -\nu_t & 1-\nu_t & -\nu_t \\ -\nu_t & -\nu_t & 1-\nu_t \end{bmatrix} \tag{32.4} $$

$$ {}^{(m;i)}\mathbf{C_{22}} = \frac{E_t}{2(1+\nu_t)} \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix} \tag{32.5} $$

or

$$ {}^{(m;i)}\mathbf{C_{11}} = \begin{bmatrix} K_t + 4G_t/3 & K_t - 2G_t/3 & K_t - 2G_t/3 \\ K_t - 2G_t/3 & K_t + 4G_t/3 & K_t - 2G_t/3 \\ K_t - 2G_t/3 & K_t - 2G_t/3 & K_t + 4G_t/3 \end{bmatrix} \tag{32.6} $$

$$ {}^{(m;i)}\mathbf{C_{22}} = \begin{bmatrix} G_t & 0 & 0 \\ 0 & G_t & 0 \\ 0 & 0 & G_t \end{bmatrix} \tag{32.7} $$

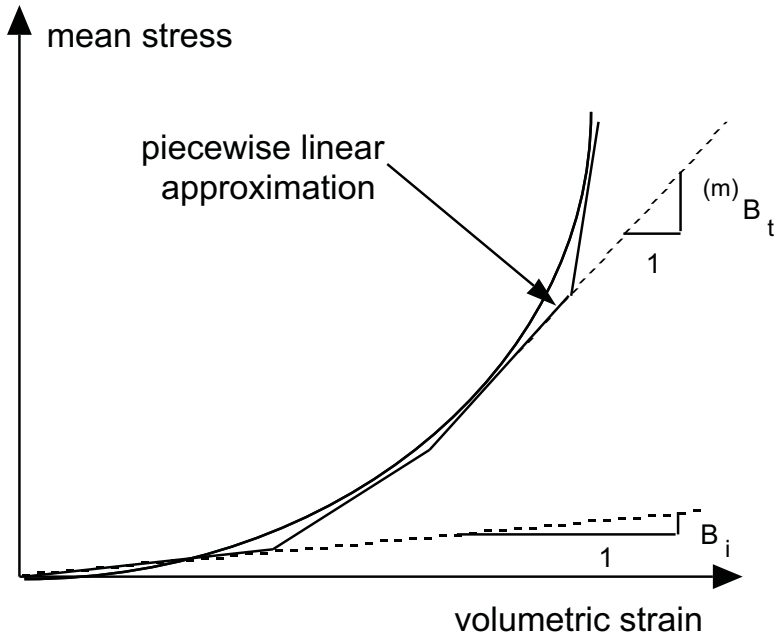

Figure 32.2: Piecewise Linear Approximation of Mean Stress versus Volumetric Strain Curve

In the above equations, E_t, ν_t, K_t and G_t represent tangent values of the elastic modulus, Poisson's ratio, the elastic bulk and elastic shear modulus, respectively.

For its success, a piecewise linear representation of non-linear behavior relies upon the determination of variations in these material parameters during the incremental loading. For this reason, quasilinear models are sometimes referred to as *variable modulus* or *variable parameter* models [26].

32.3 Advantages and Limitations of Quasilinear and Variable Modulus Elastic Models for Geomaterials

The advantages of quasilinear elastic models stem from their simplicity and their successful use in analyzing a number of different practical problems. More precisely, the advantages of such models are:

- The values of the required parameters can be readily determined from the results of a series of conventional *axisymmetric* triaxial compression tests. Use of this model thus does not require more extensive or more exotic testing programs than those which are routinely performed for other soil engineering purposes. This is an important consideration, because the cost of a few unusual or unconventional laboratory tests may easily exceed the cost of the finite element analysis for which they are performed.

- The parameters associated with the model have readily understood physical significance. This is helpful because engineering judgment is enhanced when each element of the model is comprehensible in physical terms, and the effects of changes in the parameter values can be readily anticipated [28].

- Parameter values have been determined for many soils, under both drained and undrained conditions. These data are useful for estimating parameter values when only soil type and density are known, and for judging the correctness of laboratory test results by means of comparisons with results for similar soils. Some parameter values reported in the literature are given in Section 32.5.1.

- The capabilities and limitations of the formulation are rather thoroughly documented and well-understood.

- Quasilinear models are relatively easy to implement into computer programs. This implementation typically involves an incremental/iterative solution scheme.

Since models based on a hyperbolic relation represent a highly idealized characterization of the soil, they possess some rather significant limitations. Since these limitations should be understood by anyone using the model, they are summarized below.

- In the model no account is made for the value of the intermediate principal stress σ_2. Thus the accuracy of predictions under other than axisymmetric triaxial conditions (e.g., plane strain, true triaxial, etc.) may be questionable.

- Since a Mohr-Coulomb failure criterion is assumed, the failure envelope will be straight. For many soils this is known not to be the case, however. It is pertinent to note that Andrawes et al. [4] attempted to include the effect of the curvature of the Mohr failure envelope for cohesionless soils. They proposed the following relation:

$$\phi = \phi_0 + \Delta\phi \log\left(\frac{\sigma_3}{P_a}\right)$$

  where ϕ_0 is the value of the effective friction angle for σ_3 equal to P_a, and $\Delta\phi$ is a material constant equal to the reduction in ϕ for a ten-fold increase in σ_3.

- Being based on the generalized Hooke's Law the relationships are most suitable for analysis of stresses and movements prior to failure. The relationships are capable of predicting accurately non-linear relationships between loads and movements, and it is possible to continue the analyses up to the stage where some elements experience local failure. However, when a stage is reached where the behavior of the soil mass is controlled to a large extent by the properties assigned to elements which have already failed, the results will no longer be reliable, and they may be unrealistic in terms of the behavior of real soils at and after failure. These relationships are not useful, therefore, for analyses extending up to the stage of instability of a soil mass. They are useful for predicting movements in stable earth masses.

- Special considerations must typically be included in order to properly model unloading.

- The hyperbolic relationships do not include volume change due to changes in shear stress, or "shear dilatancy." They may, therefore, be limited in the accuracy with which they can be used to predict deformations in dilatant soils such as dense sands under low confining pressures. To address this shortcoming, Bathurst and Karpurapu [6] extended the model of Duncan [28] by incorporating a dilation angle to simulate the dilation of soils.

- For some combinations of parameter values and stresses, the values of tangent Poisson's ratio ν_t calculated may exceed 0.5. However, since thermodynamic considerations limit Poisson's ratio to the range $-1.0 \leq \nu \leq 0.50$, in most computer programs that employ these parameters, ν_t is set equal to 0.49 under such circumstances.

- The parameters are *not* fundamental soil properties, but only values of empirical coefficients which represent the behavior of the soil under a limited range of conditions. The values of the parameters depend on the density of the soil, its water content, the range of pressures used in testing, and the drainage conditions. In order that the parameters will be representative of the behavior of the soil under field conditions, the laboratory test conditions must correspond to the field conditions with regard to these factors.

- As noted by Zytynski et al. [119], the variation of E_i using equation (32.19) violates the principle of conservation of energy. Thus, depending on the direction of a closed stress loop, the model will generate or dissipate energy in violation of the basic premise of elastic behavior.

- In discussing "hyperbolic" quasi-linear constitutive models, Desai and Siriwardane [26] note that:

> " They can give satisfactory results for only a limited class of problems $\cdots$. For instance, if we are interested in stress-deformation analysis of a nonlinear semi-infinite medium under a monotonically increasing point or strip load, the results could be acceptable. However, for problems involving loading and unloading, and various stress paths in the soil, results from the hyperbolic simulation may not be reliable, one of the major limitations being that the model includes only one stress path, whereas loading and/or unloading could cause a wide range of stress paths $\cdots$"

Additional details pertaining to the hyperbolic model can be found in the aforementioned references by Duncan and his co-workers, as well as in the report by Wong and Duncan [114]. The use of the model to simulate various boundary-value problems is described in sundry papers [16] and technical reports [18; 74; 72]. Khabbazian et al. [65] discuss additional shortcomings associated with quasilinear elastic constitutive models, as applied to the simulation of geosynthetic encased columns.

32.4 Available Quasi-Linear and Variable Modulus Elastic Material Idealizations

The following quasi-linear and variable modulus elastic material idealizations are currently available in the APES+ computer program:

- Quasi-linear **HYPERBOLIC_ELASTIC** material idealizations for axisymmetric triaxial conditions (see Section 32.5).
- **VARIABLE_MODULUS JANBU** nonlinear hypoelastic material idealization (see Section 32.6).
- **VARIABLE_MODULUS LADE_NELSON** nonlinear hypoelastic material idealization (see Section 32.7).
- **VARIABLE_MODULUS QUBAIN** nonlinear hypoelastic material idealization (see Section 32.8).

32.5 The MATERIAL HYPERBOLIC_ELASTIC command

Synopsis

The **MATERIAL HYPERBOLIC_ELASTIC** command is used to specify parameters associated with various forms of the quasilinear (hyperbolic) model, originally proposed by Duncan and Chang in 1970 [29]. The hyperbolic material idealization, though rather limited in scope [26] and possessing known shortcomings [65], has none the less gained popularity in the field of geotechnical engineering. Consequently, *four* forms of this material idealization are available in APES+. Details pertaining to theoretical aspects of quasilinear (hyperbolic) models are given in Section (32.5.2).

Syntax

The following syntax is used to describe the various forms of the **MATERIAL HYPERBOLIC_ELASTIC** idealization:

MATerial HYPerbolic_elastic NUMber #
(DEScription "string")
KINd DUNCAN_CHANG
(MODULUS_NUMBER_Loading #.#) (MODULUS_NUMBER_Unloading #.#)
(MODULUS_Exponent #.#)
(FRIction_angle #.#) (COHesion #.#) (FAilure_ratio #.#)
(ATMospheric_pressure #.#) (POIssons_ratio #.#)
(RELaxation_parameter #.#)

MATerial HYPerbolic_elastic NUMber #
(DEScription "string")
KINd KULHAWY_DUNCAN
(MODULUS_NUMBER_Loading #.#) (MODULUS_NUMBER_Unloading #.#)
(MODULUS_Exponent #.#)
(FRIction_angle #.#) (COHesion #.#) (FAilure_ratio #.#)
(ATMospheric_pressure #.#)
(POISSONS_NUMBER_G #.#) (POISSONS_NUMBER_F #.#)
(POISSONS_NUMBER_D #.#)
(RELaxation_parameter #.#)

MATerial HYPerbolic_elastic NUMber #
(DEScription "string")
KINd HERRMANN
(MODULUS_NUMBER_Loading #.#) (MODULUS_NUMBER_Unloading #.#)
(MODULUS_Exponent #.#)
(FRIction_angle #.#) (COHesion #.#) (FAilure_ratio #.#)
(ATMospheric_pressure #.#) (BULk_modulus #.#)
(RELaxation_parameter #.#)

MATerial HYPerbolic_elastic NUMber #
(DEScription "string")
KINd DUNCAN_1980)
(MODULUS_NUMBER_Loading #.#) (MODULUS_NUMBER_Unloading #.#)
(MODULUS_Exponent #.#)
(FRIction_angle #.#) (COHesion #.#) (FAilure_ratio #.#)
(ATMospheric_pressure #.#)
(BULK_MODULUS_Number #.#) (BULK_MODULUS_Exponent #.#)
(RELaxation_parameter #.#)

Explanatory Notes

A quasilinear elastic hyperbolic material idealization assumes that the soil does not experience stress or strain induced anisotropy; i.e., the soil may at all times be characterized by instantaneous values of the elastic (Young's) modulus E and Poisson's ratio ν. The keywords associated with the various forms of the quasilinear elastic hyperbolic material idealization are described below.

The **DUNCAN_CHANG** keyword is used to specify parameters associated with the original form of the hyperbolic model proposed by Duncan and Chang [29]. The following keywords are associated with this form of the model:

- The **MODULUS_NUMBER_LOADING** keyword is associated with the modulus number K under conditions of loading. The default value is equal to 1.0.
- The **MODULUS_NUMBER_UNLOADING** keyword is associated with the modulus number K_{ur} under conditions of unloading. The default value is equal to 1.0.
- **MODULUS_EXPONENT** refers to the exponent n associated with the definition of the instantaneous value of the elastic modulus (E_i). The default value is equal to 1.0.

- The **FRICTION_ANGLE** keyword is used to specify the angle of internal friction; this angle must be given in *degrees*. The default value is 30.0 degrees.
- The **COHESION** keyword is used to specify the value of the cohesion intercept. The default intercept is 0.0.
- The **FAILURE_RATIO** keyword is used to specify the value of the failure ratio R_f; the default value is 0.8.
- The **ATMOSPHERIC_PRESSURE** keyword is used to specify the magnitude of the atmospheric pressure, which is used to normalize the confining stress σ_3. The default **ATMOSPHERIC_PRESSURE** is equal to 1.0.
- The **POISSONS_RATIO** keyword is used to specify the value of Poisson's ratio (ν). The default **POISSONS_RATIO** is equal to 0.20.
- The **RELAXATION_PARAMETER** represents an attempt at accelerating the performance of the constitutive model [39]; the default value is 0.80.

The **KULHAWY_DUNCAN** keyword is used to specify parameters associated with the hyperbolic model proposed by Kulhawy and Duncan [73]. This form of the model attempted to improve upon the predictive capabilities of the original formulation, the latter being specified via the **DUNCAN_CHANG** keyword. The following keywords are associated with the **KULHAWY_DUNCAN** form of the model:

- The **MODULUS_NUMBER_LOADING** keyword is associated with the modulus number K under conditions of loading. The default value is equal to 1.0.
- The **MODULUS_NUMBER_UNLOADING** keyword is associated with the modulus number K_{ur} under conditions of unloading. The default value is equal to 1.0.
- **MODULUS_EXPONENT** refers to the exponent n associated with the definition of the instantaneous value of the elastic modulus (E_i). The default value is equal to 1.0.
- The **FRICTION_ANGLE** keyword is used to specify the angle of internal friction; this angle must be given in *degrees*. The default value is 30.0 degrees.
- The **COHESION** keyword is used to specify the value of the cohesion intercept. The default intercept is 0.0.
- The **FAILURE_RATIO** keyword is used to specify the value of the failure ratio R_f; the default value is 0.8.
- The **ATMOSPHERIC_PRESSURE** keyword is used to specify the magnitude of the atmospheric pressure, which is used to normalize the confining stress σ_3. The default **ATMOSPHERIC_PRESSURE** is equal to 1.0.

- The **POISSONS_NUMBER_G** keyword is used to specify the value of the Kulhawy and Duncan [73] model parameter G. This parameter represents the value of the initial Poisson's ratio at a pressure of one atmosphere. The default **POISSONS_NUMBER_G** is equal to 1.0.

- The **POISSONS_NUMBER_F** keyword is used to specify the value of the Kulhawy and Duncan [73] model parameter F. This parameter represents the rate of change of Poisson's ratio with σ_3. The default **POISSONS_NUMBER_F** is equal to 1.0.

- The **POISSONS_NUMBER_D** keyword is used to specify the value of the Kulhawy and Duncan [73] model parameter D. This parameter expresses the rate of change of the tangential Poisson's ratio (ν_t) with strain. The default **POISSONS_NUMBER_D** is equal to 1.0.

- The **RELAXATION_PARAMETER** represents an attempt at accelerating the performance of the constitutive model [39]; the default value is 0.80.

The **HERRMANN** keyword is used to specify parameters associated with a slightly modified form of the hyperbolic model as proposed by Herrmann [38]. The following keywords are associated with this form of the model:

- The **MODULUS_NUMBER_LOADING** keyword is associated with the modulus number K under conditions of loading. The default value is equal to 1.0.

- The **MODULUS_NUMBER_UNLOADING** keyword is associated with the modulus number K_{ur} under conditions of unloading. The default value is equal to 1.0.

- **MODULUS_EXPONENT** refers to the exponent n associated with the definition of the instantaneous value of the elastic modulus (E_i). The default value is equal to 1.0.

- The **FRICTION_ANGLE** keyword is used to specify the angle of internal friction; this angle must be given in *degrees*. The default value is 30.0 degrees.

- The **COHESION** keyword is used to specify the value of the cohesion intercept. The default intercept is 0.0.

- The **FAILURE_RATIO** keyword is used to specify the value of the failure ratio R_f; the default value is 0.8.

- The **ATMOSPHERIC_PRESSURE** keyword is used to specify the magnitude of the atmospheric pressure, which is used to normalize the confining stress σ_3. The default **ATMOSPHERIC_PRESSURE** is equal to 1.0.

- The **BULK_MODULUS** keyword is used to specify the value of the elastic bulk modulus (B). The default **BULK_MODULUS** is equal to 1.0.

- The **RELAXATION_PARAMETER** represents an attempt at accelerating the performance of the constitutive model [39]; the default value is 0.80.

The keyword **DUNCAN_1980** signifies another modified form of the hyperbolic model as proposed by Duncan [28]. The following keywords are associated with this form of the model:

- The **MODULUS_NUMBER_LOADING** keyword is associated with the modulus number K under conditions of loading. The default value is equal to 1.0.
- The **MODULUS_NUMBER_UNLOADING** keyword is associated with the modulus number K_{ur} under conditions of unloading. The default value is equal to 1.0.
- **MODULUS_EXPONENT** refers to the exponent n associated with the definition of the instantaneous value of the elastic modulus (E_i). The default value is equal to 1.0.
- The **FRICTION_ANGLE** keyword is used to specify the angle of internal friction; this angle must be given in *degrees*. The default value is 30.0 degrees.
- The **COHESION** keyword is used to specify the value of the cohesion intercept. The default intercept is 0.0.
- The **FAILURE_RATIO** keyword is used to specify the value of the failure ratio R_f; the default value is 0.8.
- The **ATMOSPHERIC_PRESSURE** keyword is used to specify the magnitude of the atmospheric pressure, which is used to normalize the confining stress σ_3. The default **ATMOSPHERIC_PRESSURE** is equal to 1.0.
- The **BULK_MODULUS_NUMBER** keyword is used to specify the value of the "bulk modulus number" (K_b) introduced by Duncan [28]. The default **BULK_MODULUS_NUMB** is equal to 1.0.
- The **BULK_MODULUS_EXPONENT** keyword is used to specify the value of the "bulk modulus exponent" (n) introduced by Duncan [28]. The default **BULK_MODULUS_EXPO** is equal to 1.0.
- The **RELAXATION_PARAMETER** represents an attempt at accelerating the performance of the constitutive model [39]; the default value is 0.80.

Example of Command Usage

• Duncan-Chang Form of the Model

A Duncan and Chang [29] quasilinear elastic hyperbolic material idealization is to be used with the following model parameter values: $K = 200$, $K_{ur} = 350$, $n = 0.43$, $R_f = 0.85$, $\phi = 17°$, $c = 33.0$ kPa and $\nu = 0.40$. To describe such a material idealization for the APES+ computer program, enter the following command:

```
material hyperbolic_elastic number 2  &
description "idealization of sand fill"   kind  "duncan_chang"  &
modulus_number_load 200.0 modulus_number_unload  350.0  &
modulus_exponent  0.43 failure_ratio 0.85  friction  17.0  &
cohesion  33.0 poissons_ratio 0.40
```

• Kulhawy-Duncan Form of the Model

A Kulhawy and Duncan [73] quasilinear elastic hyperbolic material idealization is to be used with: $K = 1580$, $K_{ur} = 1900$, $n = 1.05$, $R_f = 0.90$, $\phi = 42.7°$, $c = 0.0$ kPa and $G = 0.46$, $F = 0.17$ and $D = 24.0$. To describe such a material idealization, enter the following command:

```
material hyperbolic_elastic duncan_chang number 2  &
description "idealization of sand fill"  &
kind  "kulhawy_duncan"   &
modulus_number_load 1580.0  modulus_number_unload  1900.0  &
modulus_exponent  1.05  failure_ratio 0.90  friction  42.7  &
poissons_number_G  0.46   poissons_number_F  0.17   &
poissons_number_D  24.0
```

• Duncan-Chang Material Idealization of Dense Sacramento River Sand

The following commands are used to specify the material parameters associated with the Duncan-Chang [29] version of the quasilinear elastic (hyperbolic) material idealization as applied to dense Sacramento River sand:

```
material hyperbolic_elastic num  1 &
desc "parameters for DENSE Sacramento River sand"  &
kind DUNCAN_CHANG & modulus_number_load  1600.0  &
modulus_number_unload 1600.0 & modulus_exp 0.45 &
friction_angle 41.0  cohesion 0.0  &
```

```
failure_ratio 0.95 &
atmospheric_pre 101.4 &
poissons_ratio 0.20
```

• Kulhawy-Duncan Material Idealization of Dense Sacramento River Sand

The following commands are used to specify the material parameters associated with the Kulhawy-Duncan [73] version of the quasilinear elastic (hyperbolic) material idealization as applied to dense Sacramento River sand:

```
material hyperbolic_elastic num 2 &
desc "parameters for DENSE Sacramento River sand"  &
kind Kulhawy_DUNCAN &
modulus_number_load 1600.0  &
modulus_number_unload 1600.0 &
modulus_exp 0.45 &
friction_angle 41.0  cohesion 0.0  &
failure_ratio 0.95 &
atmospheric_pre 101.4 &
poissons_number_G 0.450  poissons_number_F -0.10  poissons_number_D 0.20
```

• Duncan Material Idealization of Dense Sacramento River Sand

The following commands are used to specify the material parameters associated with the Duncan [28] version of the quasilinear elastic (hyperbolic) material idealization as applied to dense Sacramento River sand:

```
material hyperbolic_elastic num 3 desc "parameters for DENSE Sacramento River sand"  &
kind DUNCAN_1980 &
modulus_number_load 1600.0 & modulus_number_unload 1600.0 &
modulus_exp 0.45 &
friction_angle 41.0 &
failure_ratio 0.95 &
bulk_modulus_number 500.0 &
bulk_modulus_exp 1.58 & atmospheric_pre 101.4
```

32.5.1 Model Parameter Values Used in Past Analyses

Tables 32.1 to 32.3 list some values of model parameter appearing in the literature. Further parameter sets are available in reports by Wong and Duncan [114] and Duncan et al. [30], as well as in sundry papers [12].

Table 32.1: Parameter Values Associated with the Basic Model of Duncan and Chang [29]

Soil Type	K K_{ur}	n	R_f	ϕ c	ν	Ref.
Ottawa silica sand	1200.0 –	0.66	0.88	38.0° 0.0	0.0	[31]
Ottawa silica sand	1355.2 –	0.61	0.91	38.4° 0.0	0.30	[82]
Sand fill	580.0 –	0.50	0.85	36° 0.0	0.30	[3]
Sand fill fill	177.0 354.0	0.43	0.85	17° 33.0 kPa	0.40	[15]
Sand fill	1000.0 –	1.00	0.90	38° 0.0	0.33	[92]
Sandy clay	290.7 –	0.41	0.96	14.2° 7.45 psi	0.30	[82]
Santa Monica Beach sand (dense)	2100.0 –	0.45	0.95	45.4° 0.0	0.15	[65]
Santa Monica Beach sand (loose)	1090.0 –	0.50	0.98	35.6° 0.0	0.26	[65]
Sacramento River sand (dense)	1600.0 –	0.45	0.95	41.0° 0.0	0.15	[66]
Sacramento River sand (loose)	400.0 –	0.50	0.98	35.0° 0.0	0.20	[66]

Table 32.2: Parameter Values Associated with the Model of [73]

Soil Type	K K_{ur}	n	R_f	ϕ c	G	F	d	Ref.
Leighton Buzzard sand	1573.0 1890.0	1.05	0.90	42.7° 0.0	0.46	0.17	24.0	[4]
silty sand /clay mix	144 100	0.06	0.87	26.0° 2.0 psi	0.20	0.10	4.8	[25]
Ottawa silica sand	908.0 –	0.59	0.86	38.0° 0.0	0.33	0.01	12.0	[116]
Sandy clay	144.0 –	0.06	0.83	26.0° 2.0 psi	0.4	0.1	4.8	[116]
Santa Monica Beach sand (dense)	2100.0 –	0.45	0.95	45.4° 0.0	0.45	-0.10	0.20	[65]
Santa Monica Beach sand (loose)	1090.0 –	0.50	0.98	35.6° 0.0	0.45	-0.10	0.05	[65]
Sacramento River sand (dense)	1600.0 –	0.45	0.95	41.0° 0.0	0.45	-0.10	0.20	[66]
Sacramento River sand (loose)	400.0 –	0.50	0.98	35.0° 0.0	0.45	-0.10	0.05	[66]

Table 32.3: Parameter Values Associated with the Modified Model of Duncan [28]

Soil Type	K K_{ur}	n	R_f	ϕ c	K_b	m	Ref.
Clay "X"	55.0 –	0.0	0.95	30° 0.0	45.0	0.0	[28]
Clay "Y"	165.0 –	0.0	0.60	0° 8.0 psi	80.0	0.0	[28]
Clean Ottawa sand	600.0 600.0	0.40	0.80	35° 1.0 kPa	300.0	0.40	[86]
Kaolinite	800.0 –	0.0	0.72	0° 28.5 psi	10000.0	0.0	[28]
Leighton Buzzard sand	950.0 –	0.80	0.70	49.6° 0.0	250.0	0.60	[61]
Ottawa sand sand	400.0 2000.0	0.85	0.86	43° 0.50 psi	700.0	0.50	[28]
Ottawa silica sand	1116.0 1500.0	0.66	0.87	38.4° 0.50 psi	907.0	0.0	[17]
Ottawa silica sand	1000.0 1500.0	0.67	0.85	38.0° 0.0	700.0	0.40	[46]
Ottawa silica sand	950.0 –	0.36	0.75	38.0° 0.0	485.0	0.64	[60]
RMC sand	950.0 –	0.50	0.75	53.0° 0.0	250.0	0.65	[62]
Sand	913.0 1485.0	0.60	0.64	46.1° 0.0	250.0	0.80	[118]
Sandy clay	370.0 500.0	0.28	0.98	15.0° 7.47 psi	210.0	0.0	[17]
Sandy clay	200.0 –	0.50	0.92	26.3° 2.0 psi	600.0	0.48	[60]
Santa Monica Beach sand (dense)	2100.0 –	0.45	0.95	45.4° 0.0	500.0	1.58	[65]
Santa Monica Beach sand (loose)	1090.0 –	0.50	0.98	35.6° 0.0	500.0	1.58	[65]
Sacramento River sand (dense)	1600.0 –	0.45	0.95	41.0° 0.0	500.0	1.58	[66]
Sacramento River sand (loose)	400.0 –	0.50	0.98	35.0° 0.0	500.0	1.58	[66]

Basic Model of Duncan and Chang

Some values of the parameters associated with the original form of the quasilinear elastic "hyperbolic" model of Duncan and Chang [29] are listed below.

♣ **Soil**: Ottawa Sand

Description

- Subrounded Ottawa silica sand ; $G_s = 2.65$, Dry density (air dried) ; $\gamma_d = 107$ lb/ft^3

Parameter Values

- $K = 1200.0$; $K_{ur} =$ n/a ; $n = 0.66$; $R_f = 0.88$; $\phi = 38.0°$; $c = 0.0$; $\nu = 0.0$

Reference
Ebeling, R. M., Peters, J. F., and Wahl, R. E., "Prediction of Reinforced Sand Wall Performance," *Geosynthetically-Reinforced Soil Retaining Walls*, J. T. H. Wu, editor, Balkema, Rotterdam: 243-253 (1992).

♣ **Soil**: Ottawa Sand

Description

- Subrounded Ottawa silica sand ; $G_s = 2.65$, Dry density (air dried) ; $\gamma_d = 107$ lb/ft^3

Parameter Values

- $K = 1355.2$; $K_{ur} =$ n/a ; $n = 0.61$; $R_f = 0.91$; $\phi = 38.4°$; $c = 0.0$; $\nu = 0.30$

Reference
Ling, H. I. and Tatsuoka, F., "Nonlinear Analysis of Reinforced Soil Structures by Modified CANDE (M-CANDE)," *Geosynthetically-Reinforced Soil Retaining Walls*, J. T. H. Wu, editor, Balkema, Rotterdam: 279–291 (1992).

♣ **Soil**: Sand

Description

- $\gamma_d = 17.7$ kN/m^3

Parameter Values

- $K = 1000.0$; K_{ur} = n/a ; $n = 1.00$; $R_f = 0.90$; $\phi = 38.0°$; $c = 0.0$; $\nu = 0.33$

Reference

Ogisako, E., Ochiai, H., Hayashi, S., and Sakai, A., "FEM Analysis of Polymer Grid Reinforced-Soil Retaining Walls and its Application to the Design Method," *Proceedings of the International Geotechnical Symposium on Theory and Practice of Earth Reinforcement*, Fukuoka Japan, Balkema Publishers: 559-564 (1988).

♣ **Soil**: Sand Fill

Parameter Values

- $K = 580.0$; K_{ur} = n/a ; $n = 0.50$; $R_f = 0.85$; $\phi = 36.0°$; $c = 0.0$; $\nu = 0.30$

Reference

Al-Hussaini, M. M. and Johnson, L. D., "Numerical Analysis of a Reinforced Earth Wall," *Proceedings of the ASCE Symposium on Reinforced Earth*, Pittsburgh, PA: 98-126 (1978).

♣ **Soil**: Sand Fill

Parameter Values

- $K = 177.0$; $K_{ur} = 354.0$; $n = 0.43$; $R_f = 0.85$; $\phi = 17.0°$; $c = 33.0$ kPa ; $\nu = 0.40$

Reference

Chalaturnyk, R. J., Scott, J. D. , and Chan, D. H. K., "Stresses and Deformations in a Reinforced Soil Slope," *Canadian Geotechnical Journal*, **27**(2): 224–232 (1990).

♣ **Soil**: Sandy Clay

Description

- Dark grey, slightly silty, fine to coarse sand and clay mixture ; $G_s = 2.66$; Dry density (air dried): $\gamma_d = 100.7$ lb/ft^3
- water content = 19.3% ; Liquid limit = 37% ; Plastic limit = 19%

Parameter Values

- $K = 290.7$; K_{ur} = n/a ; $n = 0.41$; $R_f = 0.96$; $\phi = 14.2°$; $c = 7.45$ psi ; $\nu = 0.30$.

Reference
Ling, H. I. and Tatsuoka, F., "Nonlinear Analysis of Reinforced Soil Structures by Modified CANDE (M-CANDE)," *Geosynthetically-Reinforced Soil Retaining Walls*, J. T. H. Wu, editor, Balkema, Rotterdam: 279-291 (1992).

♣ **Soil**: Sandy Clay

Description

- Density: $\gamma = 124.6$ lb/ft^3 ; Liquid limit = 25%, Plastic limit = 12%
- $D_{30} = 0.002$ mm ; $D_{60} = 0.040$ mm

Parameter Values

- $K = 280.0$; $K_{ur} = 840$; $n = 0.6$; $R_f = 0.93$; $\phi = 18.0°$; $c = 12.64$ psi
- $\nu = 0.3$ (assumed)

Reference
Shen, C. K., Bang, S., and Herrmann, L. R., "Ground Movement Analysis of Earth Support System," *Journal of the Geotechnical Engineering Division, ASCE*, **107**(GT12): 1609–1624 (1981).

♣ **Soil**: Sandy Silt

Description

- Density: $\gamma = 121.3$ lb/ft^3 ; Liquid limit = 19%, Plastic limit = 1%
- $D_{10} = 0.013$ mm ; $D_{30} = 0.045$ mm ; $D_{60} = 0.070$ mm

Parameter Values

- $K = 100.0$; $K_{ur} = 300$; $n = 0.84$; $R_f = 0.77$; $\phi = 27.0°$; $c = 7.5$ psi
- $\nu = 0.3$ (assumed)

Reference
Shen, C. K., Bang, S., and Herrmann, L. R., "Ground Movement Analysis of Earth Support System," *Journal of the Geotechnical Engineering Division, ASCE*, **107**(GT12): 1609–1624 (1981).

♣ **Soil**: Geosynthetic-Reinforced, Pile-Supported Earth Platforms

Parameter Values

For foundation soil:

- $K = 50$; $n = 0.45$; $R_f = 0.7$; $c = 0$; $\phi = 22°$; $\nu = 0.30$

For embankment fill:

- $K = 150$; $n = 0.25$; $R_f = 0.7$; $c = 0$; $\phi = 30°$; $\nu = 0.30$

Reference

Han, J. and Gabr, M. A., "Numerical Analysis of Geosynthetic-Reinforced and Pile-Supported Earth Platforms over Soft Soil," *Journal of Geotechnical and Geoenvironmental Engineering, ASCE*, **128**(1): 44–53 (2002).

♣ **Soil**: Geosynthetic-Encased Stone Columns

Description

- Density of stone column material: $\gamma = 20$ kN/m^3
- Density of foundation soil: $\gamma = 17$ kN/m^3
- Density of embankment fill: $\gamma = 17$ kN/m^3

Parameter Values

For stone column:

- $K = 1200$; $n = 0.7$; $R_f = 0.7$; $c = 0.0$; $\phi = 42°$; $\nu = 0.30$

For foundation soil:

- $K = 50$; $n = 0.5$; $R_f = 0.7$; $c = 20$ kPa ; $\phi = 0°$; $\nu = 0.45$

For embankment fill:

- $K = 150$; $n = 0.5$; $R_f = 0.7$; $c = 0.0$; $\phi = 30°$; $\nu = 0.30$

Reference

Murugesan, S. and Rajagopal, K. "Geosynthetic-Encased Stone Columns: Numerical Evaluation," *Geotextiles and Geomembranes*, **24**: 349–358 (2006).

♣ **Soil**: Estuarine Cohesive Soil

Description

- Dry density: $\gamma_d = 94$ lb/ft^3
- Liquid limit = 31% , Plastic limit = 10%
- Parameter values were determined from the results of undrained triaxial tests.

Parameter Values

- $K = 165.0$; K_{ur} = n/a ; $n = 1.10$; $R_f = 0.93$; $\phi = 19.0°$; $c = 460.0$ psf

Reference
Mitchell, J. K. and Huber, T. R. , "Performance of a Stone Column Foundation," *Journal of Geotechnical Engineering*, ASCE, **111**(2): 205–223 (1985).

♣ **Soil**: Older Marine Cohesive Soil

Description

- Dry density: $\gamma_d = 106$ lb/ft^3
- Liquid limit = 31%, Plastic limit = 10%
- Parameter values were determined from the results of undrained triaxial tests.

Parameter Values

- $K = 315.0$; K_{ur} = n/a ; $n = 0.43$; $R_f = 0.90$; $\phi = 19.0°$; $c = 460.0$ psf

Reference
Mitchell, J. K. and Huber, T. R. , "Performance of a Stone Column Foundation," *Journal of Geotechnical Engineering*, ASCE, **111**(2): 205–223 (1985).

Model of Kulhawy and Duncan

Some values of the parameters associated with the quasilinear elastic hyperbolic model proposed by Kulhawy and Duncan [73] are listed below.

♣ **Soil**: Estuarine Cohesive Soil

Description

- Dry density: $\gamma_d = 94$ lb/ft^3
- Liquid limit = 29%, Plastic limit = 9%.
- Parameter values were determined from the results of drained triaxial tests.

Parameter Values

- $K = 150.0$; K_{ur} = n/a ; $n = 0.65$; $R_f = 0.87$; $\phi = 34.0°$; $c = 0.0$
- $G = 0.21$; $F = -0.02$; $d = 6.8$

Reference
Mitchell, J. K. and Huber, T. R. , "Performance of a Stone Column Foundation," *Journal of Geotechnical Engineering*, ASCE, **111**(2): 205–223 (1985).

♣ **Soil**: Estuarine Cohesionless Soil

Description

- Dry density: $\gamma_d = 94$ lb/ft^3
- Parameter values were determined from the results of drained triaxial tests.

Parameter Values

- $K = 260.0$; K_{ur} = n/a ; $n = 0.69$; $R_f = 0.69$; $\phi = 38.0°$; $c = 0.0$
- $G = 0.24$; $F = -0.02$; $d = 13.4$

Reference
Mitchell, J. K. and Huber, T. R. , "Performance of a Stone Column Foundation," *Journal of Geotechnical Engineering*, ASCE, **111**(2): 205–223 (1985).

♣ **Soil**: Older Marine Cohesive Soil

Description

- Dry density: $\gamma_d = 106$ lb/ft^3
- Liquid limit = 27%, Plastic limit = 10%.
- Parameter values were determined from the results of drained triaxial tests.

Parameter Values

- $K = 150.0$; K_{ur} = n/a ; $n = 0.65$; $R_f = 0.84$; $\phi = 34.0°$; $c = 0.0$
- $G = 0.42$; $F = 0.42$; $d = 3.2$

Reference
Mitchell, J. K. and Huber, T. R. , "Performance of a Stone Column Foundation," *Journal of Geotechnical Engineering*, ASCE, **111**(2): 205–223 (1985).

♣ **Soil**: Older Marine Cohesionless Soil

Description

- Dry density: $\gamma_d = 106$ lb/ft^3
- Parameter values were determined from the results of drained triaxial tests.

Parameter Values

- $K = 190.0$; K_{ur} = n/a ; $n = 0.90$; $R_f = 0.67$; $\phi = 37.0°$; $c = 0.0$
- $G = 0.32$; $F = 0.18$; $d = 8.7$

Reference
Mitchell, J. K. and Huber, T. R. , "Performance of a Stone Column Foundation," *Journal of Geotechnical Engineering*, ASCE, **111**(2): 205–223 (1985).

♣ **Soil**: Sandy Gravel (stone column material)

Description

- Dry density: $\gamma_d = 116$ lb/ft^3
- Parameter values were determined from the results of drained triaxial tests.

Parameter Values

- $K = 390.0$; K_{ur} = n/a ; $n = 0.59$; $R_f = 0.86$; $\phi = 41.0°$; $c = 0.0$

- $G = 0.34$; $F = 0.20$; $d = 12.0$

Reference
Mitchell, J. K. and Huber, T. R. , "Performance of a Stone Column Foundation," *Journal of Geotechnical Engineering*, ASCE, **111**(2): 205–223 (1985).

♣ **Soil**: Leighton Buzzard Sand

Parameter Values

- $K = 1573.0$; $K_{ur} = 1890.0$; $n = 1.05$; $R_f = 0.90$; $\phi = 42.7°$; $c = 0.0$
- $G = 0.46$; $F = 0.17$; $D = 24.0$

Reference
Andrawes, K. Z., A. McGown, A., Wilson-Fahmy, R. F., and Mashhour, M. M., "The Finite Element Method of Analysis Applied to Soil-Geotextile Systems," *Proceedings of the Second International Conference of Geotextiles*, Las Vegas, Nevada, **III**: 695–700 (1982).

♣ **Soil**: Ottawa Sand

Description

- Subrounded Ottawa silica sand ; $G_s = 2.65$; Dry density (air dried): $\gamma_d = 107$ lb/ft^3

Parameter Values

- $K = 908.0$; $K_{ur} =$ n/a ; $n = 0.59$; $R_f = 0.86$; $\phi = 38.0°$; $c = 0.0$
- $G = 0.33$; $F = 0.01$; $d = 12.0$

Reference
Yin, Z. Z. and J. G. Zhu, "Stress and Deformation of Reinforced Soil Retaining Wall," *Geosynthetically-Reinforced Soil Retaining Walls*, J. T. H. Wu, editor, Balkema, Rotterdam: 205-216 (1992).

♣ **Soil**: Sandy Clay

Description

- Dark grey, slightly silty, fine to coarse sand and clay mixture.
- $G_s = 2.66$
- Water content = 19.3% ; Liquid limit = 37% ; Plastic limit = 19%

- Dry density (air dried): $\gamma_d = 100.7$ lb/ft^3

Parameter Values

- $K = 144.0$; $K_{ur} = 100.0$; $n = 0.06$; $R_f = 0.87$; $\phi = 26.0°$; $c = 2.0$ psi
- $G = 0.20$; $F = 0.10$; $d = 4.80$

Reference

Dechasakulsom, M., "Modeling Time-Dependent Behavior of Geogrids and its Application to Geosynthetically Reinforced Walls," PhD Dissertation, Department of Civil and Environmental Engineering, University of Delaware (2000).

♣ **Soil**: Sandy Clay

Description

- Dark grey, slightly silty, fine to coarse sand and clay mixture.
- $G_s = 2.66$
- Water content = 19.3% ; Liquid limit = 37% ; Plastic limit = 19%
- Dry density (air dried): $\gamma_d = 100.7$ lb/ft^3

Parameter Values

- $K = 144.0$; $K_{ur} =$ n/a ; $n = 0.06$; $R_f = 0.83$; $\phi = 26.0°$; $c = 2.0$ psi
- $G = 0.40$; $F = 0.10$; $d = 4.80$

Reference

Yin, Z. Z. and Zhu, J. G., "Stress and Deformation of Reinforced Soil Retaining Wall," *Geosynthetically-Reinforced Soil Retaining Walls*, J. T. H. Wu, editor, Balkema, Rotterdam: 205–216 (1992).

Model of Duncan

Some values of the parameters associated with the quasilinear elastic hyperbolic model proposed by Duncan [28] are listed below.

♣ **Soil**: Clean Ottawa sand

Parameter Values

- $K = 600.0$; $K_{ur} = 600.0$; $n = 0.40$; $R_f = 0.80$; $\phi = 35.0°$; $c = 1.0$ kPa
- $K_b = 300.0$; $m = 0.40$

Reference
Marcozzi, G. F., "Bearing Capacity of Rigid and Flexible Foundations on Dense Sand: An Experiment Study," Master's thesis, Department of Civil Engineering, University of Delaware, Newark, DE (1988).

♣ **Soil**: Leighton Buzzard sand

Description

- Density: $\gamma = 17.0$ kN/m^3

Parameter Values

- $K = 950.0$; $K_{ur} =$ n/a ; $n = 0.80$; $R_f = 0.70$; $\phi = 49.6°$; $c = 0.0\ kPa$
- $K_b = 250.0$; $m = 0.460$

Reference
Karpurapu, R. and Bathurst, R. J., "Numerical Investigation of Controlled Yielding of Soil-Retaining Wall Structures," *Geotextiles and Geomembranes*, **11**: 115–131 (1992).

♣ **Soil**: Ottawa sand

Parameter Values

- $K = 400.0$; $K_{ur} = 2000.0$; $n = 0.85$; $R_f = 0.86$; $\phi = 43.0°$; $c = 0.5$ psi
- $K_b = 700.0$; $m = 0.50$

Reference
Duncan, J. M., "Hyperbolic Stress-Strain Relationships," *Proceedings of the Workshop on Limit Equilibrium, Plasticity and Generalized Stress-Strain in Geotechnical Engineering*, edited by R. K. Yong and H-Y. Ko, ASCE press, 443–460 (1980).

♣ **Soil**: Ottawa Sand

Description

- Subrounded Ottawa silica sand ; $G_s = 2.65$; Dry density (air dried): $\gamma_d = 107$ lb/ft^3

Parameter Values

- $K = 1116.0$; $K_{ur} = 1500.0$; $n = 0.66$; $R_f = 0.87$; $\phi = 38.4°$; $c = 0.50$ psi
- $K_b = 907.0$; $m = 0.0$

Reference
Chou, N. N. S., "Performance Predictions of the Colorado Geotextile Test Walls," *Geosynthetically-Reinforced Soil Retaining Walls*, J. T. H. Wu, editor, Balkema, Rotterdam: 315–327 (1992).

♣ **Soil**: Ottawa Sand

Description

- Subrounded Ottawa silica sand ; $G_s = 2.65$; Dry density (air dried): $\gamma_d = 107$ lb/ft^3

Parameter Values

- $K = 1000.0$; $K_{ur} = 1500.0$; $n = 0.67$; $R_f = 0.85$; $\phi = 38.0°$; $c = 0.0$ psi
- $K_b = 700.0$; $m = 0.40$

Reference
Jaber, M., Schmertmann, G. R., and Collin, J. G., "Prediction of Geosynthetic Reinforced Wall Performance using Finite Element Analysis," *Geosynthetically-Reinforced Soil Retaining Walls*, J. T. H. Wu, editor, Balkema, Rotterdam: 305–313 (1992).

♣ **Soil**: Ottawa Sand

Description

- Subrounded Ottawa silica sand ; $G_s = 2.65$; Dry density (air dried): $\gamma_d = 107$ lb/ft^3

Parameter Values

- $K = 950.0$; K_{ur} = n/a ; $n = 0.36$; $R_f = 0.75$; $\phi = 38.0°$; $c = 0.0$ psi
- $K_b = 485.0$; $m = 0.64$

Reference
Karpurapu, R. and Bathurst, R. J., "Finite Element Predictions for the Colorado Geosynthetic-Reinforced Soil Retaining Walls," *Geosynthetically-Reinforced Soil Retaining Walls*, J. T. H. Wu, editor, Balkema, Rotterdam: 329–341 (1992).

♣ **Soil**: RMC Sand

Parameter Values

- $K = 950.0$; K_{ur} = n/a ; $n = 0.50$; $R_f = 0.75$; $\phi = 53.0°$; $c = 0.0$ psi
- $K_b = 250.0$; $m = 0.65$

Reference
Karpurapu, R. and Bathurst, R. J., "Behaviour of Geosynthetic Reinforced Soil Retaining Walls Using the Finite Element Method," *Computers and Geotechnics*, **17**: 279–299 (1995).

♣ **Soil**: Sand

Parameter Values

- $K = 913.0$; $K_{ur} = 1485.0$; $n = 0.60$; $R_f = 0.64$; $\phi = 46.1°$; $c = 0.0$ psi
- $K_b = 250.0$; $m = 0.80$

Reference
Zornberg, J. G. and Mitchell, J. K., "Finite Element Prediction of the Performance of an Instrumented Geotextile-Reinforced Wall," *Proceedings of the Eighth International Conference of the Association for Computer Methods and Advances in Geomechanics*, H. J. Siriwardane and M. M. Zaman, eds., Morgantown, West Virginia, II: 1433–1438 (1994).

♣ **Soil**: Sandy Clay

Description

- Dark grey, slightly silty, fine to coarse sand and clay mixture ; $G_s = 2.66$; Dry density (air dried): $\gamma_d = 100.7$ lb/ft^3
- water content = 19.3% ; Liquid limit = 37% ; Plastic limit = 19%.

Parameter Values

- $K = 370$, $K_{ur} = 500$; $n = 0.28$; $R_f = 0.98$; $\phi = 15.0°$; $c = 7.47$ psi : $K_b = 210.0$, $m = 0.0$

Reference

Chou, N. N. S., "Performance Predictions of the Colorado Geotextile Test Walls," *Geosynthetically-Reinforced Soil Retaining Walls*, J. T. H. Wu, editor, Balkema, Rotterdam: 315–327 (1992).

♣ **Soil**: Sandy Clay

Description

- Dark grey, slightly silty, fine to coarse sand and clay mixture ; $G_s = 2.66$; Dry density (air dried): $\gamma_d = 100.7$ lb/ft^3
- water content = 19.3% ; Liquid limit = 37% ; Plastic limit = 19%.

Parameter Values

- $K = 200.0$; $K_{ur} =$ n/a ; $n = 0.50$; $R_f = 0.92$; $\phi = 26.3°$; $c = 2.0$ psi ; $K_b = 600.0$, $m = 0.48$

Reference

Karpurapu, R. and Bathurst, R. J., "Finite Element Predictions for the Colorado Geosynthetic-Reinforced Soil Retaining Walls," *Geosynthetically-Reinforced Soil Retaining Walls*, J. T. H. Wu, editor, Balkema, Rotterdam: 329–341 (1992).

♣ **Soil**: Three Clays

For Clay "X":

- $K = 55.0$; $n = 0.0$; $R_f = 0.95$; $\phi = 30°$; $c = 0.0$; $K_b = 45.0$; $m = 0.0$

For Clay "Y":

- $K = 165.0$; $n = 0.0$; $R_f = 0.60$; $\phi = 0°$; $c = 8.0$ psi ; $K_b = 80.0$; $m = 0.0$

For Kaolinite:

- $K = 800.0$; $n = 0.0$; $R_f = 0.72$; $\phi = 0°$; $c = 28.5$ psi ; $K_b = 10000.0$; $m = 0.0$

Reference

Duncan, J. M., "Hyperbolic Stress-Strain Relationships," *Proceedings of the Workshop on Limit Equilibrium, Plasticity and Generalized Stress-Strain in Geotechnical Engineering*, edited by R. K. Yong and H-Y. Ko, ASCE press, 443–460 (1980).

32.5.2 Theoretical Considerations: Quasilinear Elastic Models Based on a Hyperbolic Relation

Konder and his co-workers [67; 69; 70; 68] approximated stress-strain curves for both clays and sands by the form shown in Figure 32.3. Analytically, such curves are represented by the following *hyperbolic* relation:

$$\sigma_1 - \sigma_3 = \frac{\varepsilon_1}{a + b\varepsilon_1} \tag{32.8}$$

where σ_1 and σ_3 are the major and minor principal stresses, respectively, ε_1 is the major principal strain, and a and b are model parameters whose values are determined by fitting experimental data.

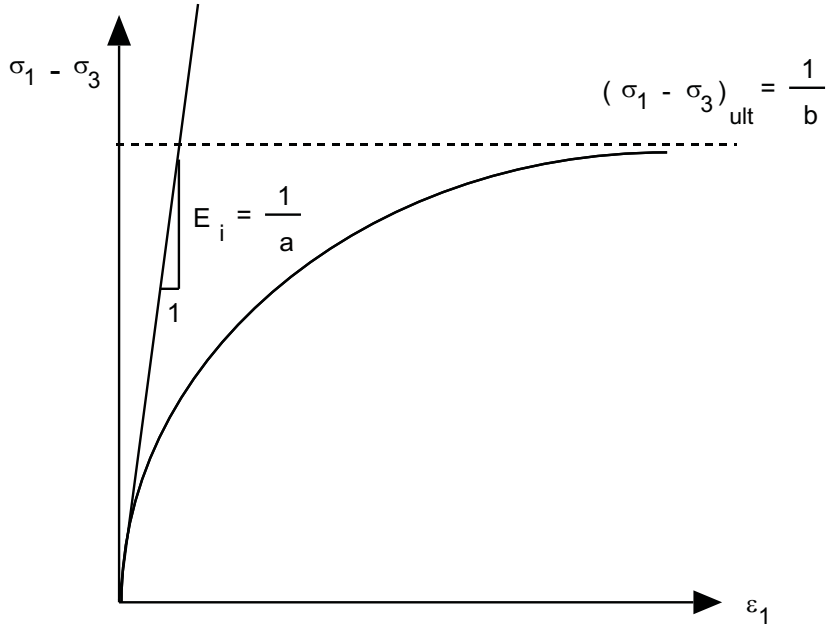

Figure 32.3: Hypothetical curve of principal stress difference versus axial strain for soil.

Assuming σ_3 constant, the tangent modulus E_t is given by

$$E_t = \frac{\partial(\sigma_1 - \sigma_3)}{\partial\varepsilon_1} = \frac{(a + b\varepsilon_1)(1) - b\varepsilon_1}{(a + b\varepsilon_1)^2} = \frac{a}{(a + b\varepsilon_1)^2} \tag{32.9}$$

Solving equation (32.8) for ε_1 gives

$$\varepsilon_1 = \frac{a(\sigma_1 - \sigma_3)}{1 - b(\sigma_1 - \sigma_3)} \tag{32.10}$$

Substituting this expression into equation (32.9) gives

$$E_t = \frac{1}{a}[1 - b(\sigma_1 - \sigma_3)]^2 \tag{32.11}$$

We next note that the initial tangent modulus E_i is the slope of the stress-strain curve when $(\sigma_1 - \sigma_3) = 0$. Evaluating equation (32.11) for this condition gives

$$E_i = \frac{1}{a}\left[1 - b\,(0)\right]^2 \quad \Rightarrow \quad a = \frac{1}{E_i} \tag{32.12}$$

In addition, we note that as $(\sigma_1 - \sigma_3) \to (\sigma_1 - \sigma_3)_{ult}$, $E_t \to 0$. Here $(\sigma_1 - \sigma_3)_{ult}$ is the asymptotic value of the principal stress difference at "infinite" strain (Figure 32.3). Evaluating equation (32.11) for this condition gives

$$E_t = \frac{1}{a}\left[1 - b\,(\sigma_1 - \sigma_3)_{ult}\right]^2 = 0 \quad \Rightarrow \quad b = \frac{1}{(\sigma_1 - \sigma_3)_{ult}} \tag{32.13}$$

Substituting the expressions for a and b into equation (32.11) gives the following expression for E_t:

$$E_t = E_i \left[1 - \frac{(\sigma_1 - \sigma_3)}{(\sigma_1 - \sigma_3)_{ult}}\right]^2 \tag{32.14}$$

Clearly, as $(\sigma_1 - \sigma_3) \to (\sigma_1 - \sigma_3)_{ult}$, $E_t \to 0$.

Remark: The fact that equation (32.8) involves only ε_1 and the principal stress difference implies an *axisymmetric* triaxial material state (i.e., $\sigma_2 = \sigma_3$, $\varepsilon_2 = \varepsilon_3$).

Remark: From the form of the stress-strain curve shown in Figure 32.3, it is evident that the hyperbolic idealization can only be accurate for normally consolidated or lightly overconsolidated clays and for loose sands; that is, in cases where there is no appreciable drop from a peak principal stress difference to a residual value.

Remark: The hyperbolic idealization says nothing about the volume change of the material. Indeed, since the model assumes elastic isotropy, it follows that no volume changes will be produced upon shearing, which is *counter* to experimental observations.

Remark: If equation (32.8) is re-written as

$$\frac{\varepsilon_1}{\sigma_1 - \sigma_3} = b\varepsilon_1 + a$$

and if we view ε_1 as the abscissa and $\varepsilon_1/(\sigma_1 - \sigma_3)$ as the ordinate, then the hyperbolic stress-strain idealization plots as a straight line (Figure 32.4). The slope of this line is b, and the intercept is a.

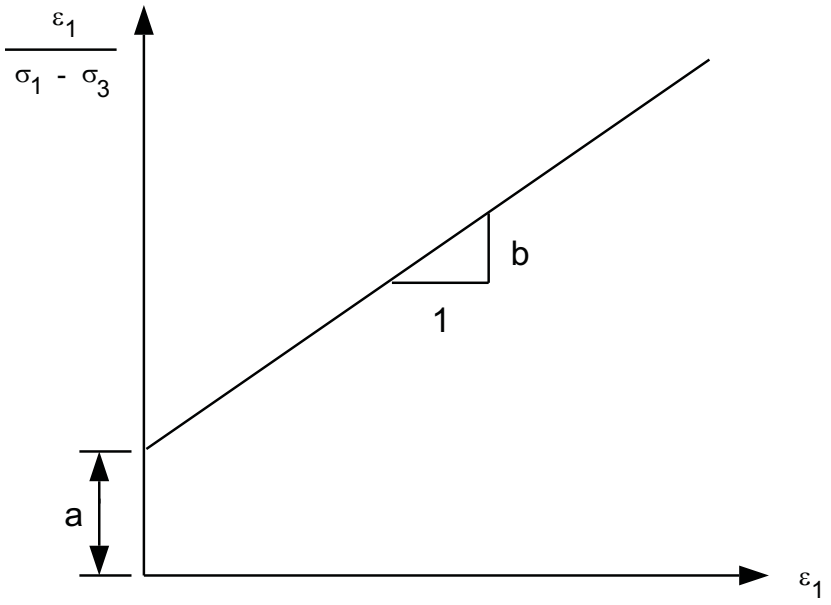

Figure 32.4: Transformed hyperbolic stress-strain relations.

32.5.2.1 Model of Duncan and Chang [29]

Citing the previous experimental findings of Janbu [47], Duncan and Chang [29] noted that each of the constants a and b should be dependent on the minor principal (confining) effective stress σ_3. More precisely, they suggested that E_i vary in the following manner:

$$E_i = \frac{1}{a} = K p_a \left(\frac{\sigma_3}{p_a}\right)^n \tag{32.15}$$

where K is a dimensionless "modulus number," p_a is the atmospheric pressure (with the same units as σ_3), and n is a dimensionless "modulus exponent" that determines the rate of variation of E_i with σ_3.

Duncan and Chang further suggested the following relation between principal stress difference at failure (or the "compressive strength") $(\sigma_1 - \sigma_3)_f$ and its ultimate value:

$$(\sigma_1 - \sigma_3)_f = R_f \, (\sigma_1 - \sigma_3)_{ult} \tag{32.16}$$

where R_f is a "failure ratio" ($R_f \leq 1.0$).

The principal stress difference at failure was assumed to be related to the minor principal (confining) stress through the Mohr-Coulomb failure criterion; viz.,

$$(\sigma_1 - \sigma_3)_f = \frac{2(c \cos\phi + \sigma_3 \sin\phi)}{1 - \sin\phi} \tag{32.17}$$

where c is the cohesion intercept and ϕ is the angle of internal friction.

Using equations (32.16) and (32.17), it follows that

$$b = \frac{1}{(\sigma_1 - \sigma_3)_{ult}} = \frac{R_f}{(\sigma_1 - \sigma_3)_f} = \frac{R_f(1 - \sin\phi)}{2(c \cos\phi + \sigma_3 \sin\phi)} \tag{32.18}$$

Substituting equations (32.15) and (32.18) into the first of equation (32.14) gives

$$E_t = K p_a \left(\frac{\sigma_3}{p_a}\right)^n \left[1 - \frac{R_f(1-\sin\phi)(\sigma_1-\sigma_3)}{2(c\,\cos\phi + \sigma_3 \sin\phi)}\right]^2 \tag{32.19}$$

In general, the value of E_t will be too low to properly account for unloading. Consequently, based on the results of a number of loading-unloading-reloading tests on sands, Duncan and Chang [29] proposed the following relationship for the elastic modulus associated with unloading and reloading:

$$E_{ur} = K_{ur} p_a \left(\frac{\sigma_3}{p_a}\right)^n \tag{32.20}$$

where E_{ur} is the unloading-reloading modulus, and K_{ur} is the corresponding dimensionless modulus number. Thus, for histories involving loading, unloading and reloading, the *same* value (E_{ur}) of the elastic modulus is used. Admittedly this is a simplification of the actual material behavior.

Experimental results indicate that, in general, $E_{ur} > E_t$. In particular, based on the results of a large number of laboratory experiments, Wong and Duncan [114] concluded that K_{ur} was 1 to 3 times higher than K_i, whereas n was essentially identical for initial loading and for unloading-reloading.

Duncan and Chang [29] assumed the second elastic constant, Poisson's ratio (ν), to be *constant*. Kulhawy and Duncan [73] subsequently modified this assumption in the manner described in the next sub-section.

32.5.2.2 Modifications Proposed by Kulhawy and Duncan [73]

While maintaining the same expressions for E_i, E_t and E_{ur} as presented by Duncan and Chang [29], Kulhawy and Duncan [73] proposed the following relation between the axial strain ε_a and the radial strain ε_r:

$$\varepsilon_a = -\frac{\varepsilon_r}{f + D\varepsilon_r} \tag{32.21}$$

where f and D are model parameters. Noting that the tangent Poisson's ratio is defined by

$$\nu_t = -\frac{\partial \varepsilon_r}{\partial \varepsilon_a} = \frac{f}{(1 - D\varepsilon_a)^2} \tag{32.22}$$

we see that f equals the initial value ν_i of the tangent Poisson's ratio at zero strain and d expresses the rate of change of ν_i with strain.

Noting that ν_i generally decreases with increasing confining stress σ_3, Kulhawy and Duncan proposed the following expression:

$$\nu_i = G - F \log\left(\frac{\sigma_3}{p_a}\right) \equiv f \tag{32.23}$$

where G is the value of ν_i at $\sigma_3 = p_a$ and F is the decrease of ν_i for a ten-fold increase in σ_3. Substituting equation (32.23) into (32.22) gives

$$\nu_t = \frac{G - F \log\left(\frac{\sigma_3}{p_a}\right)}{(1 - D\varepsilon_a)^2} \tag{32.24}$$

where D is a model parameter.

From equation (32.8) it follows that the major principal strain is

$$\varepsilon_1 = \frac{a(\sigma_1 - \sigma_3)}{1 - b(\sigma_1 - \sigma_3)} \tag{32.25}$$

Assuming that $\varepsilon_1 = \varepsilon_a$, and using equations (32.15) and (32.18) leads to

$$\varepsilon_1 = \frac{(\sigma_1 - \sigma_3)}{Kp_a \left(\frac{\sigma_3}{p_a}\right)^n \left[1 - \frac{R_f(1 - \sin\phi)(\sigma_1 - \sigma_3)}{2(c\,\cos\phi + \sigma_3 \sin\phi)}\right]} \tag{32.26}$$

Equation (32.24), with $\varepsilon_1 = \varepsilon_a$ given by equation (32.26), is thus used to compute the tangent Poisson's ratio; viz.,

$$\nu_t = \frac{G - F \log\left(\frac{\sigma_3}{P_a}\right)}{\left[1 - \frac{D\,(\sigma_1 - \sigma_3)}{KP_a \left(\frac{\sigma_3}{P_a}\right)^n \left[1 - \frac{R_f\,(1 - \sin\phi)\,(\sigma_1 - \sigma_3)}{2\,(c\cos\phi + \sigma_3 \sin\phi)}\right]^2}\right]^2} \tag{32.27}$$

The description of Poisson's ratio is thus controlled by values of the three model parameters G, F and D.

32.5.2.3 Modification Proposed by Herrmann [39]

Herrmann [39] presented a slightly modified version of the Duncan-Chang model. The modification was made in order to avoid a serious problem that was observed in a number of analyses. In particular, the problem occurred for regions where the confining pressure σ_3 was sufficiently low to yield a low modulus, but not low enough to yield a high Poisson's ratio. The result was that the soil in these regions experienced large decreases in volume; i.e., it effectively disappeared, thus violating the conservation of mass. To prevent this problem, the soil was assumed to possesses a constant bulk modulus B. The Poisson's ratio was then calculated from the following expression:

$$\nu_t = \frac{1}{2}\left(1 - \frac{E_t}{3B}\right) \tag{32.28}$$

A check is commonly made to ensure that $0 \leq \nu_t \leq 0.49$.

32.5.2.4 Modification Proposed by Duncan [28]

While maintaining the same expressions for E_i, E_t and E_{ur} as presented by Duncan and Chang [29], Duncan [28] assumed that B depends only on the confining stress σ_3 and is

independent of the principal stress difference. The following equation for the tangent elastic bulk modulus was thus proposed:

$$B_t = K_b p_a \left(\frac{\sigma_3}{P_{atm}} \right)^m \tag{32.29}$$

where K_b is a dimensionless "bulk modulus number" (equal to the value of B/p_a at $\sigma_3 = p_a$) and m is a dimensionless "bulk modulus exponent" (equal to the change in B/p_a for a ten-fold increase in σ). For the case of loading-unloading-reloading histories, no modification of equation (32.29) was proposed by Duncan [28].

Using equation (32.29) for the bulk modulus, Poisson's ratio is computed from

$$\nu_t = \frac{1}{2} \left(1 - \frac{E_t}{3B_t} \right) \tag{32.30}$$

A check is commonly made to ensure that $0 \leq \nu_t \leq 0.49$.

32.6 The VARIABLE_MODULUS JANBU command

Synopsis

The **VARIABLE_MODULUS JANBU** command is used to specify the parameters associated with the following nonlinear hypoelastic model for the instantaneous elastic modulus (E_i) proposed by Janbu [47]:

$$E_i = K_i p_a \left(\frac{\sigma_3'}{p_a} \right)^n$$

where K_i is a dimensionless "modulus number," p_a is the atmospheric pressure (with the same units as σ_3'), and $n$is a dimensionless "modulus exponent" that determines the rate of variation of E_i with σ_3'.[4] Additional details pertaining to theoretical aspects of Janbu's model are given in Section (32.6.2).

Syntax

The following syntax is used to describe the various forms of the **VARIABLE_MODULUS JANBU** idealization:

MATerial VARiable_modulus JANbu NUMber #
(DEScription "string")
(MODULUS_Number #.#)
(MODULUS_Exponent #.#)
(POIssons_ratio #.#)
(ATMospheric_pressure #.#)

Explanatory Notes

- The **NUMBER** keyword is used to specify the (global) number of the material associated with the incompressible isotropic elastic idealization. The default material number is one (1).
- The optional alphanumeric string associated with the **DESCRIPTION** keyword must be enclosed in double quotes ("). It is used solely to describe the material being idealized to the analyst. The **DESCRIPTION** string is printed as part of the "echo" of the material.

[4] It is evident that, in light of the above definitions E_i will have the same units as p_a.

- The default value of the dimensionless **MODULUS_NUMBER** (K_i) is one (1.0).
- The default value of the dimensionless **MODULUS_EXPONENT** (n) is one (1.0).
- The keyword **POISSONS_RATIO** is used to specify the value of the Poisson's ratio (ν). The default value for ν is 0.20.
- The default value of **ATMOSPHERIC_PRESSURE** is 101.4 (kPa).

Example of Command Usage

The following commands are used to specify the material parameters for variable modulus Janbu material idealization:

```
material variable_modulus Janbu num 1  &
desc "parameters for Sacramento River sand" &
modulus_number  1000.0  modulus_exponent 1.00 &
atmospheric_pre 101.4 &
poissons_ratio 0.20
```

32.6.1 Model Parameter Values Used in Past Analyses

The model parameters K_i and n can be determined by plotting E_i/p_a for several σ_3'/p_a values on a log-log diagram and fitting a straight line. The slope of this line is n. The parameter K_i is the value of E_i/p_a at $\sigma_3'/p_a = 1$.

Janbu [47] presented the results of a large number of tests on sands and silty clays. He determined values of K_i ranging from 50 to 500 and values of n ranging from 0.35 to 0.55.

Lade and Kim [78] presented the following values for the parameters associated with Janbu's model. Also listed is the Poisson's ratio (ν) and, where applicable, the relative density (D_r).

- **Fine Silica Sand** [78] ($D_r = 30\%$): $K_i = 1170$, $n = 0.53$, $\nu = 0.20$.
- **Sacramento River Sand** [80] ($D_r = 100\%$): $K_i = 1460$, $n = 0.47$, $\nu = 0.20$.
- **Sacramento River Sand** [80] ($D_r = 38\%$): $K_i = 890$, $n = 0.34$, $\nu = 0.20$.
- **Painted Rock Material** [2] ($D_r = 100\%$): $K_i = 891$, $n = 0.51$, $\nu = 0.20$.
- **Painted Rock Material** [2] ($D_r = 70\%$): $K_i = 314$, $n = 0.75$, $\nu = 0.20$.
- **Crushed Napa Basalt** [1] ($D_r = 100\%$): $K_i = 470$, $n = 0.58$, $\nu = 0.20$.
- **Crushed Napa Basalt** [1] ($D_r = 70\%$): $K_i = 481$, $n = 0.36$, $\nu = 0.20$.
- **Monterey 0 Sand** [75; 77] $D_r = 98\%$): $K_i = 2300$, $n = 0.80$, $\nu = 0.20$.
- **Monterey 0 Sand** [75; 77] $D_r = 27\%$): $K_i = 1600$, $n = 0.86$, $\nu = 0.20$.
- **Edgar Kaolinite Clay** [78]: $K_i = 110$, $n = 1.23$, $\nu = 0.25$.

32.6.2 Theoretical Considerations

The major premise of Janbu's model [47] is that the initial tangent modulus E_i (e.g., the initial slope of the stress-strain curve associated with an axisymmetric triaxial compression test) may be expressed as a power function of the initially isotropic, effective confining stress σ_3'. In particular,

$$E_i = K_i p_a \left(\frac{\sigma_3'}{p_a}\right)^n \tag{32.31}$$

where K_i is a dimensionless "modulus number," p_a is the atmospheric pressure (with the same units as σ_3'), and $n$is a dimensionless "modulus exponent" that determines the rate of variation of E_i with σ_3'.

Based on stress-strain data obtained from axisymmetric triaxial tests, it is evident that the unloading-reloading modulus is higher than the initial tangent modulus E_i, which may be influenced by inelastic deformations. Consequently, Duncan and Chang [29] proposed the following expression for the elastic modulus:

$$E_{ur} = K_{ur} p_a \left(\frac{\sigma_3}{p_a}\right)^n \tag{32.32}$$

where E_{ur} is the unloading-reloading modulus, and K_{ur} is the corresponding dimensionless modulus number. Thus, for histories involving loading, unloading and reloading, the *same* value (E_{ur}) of the elastic modulus is used. Based on the results of a large number of laboratory experiments, Wong and Duncan [114] concluded that K_{ur} was 1 to 3 times higher than K_i, whereas n was essentially identical for initial loading and for unloading-reloading.

Equation (32.31) has been rather widely used and appears to capture the elastic response of soils rather well. However, as noted by Zytynski et al. [119], the variation of E_i using this equation violates the principle of conservation of energy. Thus, depending on the direction of a closed stress loop, the model will generate or dissipate energy in violation of the basic premise of elastic behavior.

32.7 The VARIABLE_MODULUS LADE_NELSON command

Synopsis

The **VARIABLE_MODULUS LADE_NELSON** command is used to specify the parameters associated with the nonlinear hypoelastic model proposed by Lade and Nelson [79].

Based on theoretical considerations involving the principle of conservation of energy, Lade and Nelson [79] proposed the following isotropic hypoelastic model for the elastic modulus E:

$$E = Mp_a \left[\left(\frac{I_{1\sigma}}{p_a} \right)^2 + R \frac{J_2}{(p_a)^2} \right]^{\lambda}$$

where $I_{1\sigma}$ is the first invariant of the stress (or effective stress) tensor, J_2 is the second invariant of the deviatoric stress tensor, p_a is the atmospheric pressure (expressed in the same units as E, $I_{1\sigma}$ and $\sqrt{J_2}$), M is a dimensionless modulus number, λ is a dimensionless exponent, and $R = 6(1+\nu)/(1-2\nu) = 9K/G$. The Poisson's ratio ($\nu$) is assumed to be constant. Additional details pertaining to theoretical aspects of the Lade-Nelson model are given in Section (32.7.2).

Syntax

The following syntax is used to describe the various forms of the **VARIABLE_MODULUS LADE_NELSON** idealization:

MATerial VARiable_modulus LADe_nelson NUMber #
(DEScription "string")
(M_Parameter #.#)
(LAMbda #.#)
(POIssons_ratio #.#)
(ATMospheric_pressure #.#)

Explanatory Notes

- The **NUMBER** keyword is used to specify the (global) number of the material associated with the incompressible isotropic elastic idealization. The default material number is one (1).

- The optional alphanumeric string associated with the **DESCRIPTION** keyword must be enclosed in double quotes ("). It is used solely to describe the material being idealized to the analyst. The **DESCRIPTION** string is printed as part of the "echo" of the material.
- The default value of the **M_PARAMETER** modulus (M) is 500.0.
- The default value of the exponent **LAMBDA** (λ) is 0.25.
- The keyword **POISSONS_RATIO** is used to specify the value of the Poisson's ratio (ν). The default value for ν is 0.20.
- The default value of **ATMOSPHERIC_PRESSURE** is 101.4 (kPa).

Example of Command Usage

The following commands are used to specify the material parameters for variable modulus Lade-Nelson material idealization:

```
material variable_modulus Lade_Nelson  num 2  &
desc "parameters for Sacramento River sand"  &
m_parameter 400.0   lambda 0.20 &
atmospheric_pre 101.4 &
poissons_ratio 0.20
```

32.7.1 Model Parameter Values Used in Past Analyses

The model parameters M and λ can be determined by plotting E/p_a versus $[(I_{1\sigma}/p_a)^2 + R(J_2/(p_a)^2)]$ values on a $\log - \log$ diagram and fitting a straight line.

- The slope of this line is λ.
- The parameter M is the value of E/p_a at $[(I_{1\sigma}/p_a)^2 + R(J_2/(p_a)^2)] = 1$.

Lade and Kim [78] presented the following values for the parameters associated with their model. Also listed is the Poisson's ratio (ν) and, where applicable, the relative density (D_r).

- **Sand No. D** [76] ($D_r = 89\%$): $M = 400$, $\lambda = 0.37$, $\nu = 0.23$.
- **Fine Silica Sand** [76] ($D_r = 30\%$): $M = 440$, $\lambda = 0.22$, $\nu = 0.27$.
- **Sacramento River Sand** [80] ($D_r = 100\%$): $M = 900$, $\lambda = 0.28$, $\nu = 0.20$.
- **Sacramento River Sand** [80] ($D_r = 38\%$): $M = 510$, $\lambda = 0.28$, $\nu = 0.20$.
- **Painted Rock Material** [2] ($D_r = 100\%$): $M = 920$, $\lambda = 0.24$, $\nu = 0.20$.
- **Painted Rock Material** [2] ($D_r = 70\%$): $M = 350$, $\lambda = 0.33$, $\nu = 0.20$.
- **Crushed Napa Basalt** [1] ($D_r = 100\%$): $M = 1050$, $\lambda = 0.17$, $\nu = 0.20$.
- **Crushed Napa Basalt** [1]($D_r = 70\%$): $M = 590$, $\lambda = 0.19$, $\nu = 0.20$.
- **Monterey 0 Sand** [75; 77] $D_r = 98\%$): $M = 1120$, $\lambda = 0.33$, $\nu = 0.17$.
- **Monterey 0 Sand** [75; 77] $D_r = 27\%$): $M = 800$, $\lambda = 0.26$, $\nu = 0.17$.
- **Santa Monica Beach Sand** [76] ($D_r = 88.5\%$): $M = 1270$, $\lambda = 0.23$, $\nu = 0.15$.
- **Santa Monica Beach Sand** [76] ($D_r = 64.9\%$): $M = 1050$, $\lambda = 0.24$, $\nu = 0.19$.
- **Santa Monica Beach Sand** [76] ($D_r = 39.2\%$): $M = 820$, $\lambda = 0.26$, $\nu = 0.22$.
- **Santa Monica Beach Sand** [76] ($D_r = 18.4\%$): $M = 600$, $\lambda = 0.27$, $\nu = 0.26$.

32.7.2 Theoretical Considerations

Based on a rather extensive review of the literature, Lade and Nelson [79] determined that the elastic behavior of granular materials was isotropic and that Poisson's ratio is constant (recall the discussion of isotropy and Poisson's ratio presented in earlier sub-sections).

Lade and Nelson [79] suggested that the elastic modulus E, the bulk modulus K, and the shear modulus G should vary with the mean normal stress $\sigma_{kk}/3$ and with the deviatoric stress s_{ij}. In addition, energy should neither be generated nor dissipated in closed-loop stress or strain paths.

Based on theoretical considerations involving the principle of conservation of energy, Lade and Nelson [79] proposed the following isotropic hypoelastic model for the elastic modulus E:

$$E = Mp_a \left[\left(\frac{I_{1\sigma}}{p_a} \right)^2 + R \frac{J_2}{(p_a)^2} \right]^{\lambda} \tag{32.33}$$

where $I_{1\sigma}$ is the first invariant of the stress (or effective stress) tensor, J_2 is the second invariant of the deviatoric stress tensor, p_a is the atmospheric pressure (expressed in the same units as E, $I_{1\sigma}$ and $\sqrt{J_2}$), M is a dimensionless modulus number, λ is a dimensionless exponent, and $R = 6(1+\nu)/(1-2\nu) = 9K/G$. The Poisson's ratio ($\nu$) is assumed to be constant.

Since the model of Lade and Nelson uses stress invariants, it is seen to be more general than the simpler expression given by Janbu [47]; viz.,

$$E_i = K_i p_a \left(\frac{\sigma_3'}{p_a} \right)^n \tag{32.34}$$

where K_i is a dimensionless "modulus number," p_a is the atmospheric pressure (with the same units as σ_3'), and $n$is a dimensionless "modulus exponent" that determines the rate of variation of E_i with σ_3'. Janbu's idealization is, strictly speaking, applicable only to triaxial (axisymmetric) conditions.

32.8 The VARIABLE_MODULUS QUBAIN command

Synopsis

The **MATERIAL VARIABLE MODULUS QUBAIN** command is used to specify the parameters associated with the nonlinear hypoelastic model proposed by Qubain et al. [100]. Additional details pertaining to theoretical aspects of this model are given in Section (32.8.2).

Syntax

The following syntax is used to describe the various forms of the **VARIABLE_MODULUS QUBAIN** idealization:

MATerial VARiable_modulus QUBain NUMber #
(DEScription "string")
(INItial_bulk #.#)
(EPS_ultimate #.#)
(POIssons_ratio #.#)
(ATMospheric_pressure #.#)

Explanatory Notes

- The **NUMBER** keyword is used to specify the (global) number of the material associated with the incompressible isotropic elastic idealization. The default material number is one (1).
- The optional alphanumeric string associated with the **DESCRIPTION** keyword must be enclosed in double quotes ("). It is used solely to describe the material being idealized to the analyst. The **DESCRIPTION** string is printed as part of the "echo" of the material.
- The default value of the **INITIAL_BULK** modulus is one (1.0).
- The default value of the ultimate strain **EPS_ULTIMATE** is 0.01.
- The keyword **POISSONS_RATIO** is used to specify the value of the Poisson's ratio (ν). The default value for ν is 0.20.
- The default value of **ATMOSPHERIC_PRESSURE** is 101.4 (kPa).

Example of Command Usage

The following commands are used to specify the material parameters for variable bulk modulus material idealization of Qubain et al. [100]:

```
material variable_modulus Lade_Nelson  num 2  &
desc "parameters for Sacramento River sand"  &
m_parameter 400.0   lambda 0.20 &
atmospheric_pre 101.4 &
poissons_ratio 0.20
```

Examples of Command Usage

The following commands are used to specify the material parameters for variable bulk modulus material idealization of Qubain et al. [100] for loose (relative density = 20 %) and dense (relative density = 80 %) Chattahoochee sand:

```
material variable_modulus Qubain num 1  &
desc "parameters for LOOSE Chattahoochee sand" &
initial_bulk  26316.0  eps_ultimate 0.356 &
poissons_ratio 0.25

material variable_modulus Qubain num 2  &
desc "parameters for DENSE Chattahoochee sand" &
initial_bulk  26316.0  eps_ultimate 0.384  &
poissons_ratio 0.25
```

32.8.1 Model Parameter Values Used in Past Analyses

The model parameters b and m are determined by plotting ε_v/σ'_m versus ε_v and fitting a straight line in the manner shown in Figure 32.6. The slope of this line is m; the y-intercept is b. Because they have more physical meaning, the quantities K_{in} $(= 1/b)$ and ε_{v_ult} $(= b/m)$ are typically used instead of b and m.

Qubain [99] performed a series of axisymmetric triaxial compression tests (both with shear stress and under isotropic conditions) on loose, medium dense and dense samples of a medium to fine, uniform, quartz sand ($G_s = 2.63$). The following model parameter values were determined from the isotropic compression tests:

- **Loose Sand** ($D_r = 28$ %), $K_{in} = 13,900\,kPa$ and $\varepsilon_{v\_ult} = 2.66$ %.
- **Medium Dense Sand** ($D_r = 51$ %), $K_{in} = 15,300\,kPa$ and $\varepsilon_{v\_ult} = 1.73$ %.
- **Dense Sand** ($D_r = 88$ %), $K_{in} = 14,600\,kPa$ and $\varepsilon_{v\_ult} = 0.98$ %.

where D_r is the relative density.

In addition, Qubain et al. [100] gave the following values for the model parameters associated with loose and dense Chattahoochee sand:

- **Loose** ($D_r = 20$ %) [112]: $K_{in} = 26,316\,kPa$ and $\varepsilon_{v\_ult} = 35.6$ %.
- **Dense** ($D_r = 80$ %) [112]: $K_{in} = 83,333\,kPa$ and $\varepsilon_{v\_ult} = 38.4$ %.

The validity of this constitutive relationship has been shown by Qubain et al. [100]. Model simulations agreed quite closely with the results of isotropic compression experiments on the aforementioned Chattahoochee sand.

32.8.2 Theoretical Considerations

Consider a typical generalized mean stress-volumetric strain diagram associated with an isotropic compression test on sand (Figure 32.5). It is evident that the tangential bulk modulus increases with increasing stress-strain levels. Sand obviously sand stiffens upon loading. Using a somewhat more phenomenological approach as compared to Lade and Nelson [79], Qubain et al. [100] have captured this unique property; the associated constitutive model is described in this sub-section.

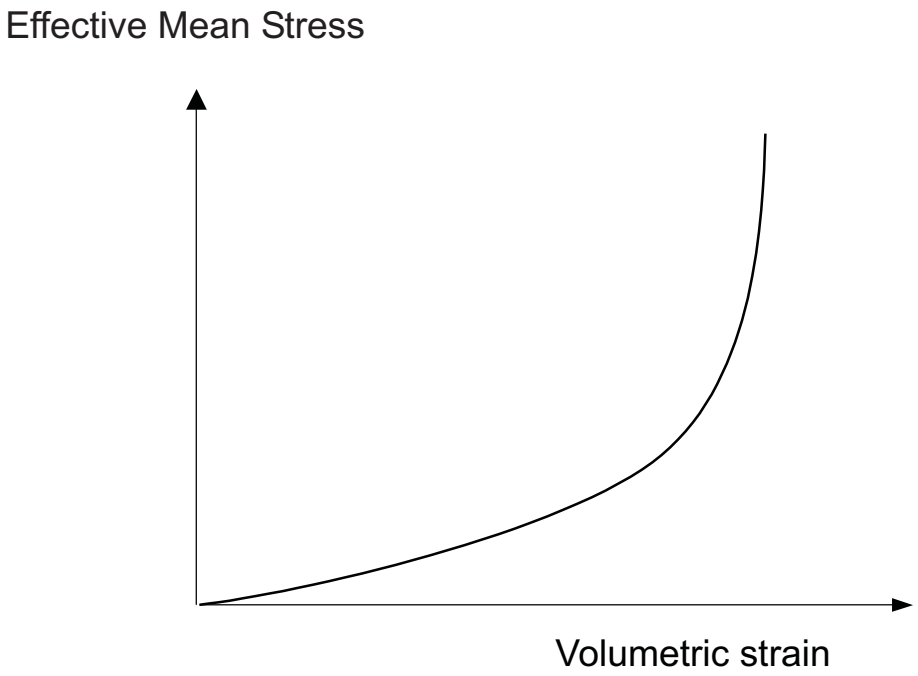

Figure 32.5: Generalized stress-strain diagram for isotropic compression of sand.

Before proceeding, it is important to note that the general shape of the isotropic stress-strain response holds *both* for dense and loose sand specimens. The only difference is the degree of curvature of the mean stress-volumetric strain curve. The same is not true for triaxial (axisymmetric) test results.

Certain researchers [67; 69] have suggested the use of a hyperbola to represent the above simple constitutive relationships such as that shown in Figure 32.5. The hyperbola offers the advantage that it can be manipulated in such a way as to produce the straight-line representation of the response. In particular, Qubain et al. [100] derived the following representation for the effective mean stress in terms of the volumetric strain ε_v and the two model parameters b and m:

$$\frac{\varepsilon_v}{\sigma'_m} = b - m\varepsilon_v \tag{32.35}$$

This relationship is shown schematically in Figure 32.6.

Solving for the mean effective stress from equation (32.35) gives

$$\sigma'_m = \frac{\varepsilon_v}{b - m\varepsilon_v} \tag{32.36}$$

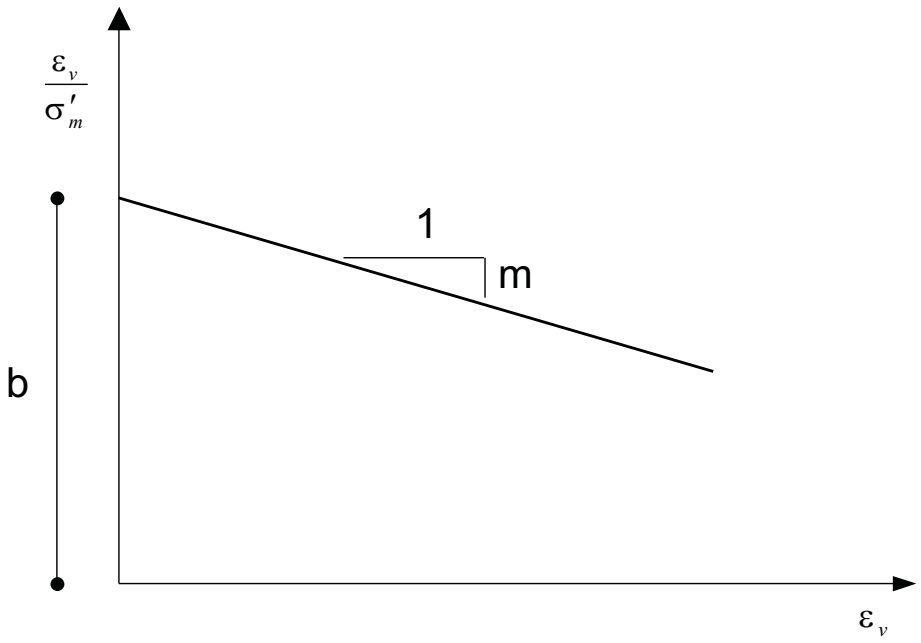

Figure 32.6: Transformed hyperbolic stress-strain diagram for isotropic compression on sand.

The initial tangential bulk modulus is then

$$K_{in} = \left(\frac{d\sigma'_m}{d\varepsilon_v}\right)_{\varepsilon_v=0} = \frac{1}{b} \tag{32.37}$$

The initial bulk modulus is thus simply the inverse of the intercept of the straight line shown in the Figure 32.6. It is next necessary to determine the *ultimate* volumetric strain. This is achieved solving equation (32.35) for ε_v and then evaluating the resulting expression as σ'_m approaches infinity in the limit. This gives the following result:

$$\varepsilon_{v\_ult} = \lim_{\sigma'_m \to \infty} \frac{b\sigma'_m}{1 + m\sigma'_m} = \lim_{\sigma'_m \to \infty} \frac{b}{m} = \frac{b}{m} \tag{32.38}$$

The tangential bulk modulus K_t is next obtained by differentiating equation (32.36) with respect to ε_v. This gives the following expression:

$$K_t = \left(\frac{d\sigma'_m}{d\varepsilon_v}\right) = \frac{b}{b^2 - 2bm\varepsilon_v + m^2(\varepsilon_v)^2} \tag{32.39}$$

The modulus K_t can likewise be written as a function of the volumetric strain and the initial bulk modulus K_{in} and the ultimate volumetric strain ε_{v_ult}; viz.,

$$K_t = \frac{K_{in}}{1 - 2\left(\frac{\varepsilon_v}{\varepsilon_{v\_ult}}\right)\left(\frac{\varepsilon_v}{\varepsilon_{v\_ult}}\right)^2} \tag{32.40}$$

Chapter 33

MATERIAL Idealizations Suitable For Polymeric Reinforcement

This chapter contains information associated with the description of material idealizations suitable for simulating the behavior of polymeric inclusions (e.g., geosynthetics) that are available for use in the APES$^+$ computer program.

33.1 Available Material Idealizations Suitable For Polymeric Reinforcement

The following material idealizations, suitable for simulating the behavior of polymeric inclusions (e.g., geosynthetics), are currently available in the APES$^+$ computer program:

- Time-dependent **GEOSYNTHETIC D_K** material idealization (after Dechasakulsom and Kaliakin) suitable for geosynthetics (see Section 33.2).
- Time-independent **GEOSYNTHETIC L_T** material idealization (after Ling and Tatsuoka [82]) suitable for geosynthetics (see Section 33.3).

33.2 The MATERIAL GEOSYNTHETIC D_K command

Synopsis

The **MATERIAL GEOSYNTHETIC D_K** command is used to describe the time-dependent material characterization of Dechasakulsom and Kaliakin [56] suitable for geosynthetic reinforcement.

Syntax

The following syntax is used to describe the various forms of the **MATERIAL GEOSYNTHETIC D_K** idealization:

MATerial GEOsynthetic D_K NUMber #
(DEScription "string")
(C_1 #.#) (C_2 #.#) (ALPha #.#) (BETa #.#)
(INItial_stiffness #.#)

Explanatory Notes

The **MATERIAL GEOSYNTHETIC D_K** command is commonly used in conjunction with a finite deformation (L2FDP0) line element. The linear specific constitutive law for the material is assumed to be of the form

$$f(t) = \frac{c_1\,\varepsilon^{c_2}}{1+\alpha\,t^{\beta}}$$

where f is the force in the geosynthetic per unit width, ε is the (assumed uniaxial) strain in the geosynthetic, and c_1, c_2, α, and β are constant model parameters.

- The **NUMBER** keyword is used to specify the (global) number of the material associated with the incompressible isotropic elastic idealization. The default material number is one (1).
- The optional alphanumeric string associated with the **DESCRIPTION** keyword must be enclosed in double quotes ("). It is used solely to describe the material being idealized to the analyst. The **DESCRIPTION** string is printed as part of the "echo" of the material.
- The keyword **C_1** corresponds to the parameter c_1. The default value of **C_1** is 1.0.

- The keyword **C_2** corresponds to the dimensionless parameter c_2. The default value of **C_2** is 0.0.
- The keyword **ALPHA** corresponds to the parameter α. The default value of **ALPHA** is 0.10.
- The keyword **BETA** corresponds to the parameter β. The default value of **BETA** is 0.10.
- The **INITIAL_STIFFNESS** keyword is used to specify the initial slope of the force versus strain curve. The default **INITIAL_STIFFNESS** is 1.0.

Example of Command Usage

To specify a time-dependent material characterization suitable for polypropylene geogrids with $c_1 = 4.54$, $c_2 = 0.55$, $\alpha = 0.60$, $\beta = 0.16$, and initial stiffness of 120.4, enter the following command for material characterization of Dechasakulsom and Kaliakin [56]:

```
material geosynthetic D_K num 1  desc "PP geogrid idealization" &
c_1  4.54  c_2  0.55  alpha 0.60  beta 0.16  init 120.4
```

33.3 The MATERIAL GEOSYNTHETIC L_T command

Synopsis

The **MATERIAL GEOSYNTHETIC L_T** command is used to describe the time-independent material characterization of Ling and Tatsuoka [82] suitable for geosynthetic reinforcement.

Syntax

The following syntax is used to describe the various forms of the **MATERIAL GEOSYNTHETIC L_T** idealization:

MATerial GEOsynthetic L_T NUMber #
(DEScription "string")
(INItial_stiffness #.#) (ULTimate_force #.#) (FAIlure_ratio #.#)

Explanatory Notes

The **MATERIAL GEOSYNTHETIC L_T** command is commonly used in conjunction with a finite deformation (L2FDP0) line element. The linear specific constitutive law for the material is assumed to be of the form

$$f(t) = \frac{\varepsilon}{A + B\,\varepsilon}$$

where f is the force in the geosynthetic per unit width, and ε is the (assumed uniaxial) strain in the geosynthetic, and A and B are constant model parameters.

- The **NUMBER** keyword is used to specify the (global) number of the material associated with the incompressible isotropic elastic idealization. The default material number is one (1).
- The optional alphanumeric string associated with the **DESCRIPTION** keyword must be enclosed in double quotes ("). It is used solely to describe the material being idealized to the analyst. The **DESCRIPTION** string is printed as part of the "echo" of the material.
- The **INITIAL_STIFFNESS** keyword corresponds to a value that is equal to $1/A$. The default value of **INITIAL_STIFFNESS** is 1.0.

- The **ULTIMATE_FORCE** keyword corresponds to a value that is equal to $1/B$. The default value of **INITIAL_FORCE** is 1.0.
- The **FAILURE_RATIO** keyword corresponds to a value that is equal to the failure ratio R; this parameter is included in the formulation so as to allow for failure of a geosynthetic to occur at a load level that is lower than the ultimate value. The default value of **FAILURE_RATIO** is 0.90.

Example of Command Usage

To specify the time-independent material characterization of Ling and Tatsuoka [82] for the non-woven, heat-bonded polypropylene geosynthetic used by Wu [115], enter the following command:

```
 material geosynthetic L_T num 1  desc "Denver non-woven PP geotextile" &
init 484.5  ultimate  35.43  reduction  0.90
```

33.3.1 Theoretical Considerations: Ling and Tatsuoka [82] Material Characterization

The time-independent material characterization of Ling and Tatsuoka [82] suitable for geosynthetic reinforcement has the following form:

$$f(t) = \frac{\varepsilon}{A + B\,\varepsilon} \tag{33.1}$$

where f is the force in the geosynthetic per unit width, ε is the (assumed uniaxial) strain in the geosynthetic, and A and B are constants. The instantaneous tangent stiffness is computed as follows:

$$J = \frac{df}{d\varepsilon} = \frac{A}{(A + B\varepsilon)^2} \tag{33.2}$$

Noting that, from equation (33.1)

$$\varepsilon = \frac{Af}{1 - Bf} \tag{33.3}$$

it follows that, upon substitution of equation (33.3) into (33.2), that

$$J = \frac{1}{A}(1 - Bf)^2 \tag{33.4}$$

The constants A and B can be given physical meaning by noting that, from equation (33.2), when $\varepsilon = 0$,

$$J \equiv J_i = \frac{1}{A} \tag{33.5}$$

where J_i represents the initial stiffness. In addition, that when $\varepsilon = \infty$,

$$J = 0 \quad ; \quad f \equiv f_{ult} = \frac{1}{B} \quad \Rightarrow \quad B = \frac{1}{f_{ult}} \tag{33.6}$$

where f_{ult} represents the ultimate value of the force per unit width in the geosynthetic. Thus, equation (4) is written in the following more convenient form:

$$J = J_i \left(1 - \frac{f}{f_{ult}}\right)^2 \tag{33.7}$$

In order to allow for failure of a geosynthetic to occur at a load level that is lower than the ultimate, equation (33.7) is slightly modified to read

$$J = J_i \left(1 - \frac{f}{R\, f_{ult}}\right)^2 \tag{33.8}$$

where R is the failure ratio, with $0 < R \leq 1.0$.

33.4 The MATERIAL RAMBERG-OSGOOD command

Synopsis

The **MATERIAL RAMBERG-OSGOOD** command is used to describe the time-independent Ramberg-Osgood [101] material idealization. This idealization has the following form:

$$\varepsilon = \frac{\sigma}{E} + \alpha \frac{\sigma_r}{E} \left(\frac{\sigma}{\sigma_r} \right)^m$$

where ε is the uniaxial strain in the material and σ is the associated stress. The elastic modulus is denoted by E, and α and m are dimensionless model parameters. The latter is an integer (input as a floating-point number) that specifies the strain-hardening characteristics of the material. Finally, σ_r is a reference stress. The first term (σ/E) in the above equation is seen to be the elastic strain; the second term is the plastic strain. The tangent modulus for the material is

$$E_T = \frac{\partial \sigma}{\partial \varepsilon} = \frac{E}{1 + \alpha\, m \left(\frac{\sigma}{\sigma_r} \right)^{m-1}}$$

Syntax

The following syntax is used to describe the various forms of the **MATERIAL RAMBERG-OSGOOD** idealization:

MATerial RAMberg-osgood NUMber #
(DEScription "string")
(MODulus #.#) (ALPha #.#) (M_PARameter #.#) (SIGma_r #.#)

Explanatory Notes

- The **NUMBER** keyword is used to specify the (global) number of the material associated with the incompressible isotropic elastic idealization. The default material number is one (1).
- The optional alphanumeric string associated with the **DESCRIPTION** keyword must be enclosed in double quotes ("). It is used solely to describe the material being idealized to the analyst. The **DESCRIPTION** string is printed as part of the "echo" of the material.

- The **MODULUS** keyword corresponds to the elastic modulus E. The default value of E is 300,000.
- The **ALPHA** keyword corresponds to the elastic modulus α. The default value of α is 1.0.
- The **M_PARAMETER** keyword corresponds to the elastic modulus m. The default value of m is 1.0.
- The **SIGMA_R** keyword corresponds to the elastic modulus σ_r. The default value of σ_r is 1.0.

Remark: If m is very large, then the plastic strain remains small until σ approaches σ_r, and increases rapidly when σ exceeds σ_r. Consequently, σ_r may be regarded as an approximate yield stress.

Remark: In the limit as m approaches infinity, the plastic strain is zero when $\sigma < \sigma_r$, is determinate when $\sigma = \sigma_r$, and is infinite when $\sigma > \sigma_r$) and is thus impossible). This limiting case describes a *perfectly plastic* solid with yield stress of σ_r.

Remark: The Ramberg-Osgood idealization is particularly well-suited to representing the virgin curve of work-hardening solids, especially ones without a sharply defined yield stress [84].

Example of Command Usage

To specify a hypothetical Ramberg-Osgood material idealization, enter the following command:

```
mat ramberg-osgood number 1 &
     desc "arbitrary material parameters" &
     modulus 1.20e+07 alpha 0.4286 m_param 11.0 sigma_r 43500.0
```

33.5 The MATERIAL VISCOELASTIC command

Synopsis

The **MATERIAL VISCOELASTIC** command is used to describe a linear viscoelastic material characterization. It is commonly used in conjunction with a finite deformation (**L2FDP0**) line element. In the past, such elements have been used to model the time-dependent behavior of geogrid reinforcement [98]. The linear viscoelastic constitutive law for the material is assumed to have the following form:

$$\sigma(t) = \int_0^t \phi(t-\tau)\,\frac{\partial \varepsilon}{\partial \tau}\,d\tau$$

where the relaxation function is given by

$$\phi\,(t) = \alpha_0 + \alpha_1 \exp\,(-\beta_1 t)$$

where α_0 and α_1 have units of stress (FL^{-2}), and β_1 has units of t^{-1}. For plane strain idealizations, $\phi\,(t)$ must represent the "effective" relaxation function; i.e., it must include the Poisson stiffening due to the plane strain constraint. For axisymmetric idealizations, it is assumed that the Poisson's effect is negligible.

Syntax

The following syntax is used to describe the various forms of the **MATERIAL VISCOELASTIC** idealization:

MATerial VIScoelastic NUMber #
(DEScription "string")
(ALPHA0 #.#) (ALPHA1 #.#) (BETa1 #.#)

Explanatory Notes

- The **NUMBER** keyword is used to specify the (global) number of the material associated with the incompressible isotropic elastic idealization. The default material number is one (1).
- The optional alphanumeric string associated with the **DESCRIPTION** keyword must be enclosed in double quotes ("). It is used solely to describe the material being idealized to the analyst. The **DESCRIPTION** string is printed as part of the "echo" of the material.

- The **ALPHA0** keyword corresponds to the parameter α_0. The default value of α_0 is 1.0.
- The **ALPHA1** keyword corresponds to the parameter α_1. The default value of α_1 is 0.0.
- The **BETA1** keyword corresponds to the parameter β_1. The default value of β_1 is 0.0.

To gain insight into the parameters α_0, α_1 and β_1, the following observations are noteworthy:

- At $t = 0$, the stress in the one-dimensional element is equal to $\alpha_0 + \alpha_1$.
- The slope of the relaxation curve (i.e., stress or force vs. time) is influenced by changes in β_1. This is illustrated in Figure 33.1, in which "normalized stress" is equal to the current stress divided by the initial stress.
- Larger values of β_1 result in more rapid stress relaxation.
- Larger values of α_1 result in more rapid stress relaxation. This is shown in Figure 33.2.

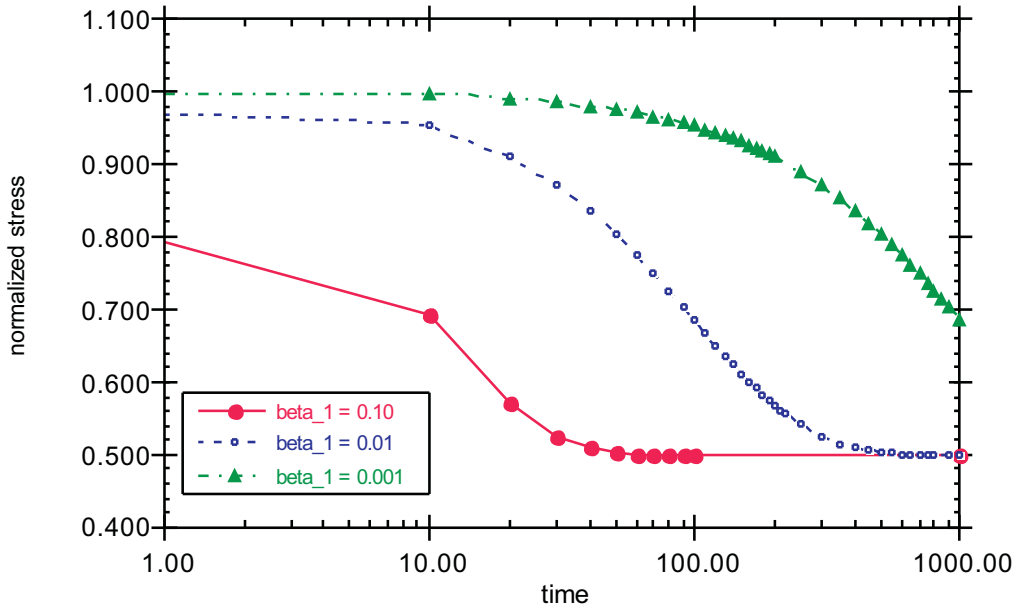

Figure 33.1: Effect of Varying β_1 on the Stress Relaxation Response.

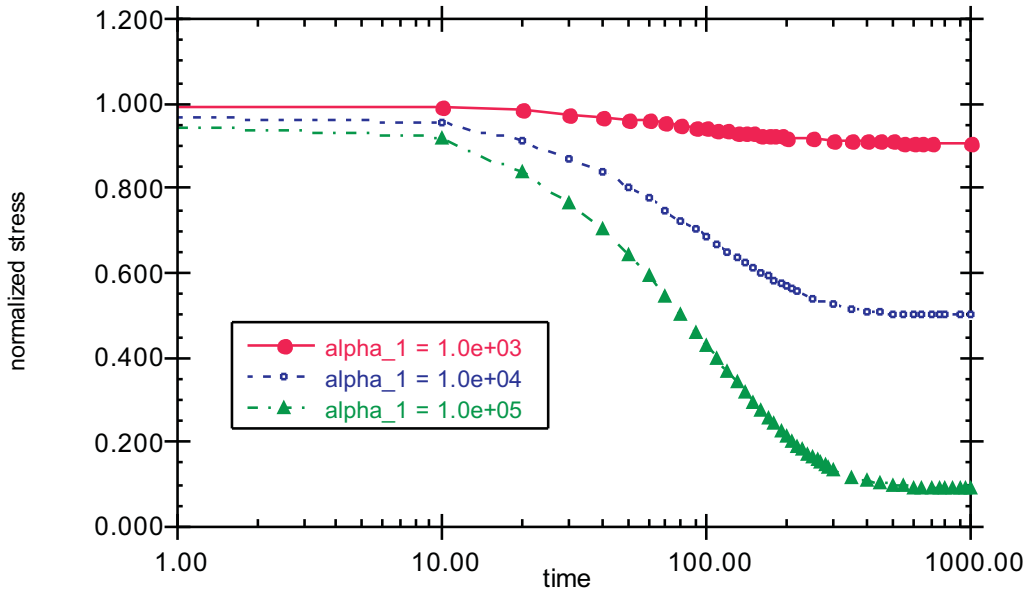

Figure 33.2: Effect of Varying α_1 on the Stress Relaxation Response.

Example of Command Usage

To specify a viscoelastic material idealization with $\alpha_0 = 5.0 \times 10^4$, $\alpha_1 = 1.0 \times 10^4$ and $\beta_1 = 0.10$, enter the following command:

```
mat viscoelastic  number 1 &
     desc "elastic material for bar elements" &
        alpha0  5.0e+04  alpha1  1.0e+04  beta1  1.0e-01
```

Chapter 34

MATERIAL Idealizations Suitable For Interface Elements

This chapter contains information associated with the description of material idealizations suitable for use in conjunction with interface elements that are available for use in the APES$^+$ computer program.

34.1 Available Material Idealizations Suitable For Interface Elements

The following material idealizations, suitable for use in conjunction with interface elements, are currently available in the APES$^+$ computer program:

- "Standard" material idealization for use with zero-thickness interface elements (see Section 34.2).

34.2 The MATERIAL INTERFACE STANDARD command

Synopsis

The **MATERIAL INTERFACE STANDARD** command is used to describe a standard Mohr-Coulomb type friction idealization for zero-thickness interface elements. The criterion for slip along the interface is controlled by the values specified for the friction coefficient (equal to the tangent of the friction angle), and for the cohesion.

This material idealization is used in conjunction with the LK1 and LK2 zero thickness interface elements [57], as well as with the GTB zero thickness interface element [34]. The constitutive relation associated with this material idealization has the following form:

$$\begin{Bmatrix} f_s \\ f_n \end{Bmatrix} = \begin{bmatrix} k_s & k_{sn} \\ k_{sn} & k_n \end{bmatrix} \begin{Bmatrix} \delta_s \\ \delta_n \end{Bmatrix}$$

where k_s, k_{sn}, and k_n represent the tangential, coupling and normal stiffnesses (per unit length), respectively, f_s and f_n represent the tangential and normal force per unit length along the interface, respectively, and δ_s and δ_n represent the tangential and normal relative displacement, respectively, along and across the interface. A Mohr-Coulomb idealization of frictional resistance is assumed.

Syntax

The following syntax is used to describe the various forms of the **MATERIAL INTERFACE STANDARD** idealization:

MATerial INTerface STAndard NUMber ##
(DEScription "string")
(TANgential #.#) (NORmal #.#) (COUpling #.#)
(RESidual #.#) (FRIction #.#) (COHesion #.#)

Explanatory Notes

- The **NUMBER** keyword is used to specify the (global) number of the material associated with the incompressible isotropic elastic idealization. The default material number is one (1).

- The optional alphanumeric string associated with the **DESCRIPTION** keyword must be enclosed in double quotes ("). It is used solely to describe the material being idealized to the analyst. The **DESCRIPTION** string is printed as part of the "echo" of the material.
- The **FRICTION** keyword is used to specify the value of the friction coefficient, which is equal to the tangent of the friction angle. The default value for the friction coefficient is 0.50.
- The **COHESION** keyword is used to specify the value of the cohesion associated with a Mohr-Coulomb idealization of frictional resistance. The default value for the cohesion is 0.0.
- The **TANGENTIAL** keyword is used to specify the value of the tangential spring stiffness associated with a Mohr-Coulomb idealization of frictional resistance. The default value for the tangential stiffness is 1.0×10^6.
- The **NORMAL** keyword is used to specify the value of the normal spring stiffness. Material penetration along the interface is minimized by specifying *large* values for the **NORMAL** spring stiffness. The default value for the normal stiffness is 1.0×10^6.
- The **RESIDUAL** keyword is used to specify the value of the residual spring stiffness that is active once relative displacement ("slip") has occurred. The default value for the tangential stiffness is 1.0.

Example of Command Usage

The following command is used to specify a material idealization for a zero-thickness interface element:

```
mat interface  standard  number 3  &
   desc "lag bolt simulation @ concrete-wood interface"  &
       tangential  5.0e+05  normal  1.0e+08   &
       friction    0.50     cohesion  250.0
```

Chapter 35

The NODES Commands

This chapter contains information associated with the specification of nodal coordinates for finite element analyses using the APES$^+$ computer program. It consists of the following sections:

- General overview of **NODES** commands (see Section 35.1).
- Generation of nodal coordinates along a circular arc: the **NODES CIRCLE** command (see Section 35.2).
- Reading nodal numbers and coordinates from an *ASCII* file: the **NODES FROM** command (see Section 35.3).
- Specification nodal coordinates along a straight line in the plane or in space: the **NODES LINE** command (see Section 35.4).

35.1 NODES commands: An Overview

Synopsis

The following comments refer to the specification of nodal coordinates used in the mathematical model slated for analysis via APES+. It is anticipated that such models have been developed using an efficient mesh generation pre-processor. However, in the instance that such a program is not available to the analyst, commands are included that allow the analyst to describe/generate all of the nodal coordinates with the APES+ pre-processor.

APES+ provides several options that facilitate the creation of valid mathematical models. In particular, node number generation and coordinate determination schemes are available for use by the analyst. These schemes must be used in accordance with the hierarchy associated with the specification of node and element information. Specifically, the locations of the nodes lying along all edges of the body must be specified. Typically this is realized using the straight line and/or circular arc generation schemes described below in conjunction with the **NODES CIRCLE** and **NODES LINE** commands. Once the edge node locations have been determined, the surface(s) bounded by these nodes must be generated; this is realized by specifying the **GENERATE SURFACES** command. These surfaces are collections of two-dimensional quadrilateral element "patches". As such, the determination of the coordinates of nodes in the interior of the surface(s) is typically realized using a two-dimensional scheme [49]. If a two-dimensional analysis is being performed, then following the determination of all surface nodal coordinates, the mathematical model is complete with respect to the node and element determination. If, on the other hand, a three-dimensional analysis is being performed, then following the definition of all of the surface nodes, an extension of the two-dimensional generation scheme is used to determine the coordinates of the nodes interior to the volume enclosed by the defined surfaces. Special care must be taken to ensure that *all* edge nodes and *all* surface nodes are defined prior to the volume generation.

In a given mathematical model, the coordinates of each node point must be defined, but need not be input in any particular order. When numbering the nodes, not all numbers between 1 and the highest node number need correspond to actual nodes in the body.

If the location of a node is prescribed more than once in the input and the coordinates are not in agreement, the last description is used. If, however, in a second (or subsequent description) the node number is entered as negative, then the previous location associated with this node is assumed. The utility of this option is illustrated in conjunction with the **NODES LINE** command.

To assist the analyst in defining these coordinates, the APES+ computer program offers several data generation schemes. Using the **NODES LINE** command, nodal coordinates can be generated along a straight line (this is achieved using linear interpolation); this line can as well be mapped to a curve (this is achieved using quadratic interpolation). Using the **NODES CIRCLE** command, nodes can also be generated along a circular arc. The

NODES FROM command allows a complete set of nodal coordinates to be read from an *ASCII* file. Finally, a general scheme is available to compute nodal coordinates within part or all of the interior of a mesh. The use of these schemes can, for example, enable one to describe the nodal coordinates of an arbitrarily large mesh with as few as five input records.

35.2 The NODES CIRCLE Command

Synopsis

The **NODES CIRCLE** command is included in the APES+ computer program to facilitate the generation of nodal coordinates along a circular arc in the plane or in space.

Syntax

The following syntax is associated with the **NODES CIRCLE** command:

NODes CIRcle NUMber ## X1 #.# X2 #.# (X3 #.#)
INCrement ## RATio #.#
(X1_intermediate #.#) (X2_intermediate #.#) (X3_intermediate #.#)

Explanatory Notes

The **NUMBER** keyword denotes the node number at which the coordinate generation will stop (the beginning point for the generation is the node **NUMBER** specified on the previous **NODES CIRCLE** or **NODES LINE** record). The keyword **INCREMENT** is used to specify the numbering increment to be used in generating the nodal coordinates. The default numbering increment is equal to zero (0). The spacing of the generated nodes is dictated by the real number supplied in conjunction with the **RATIO** keyword. The default spacing ratio is equal to 1.0. The values for **X1_INTERMEDIATE**, **X2_INTERMEDIATE** and **X3_INTERMEDIATE** represent the coordinates of an intermediate point through which the circular arc passes. This point need not be a node associated with the mathematical model. The default coordinates for **X1**, **X2** and **X3** are zero (0.0). As such, if a specific coordinate is equal to zero, the analyst need not supply the value explicitly.

Example of Command Usage

To better illustrate the use of the **NODES CIRCLE** command, consider the generation of nodes along a circular arc the begins at node 2 and ends at node 44 (Figure 35.1). The numbering increment is equal to 7, and a spacing **RATIO** of 1.0 is adopted. The coordinates of the intermediate point are (4.0, 4.745). The two command lines required to generate these nodes are thus

```
nodes line num   2  x1  6.0  x2 -4.0
```

```
nodes circle num 44  x1 -4.0 x2  6.0  incr  7   &
  x1_int 4.0  x2_int  4.745
```

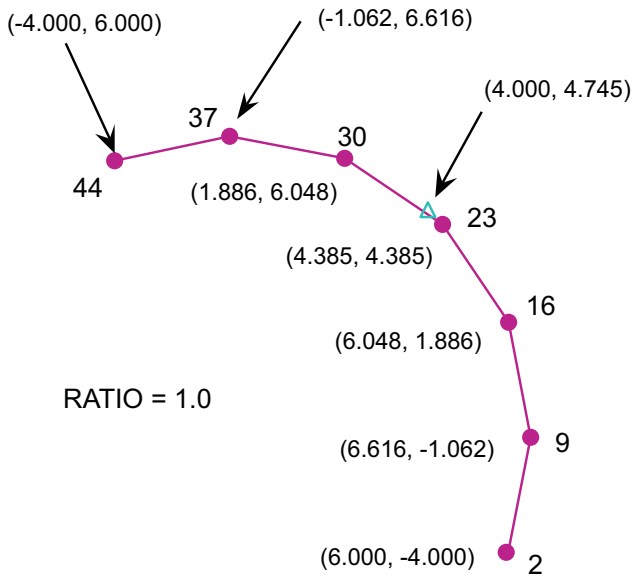

Figure 35.1: Example of nodal generation along circular arc (spacing ratio = 1.0).

The above example of nodal generation is repeated, only assuming a spacing ratio of 0.80. The resulting arc of nodes is shown in Figure 35.2.

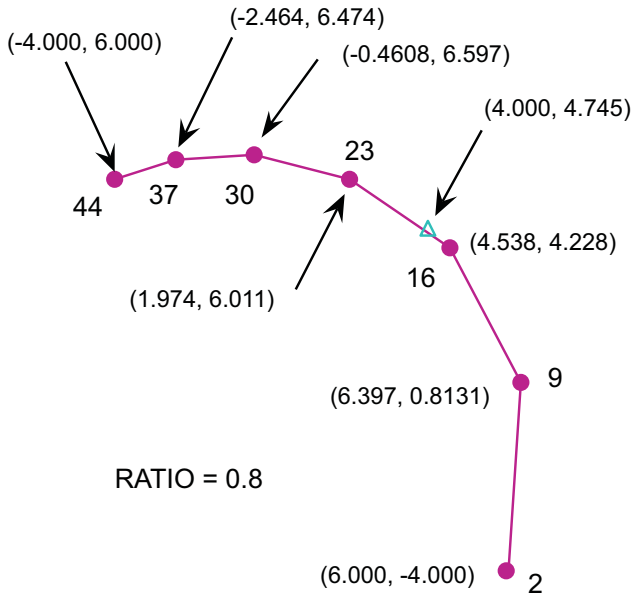

Figure 35.2: Example of nodal generation along circular arc (spacing ratio = 0.80).

35.3 The NODES FROM Command

Synopsis

The **NODES FROM** command is used to read nodal information from the *ASCII* file "string."

Syntax

The following syntax is associated with the **NODES FROM** command:

NODes FROm FILename "string"

Explanatory Notes

When using this command, the analyst must ensure that all nodal coordinates are explicitly given in this file. Since no generation and/or default values are applicable to the **NODES FROM** command, the nodal information contained in file "string" must be complete. The following list directed (free-format) is expected by the APES+ computer program:

- For two-dimensional analyses:

```
n1  x1(n1)  x2(n1)
```

- For three-dimensional analyses:

```
n1  x1(n1)  x2(n1)  x3(n1)
```

Although the **NODES FROM** command can be used in conjunction with the **NODES LINE** and **NODES CIRCLE** commands, it is most commonly used when nodal data is generated using a stand-alone pre-processor. In such a case all the nodal coordinates would be contained in a data file; i.e., the output from the pre-processor.

Example of Command Usage

In order to read all nodal coordinates from the file z_node.dat, enter either of the following command:

```
nodes  from  filename  ‘‘z_node.dat’’

nod from fil ‘‘z_node.dat’’
```

The file z_node.dat, contains the following nodal data

```
 1      0.0     0.0
 2      0.5     0.0
 3      1.1     0.0
 4      1.6     0.0
 5      2.0     0.0
 6      0.0     0.35
 7      1.0     0.6
 8      2.0     0.4
 9      0.0     0.8
10      0.45    1.0
11      0.9     1.2
12      1.4     1.0
13      2.0     0.85
14      0.0     1.2
15      1.1     1.5
16      2.0     1.2
17      0.0     2.0
18      0.6     2.0
19      1.2     2.0
20      1.6     2.0
21      2.0     2.0
```

35.4 The NODES LINE Command

Synopsis

The **NODES LINE** command is used to specify nodes along a straight line in the plane or in space.

Syntax

The following syntax is associated with the **NODES LINE** command:

NODes LINe NUMber ## X1 #.# X2 #.# (X3 #.#)
INCrement ## RATio #.#
(X1_extra #.#) (X2_extra #.#) (X3_extra #.#)

Explanatory Notes

By specifying values for **X1_EXTRA**, **X2_EXTRA** and **X3_EXTRA**, the straight line is mapped to a curve by means of a quadratic interpolation. The default **X1**, **X2** and **X3** coordinates are zero (0.0). As such if a specific coordinate is equal to zero, the analyst need not supply the value explicitly. The default **INCREMENT** value is zero (0), and the default **RATIO** is one (1.0). The default **X1_EXTRA**, **X2_EXTRA** and **X3_EXTRA** coordinates are zero (0.0).

If the location of a node is prescribed more than once in the input and the coordinates are not in agreement, the last description is used. If, however, in a second (or subsequent description) the node number is entered as *negative*, then the previous location associated with this node is assumed.

Whenever several sequential node points lie along a straight line or along a curve, the coordinate generation option associated with the **NODES LINE** command, with either linear or quadratic interpolation may be used. If such a situation exists, it is only necessary to enter the coordinates of the initial and final points of the sequence (denoted in this discussion by N and N', respectively), and the values of **INCREMENT**, **RATIO** and possibly **X1_EXTRA**, **X2_EXTRA** and **X3_EXTRA**. The constant **INCREMENT** represents the difference between any two successive node numbers in the sequence and **RATIO** defines the ratio of the distances between any two adjacent pairs of node points. The values specified in conjunction with the **X1_EXTRA**, **X2_EXTRA** and **X3_EXTRA** keywords determine whether linear or quadratic interpolation will be used.

35.4.1 Linear Interpolation

This generation option is realized if neither of the keywords **X1_EXTRA**, **X2_EXTRA** and **X3_EXTRA** are specified in a NODES LINE command. If, for the input record describing node N', the value specified in conjunction with the **INCREMENT** keyword is nonzero, then intermediate node points will be generated along a straight line between node N' and the point N described in the preceding node specification record. That is, the coordinates of the nodes (N + **INCREMENT**), (N + 2*__INCREMENT__ , . . . , N' - **INCREMENT**) will each be automatically generated (see Figure 35.3).

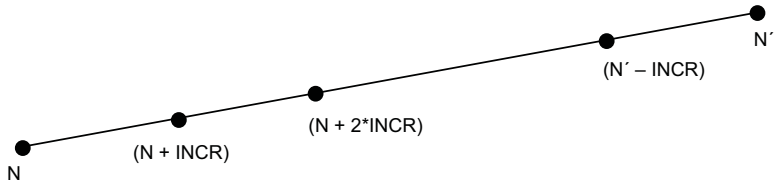

Figure 35.3: Nodal generation using linear interpolation.

The node number N', for which the specified non-zero value of **INCREMENT** triggered the generation of the line $N \rightarrow N'$, can also serve as the initial point of a line generated between it and the point described in the following record. The exterior nodes shown in Figure 35.4 can thus be generated with the following five records:

```
nodes line number  1    x1  1.25  x2  0.20
nodes line number  6    x1  7.50  x2  0.20  incr  1
nodes line number 42    x1  7.50  x2  8.10  incr  6
nodes line number 37    x1  1.25  x2  8.10  incr -1
nodes line number -1                        incr -6
```

Since in the second specification of node 1 the node number is entered as *negative*, the coordinates need not be repeated – the previous location of (1.25, 0.20) is used.

The end points of a line sequence may be entered in *either order*. For example, the segments illustrated in Figure 35.5 can be defined by specifying the nodes in either the order from 7 to 22; viz.,

```
node line number  7  x1 16.5  x2  0.50
node line number 22  x1  7.2  x2 10.8  incr 5  ratio 2.0
```

or in the order from 22 to 7; viz.,

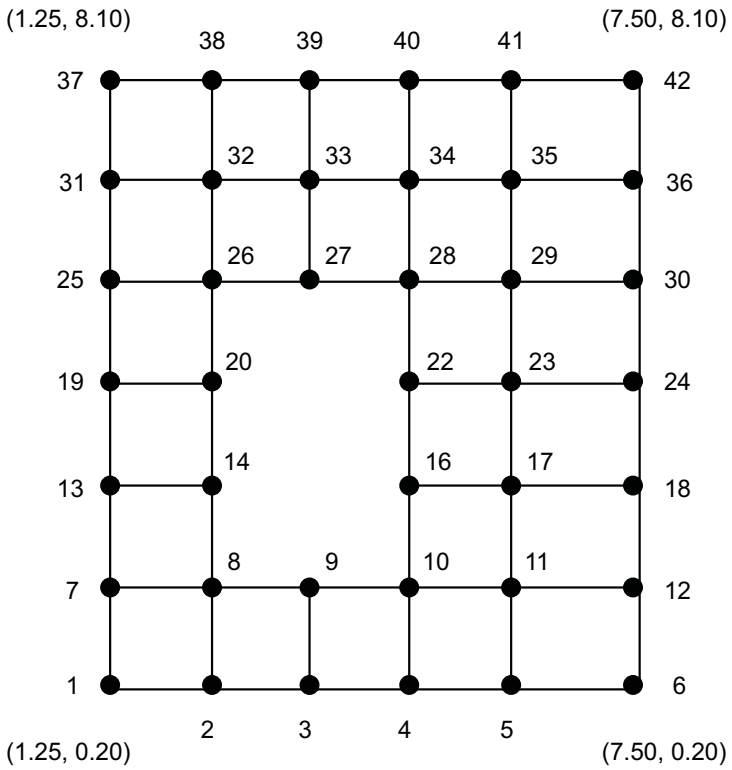

Figure 35.4: Hypothetical finite element mesh.

```
node line number 22  x1  7.2  x2  0.8
node line number  7   x1 16.5  x2  0.50  incr -5  ratio 0.50
```

The spacing of the intermediate points (e.g., nodes 12 and 17 in the above example) is controlled by the spacing **RATIO**. A value of 1.0 associated with the **RATIO** keyword results in *equally spaced* nodes. In selecting a spacing **RATIO** value, the following three formulas are of use (in the sequel the value specified in conjunction with the **RATIO** keyword is denoted by r, and the number of segments comprising a sequence of nodes is denoted by m):

1. If the total length of a line of nodes is equal to L and if $r \neq 1.0$, then the length of the first segment will equal

$$\frac{L(1-r)}{1-r^m}$$

   To better illustrate the use of this equation, consider the example shown in Figure 35.5. Instead of the nodal spacing shown, assume that the first segment (from node 7 to 12) is desired to be one quarter of the overall length L. As such, it follows that since $m = 3$ that

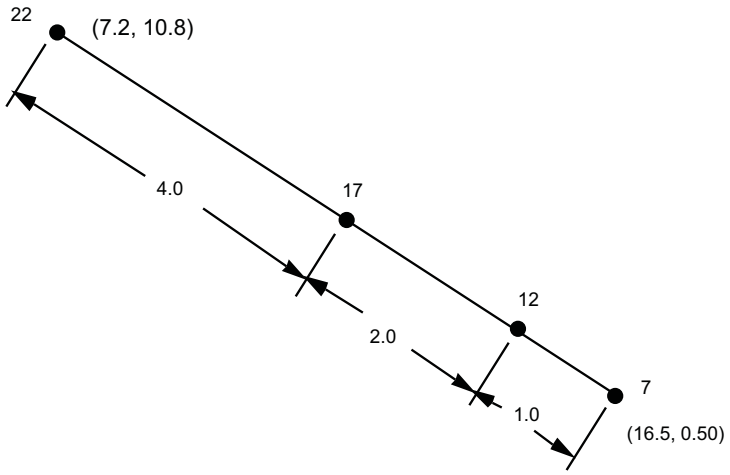

Figure 35.5: Example of linear nodal coordinate generation.

$$\frac{L(1-r)}{1-r^3} = \frac{L}{4} \;\Rightarrow\; \frac{(1-r)}{1-r^3} = \frac{1}{4} \;\Rightarrow\; r^3 - 4r + 3 = 0$$

Recalling that $r \neq 1.0$, the only practical root of this cubic is found to be $r = RATIO = 1.303$.

2. If the ratio of the lengths of the *last* and the *first* segments must equal a specified value D (i.e.; $D = \Delta L_m / \Delta L_1$), then specify a value of r equal to the quantity D^q, where $q = (m-1)^{-1}$, with $m > 1$. For the example shown above, if the ratio of the last segment to the first is desired to be equal to 1.50, then since $q = (m-1)^{-1} = (3-1)^{-1} = 0.50$, it follows that $r = RATIO = (1.50)^{0.50} = 1.225$.

3. Suppose a line of nodes has been generated with a particular value of **RATIO** and that this line is to now be regenerated with *twice* as many segments (i.e.; with twice as many node points minus one). Furthermore suppose that it is desired to have the locations of the first set of nodes be retained in the second set. To achieve the above-mentioned goals use $\sqrt{r}$ as the new spacing ratio.

35.4.2 Quadratic Interpolation

In this option quadratic isoparametric generation is employed, and facilitates the placement of node points along curved lines. The option is realized by specifying either of the keywords **X1_EXTRA**, **X2_EXTRA**, or **X3_EXTRA**. The values associated with these keywords represent the $x1$-, $x2$- and $x3$-coordinates of an intermediate point through which the line of nodes will pass. In general, this intermediate point does not coincide with any node point.

For the case of quadratic interpolation, the location of a node point is determined from the following mapping (refer to Figure 35.6 shown below which, for simplicity, is restricted only to two-dimensional space):

$$x_1(\xi_p) = \frac{1}{2}\xi_p\,(\xi_p - 1)\,x_{1N} + \left(1 - \xi_p^2\right) x_{1e} + \frac{1}{2}\xi_p\,(\xi_p + 1)\,x_{1N'}$$

$$x_2(\eta_p) = \frac{1}{2}\eta_p\,(\eta_p - 1)\,x_{2N} + \left(1 - \eta_p^2\right) x_{2e} + \frac{1}{2}\eta_p\,(\eta_p + 1)\,x_{2N'}$$

$$x_3(\zeta_p) = \frac{1}{2}\zeta_p\,(\zeta_p - 1)\,x_{3N} + \left(1 - \zeta_p^2\right) x_{3e} + \frac{1}{2}\zeta_p\,(\zeta_p + 1)\,x_{3N'}$$

where

- (x_{1N}, x_{2N}, x_{3N}) = coordinates of the initial node in the sequence,
- (x_{1e}, x_{2e}, x_{3e}) = coordinates of the intermediate point (not necessarily a node) in the sequence,
- $(x_{1N'}, x_{2N'}, x_{3N'})$ = coordinates of the final node in the sequence.
- ξ and η are local (natural) coordinates ($-1 \le \xi,\, \eta \le 1$) defined by

$$\xi \equiv \frac{x_1 - 0.5(x_{1N'} + x_{1N})}{0.5(x_{1N'} - x_{1N})}\;;\; \eta \equiv \frac{x_2 - 0.5(x_{2N'} + x_{2N})}{0.5(x_{2N'} - x_{2N})}$$

- (ξ_p, η_p) = coordinates of node "p" in (ξ, η) space.

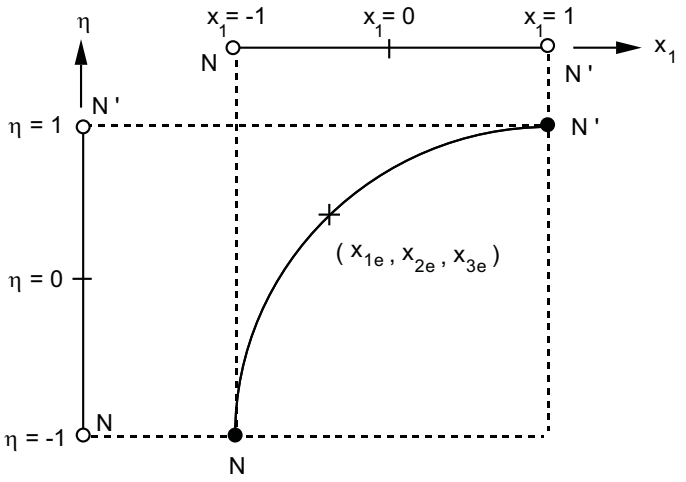

Figure 35.6: Nodal Coordinate generation using quadratic interpolation.

The description of the keywords **INCR** and **RATIO** given in conjunction with the description of linear interpolation applies directly to the quadratic interpolation, only with respect to (ξ, η, ζ) space.

The following example serves to better compare and contrast the linear and quadratic interpolation options. Consider the generation from node 5 (2.50, 0.80, 0.0) to node 25 (22.50,

13.14, 0.0). Using linear interpolation with spacing ratios of 1.0 and 0.75, the coordinates shown in Figures 35.7a and b, respectively, are generated.

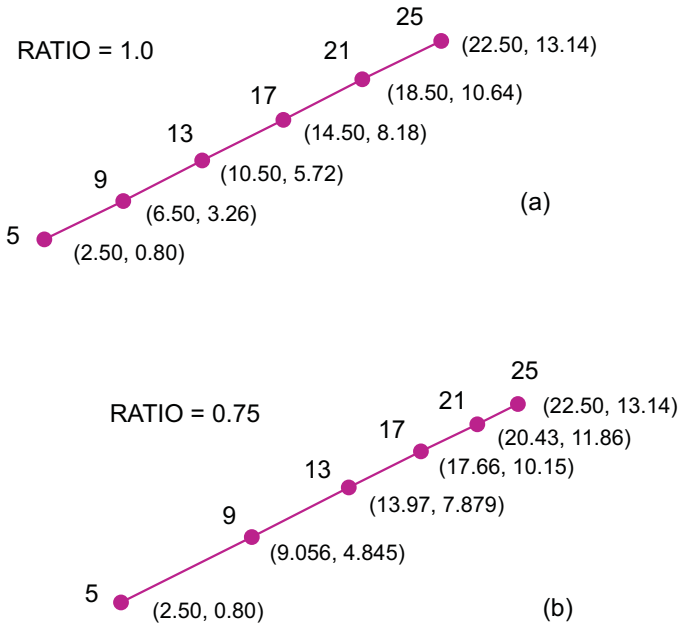

Figure 35.7: Example of nodal coordinate generation using linear interpolation: a) RATIO = 1.0, b) RATIO = 0.75.

Quadratic interpolation is next employed, with the "extra" (intermediate) point chosen to have the coordinates (16.0, 13.5, 0.0). The nodal coordinates generated using spacing ratios of 1.0 and 0.75 are shown in Figures 35.8a and b, respectively.

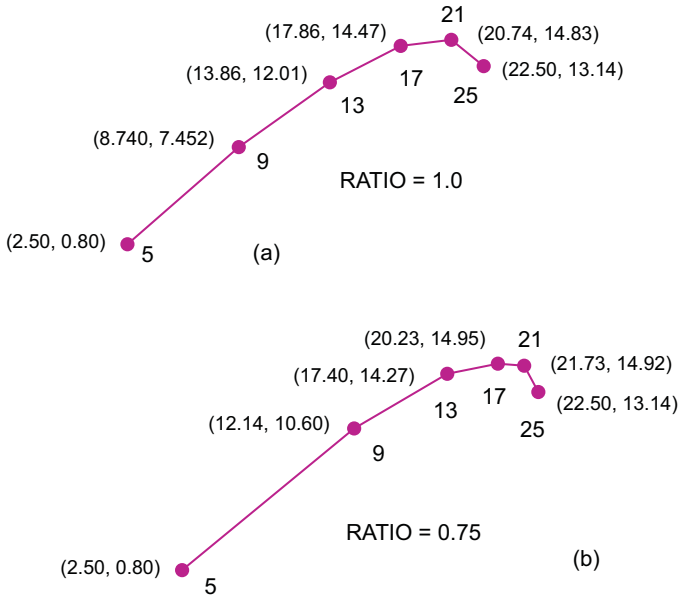

Figure 35.8: Example of nodal coordinate generation using quadratic interpolation: a) RATIO = 1.0, b) RATIO = 0.75.

Example of Command Usage

The simplest application of the **NODES LINE** command is to specify the location of a single node. For example, if the coordinates of node 536 are (12.83, -5.283, 0.54), enter the following command:

```
nodes  line  number 536  x1 12.83  x2 -5.283  x3  0.54
```

Using the following set of commands, the *boundary* of the nine-node region shown in Figure 35.9 is described.

```
nod line number  1
nod line number  3  x1 10.0           inc  1
nod line number  9  x1 10.0  x2 10.0  inc  3
nod line number  7           x2 10.0  inc -1
nod line number -1                    inc -3
```

Note that advantage has been taken of the fact that the default coordinate values are equal to zero (0.0).

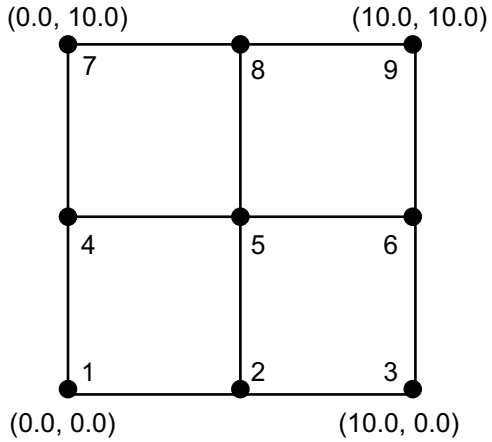

Figure 35.9: Sample nine-node finite element mesh.

Chapter 36

The SPECIFICATION Commands

Since it has not been modified to account for constraints imposed at boundary and possibly interior nodes, following assembly, the system property matrix **K** is singular at this stage of the analysis. In order to specialize the problem, the nodal constraints must next be considered [52].

In finite element analyses, constraints on the *nodal* values of the primary dependent variables or their derivatives are required in order to account for, or approximate, the actual constraints imposed on the body being analyzed. It is only through such constraints that a problem can be uniquely posed. Since the values of the primary dependent variables, or their derivatives, can be specified at *any node* in the mesh, the previous notion of "boundary conditions" must be generalized to *nodal specifications*.

The **SPECIFICATION** commands available in the APES$^+$ computer program consist of the following groups of commands:

- The **SPECIFICATION CONCENTRATED** commands (see Section 36.1).
- The **SPECIFICATION LINE LINEAR** commands (see Section 36.2).
- The **SPECIFICATION LINE QUADRATIC** commands (see Section 36.3).

36.1 The SPECIFICATION CONCENTRATED Commands

The information supplied using **SPECIFICATION CONCENTRATED** commands allows the user to specify known values of the primary dependent variable(s) (e.g., displacement, pore pressure, temperature, etc.) and secondary dependent variables (e.g., forces, flows, heat flux, etc.) at boundary of interior nodes. Associated with each primary of secondary dependent variable is a suitable history function (refer to the **FUNCTION** commands, which are discussed in Chapter 12).

If several nodes in a sequence all have identical specifications associated with them, then these specifications can all be generated with a single input record. This is achieved by supplying an appropriate range of nodes in conjunction with the **NODES** keyword that is associated with a **SPECIFICATION CONCENTRATED** command.

The following information **SPECIFICATION CONCENTRATED** commands are available in APES+:

- The **SPECIFICATION CONCENTRATED FLOW** command (see Section 36.1.1).
- The **SPECIFICATION CONCENTRATED MECHANICAL** command (see Section 36.1.2).
- The **SPECIFICATION CONCENTRATED ROTATIONAL** command (see Section 36.1.3).
- The **SPECIFICATION CONCENTRATED SCALAR** command (see Section 36.1.4).

36.1.1 The SPECIFICATION CONCENTRATED FLOW Command

Synopsis

The **SPECIFICATION CONCENTRATED FLOW** command is used to describe concentrated nodal hydraulic specifications (i.e., a known value of flow or pore fluid pressure) associated with a coupled flow-mechanical analysis (refer to the **ANALYSIS TYPE** command, which is discussed in Section 5.9).

Syntax

The following syntax is associated with the **SPECIFICATION CONCENTRATED FLOW** command:

SPECIFICAtion CONcentrated MEChanical NODes #:#:# [HIStory ##]
[FLOw | PREssure] [VALue #.#]

Explanatory Notes

A flow has units of L^3t^{-1}; pressures have units of FL^{-2}.

The default flow specification at a node is a **FLOW** equal to a **VALUE** of zero (0.0), with an associated history function equal to -2 (refer to the **FUNCTION** commands, which are discussed in Chapter 12).

Example of Command Usage

In either a two- or three-dimensional analysis, it is desired to specify conditions of zero flow at nodes 1 to 10. In addition, zero pressure is to be specified at nodes 11 and 12. Since the default specification type (i.e., flow), history (equal to -2), and value (equal to zero) are applicable, the following commands are thus sufficient:

```
specification conc  flow nodes 1:10:1

specification concentrated flow nodes 11:12  pressure
```

36.1.2 The SPECIFICATION CONCENTRATED MECHANICAL Command

Synopsis

The **SPECIFICATION CONCENTRATED MECHANICAL** command is used to specify known values of displacements, forces and translational springs at boundary and interior nodes.

Syntax

The following syntax is associated with the **SPECIFICATION CONCENTRATED MECHANICAL** command:

SPECIFICAtion CONcentrated MEChanical NODes #:#:#
[X1_Angle #.#] [X2_Angle #.#] [X3_Angle #.#]
[1_Force | 1_Displacement | 1_Spring] [1_History ##] [1_Value #.#]
[2_Force | 2_Displacement | 2_Spring] [2_History ##] [2_Value #.#]
[3_Force | 3_Displacement | 3_Spring] [3_History ##] [3_Value #.#]

Explanatory Notes

The **SPECIFICATION CONCENTRATED MECHANICAL** command is used to describe nodal specifications (i.e., forces, displacements, and translational spring stiffnesses) associated with a mechanical analysis (refer to the **ANALYSIS TYPE** command, which is discussed in Section 5.9). The translational spring stiffness has units of FL^{-1}.

In each coordinate direction, the default mechanical specification at a node is a **FORCE** equal to a **VALUE** of zero (0.0), with an associated **HISTORY** function equal to -2 (refer to the **FUNCTION** commands, which are discussed in Chapter 12). Thus, at nodes where the displacements in the x_1, x_2, and x_3 coordinate directions are *unknown* and no force is applied, there is no need to supply concentrated nodal specification records for such nodes, as these values are pre-set upon initiation of the program.

Known values of displacement, acting in the global x_1, x_2, and x_3 coordinate directions can be specified at any number of locations in a body by placing nodes at these locations. Displacements acting in the *positive* x_1, x_2, and x_3 coordinate directions are *positive*; those acting in the *negative* coordinate directions are *negative*.

Similarly, concentrated nodal forces, acting in the global x_1, x_2, and x_3 coordinate directions can be specified at any number of locations in a body by placing nodes at these locations. Concentrated nodal forces acting in the *positive* x_1, x_2, and x_3 coordinate directions are *positive*; those forces acting in the *negative* coordinate directions are *negative*.

If two or more contradictory displacement specifications are made in a given global coordinate direction at the same node, the *first* of these is arbitrarily assumed to be correct. If two or more nodal forces are specified in a given global coordinate direction at the same node, their *sum* is used. Finally, if both a displacement and a force are specified in a given global coordinate direction at the same node, the force (since it has no effect on the solution) is *ignored*.

Example of Command Usage

In a two-dimensional analysis, it is desired to describe a roller support (i.e., zero force specified in the x_1-coordinate direction; zero displacement in the x_2-direction) at node 157. At nodes 913 to 941, the support is a pin (i.e., zero displacements specified in the x_1 and x_2-coordinate directions). Since the default history (equal to -2) and force/displacement values (equal to 0.0) are applicable, the following commands are sufficient to describe the aforementioned specifications:

```
specification conc  mech nodes 157  2_displacement

specification concentrated mech nodes 913:941  1_disp 2_disp
```

It is also desired to specify a concentrated force at node 654 having a components equal to 4.50 in the positive x_1 coordinate direction and 20.5 in the negative x_2 coordinate direction, as well as a history function number 1. The following command thus describes the aforementioned specifications:

```
specification conc  mech nodes 654  1_force 1_value 4.50 1_hist 1  &
                             2_force 2_value -20.5  2_hist 1
```

At interior nodes, as well as at ones located along a boundary whose outward normal lies parallel to one of the coordinate directions, the applied concentrated forces, displacements or linear translational springs are specified in terms of the global x_1, x_2, and x_3 coordinates. Alternately, if nodal forces, displacements or linear translational springs are not oriented along one of these coordinate directions, then these quantities can be specified in terms of local (rotated) x'_1, x'_2, and x'_3 coordinates. Figure 36.1 shows an example of a rotated coordinate system in two-dimensions.

The angles **X1_Angle**, **X2_Angle**, and **X3_Angle** represent rotation angles (measured in degrees), that define this rotated coordinate system. A positive rotation is assumed to act

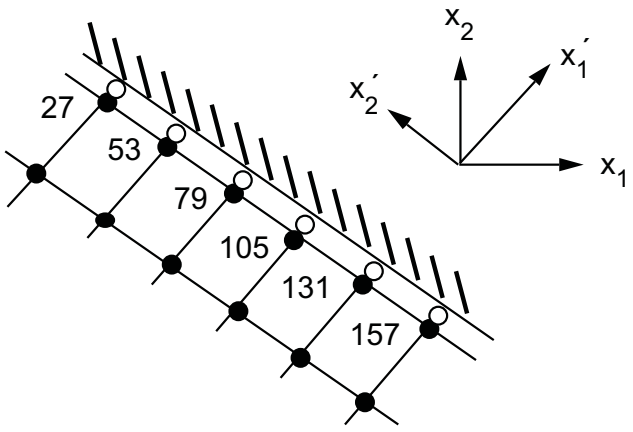

Figure 36.1: Boundary with outward normal not coincident with global coordinate directions.

in a right-handed sense with respect to the axis of rotation. The assumed order of rotation is: rotation about the global x_1-axis, followed by rotation about the global x_2-axis, followed by about the global x_3-axis. These rotations are *not commutative*; they must thus be specified in the aforementioned order. For two-dimensional analyses, only the **X3_Angle** command applies; in this case, counterclockwise angles (in degrees), measured from the global x_1-axis, are *positive*.

Example of Command Usage

Assuming $\theta_3 = 48°$, the nodal specifications shown in Figure 36.1 could be generated by the following record:

```
specification conc  mech nodes 27:157:26 x3_angle 48.0 1_disp
```

where the history function number -2 (i.e., the default case, which is built into the APES+ computer program) is assumed, and full use is made of the other default quantities. The same result could likewise have been realized by using the following record:

```
specification conc  mech nodes 157:27:-26 x3_angle 48.0 1_disp
```

36.1.2.1 Extended Example: Insight into Translational Spring Nodal Specifications for BE2_2D elements

In order to gain insight into the effect of translational spring specifications on the response of C^1 beam elements, consider the cantilever beam shown in Figure 36.2. Assume $E = 3.0\,\text{x}\,10^5$, $I = 0.010$ and $Q = 100$.

From Bernoulli-Euler beam theory, the transverse displacement at the mid-span of a cantilever beam is

Table 36.1: Cantilever Beam with Translational Spring Nodal Specification

k^t	Mid-span displacement	Displacement at $x = L$	Rotation at $x = L$
0.000e+00	1.389e+00	3.472e+00	4.167e-01
1.000e+02	3.934e-01	2.867e-01	-6.116e-02
1.000e+04	3.048e-01	3.122e-03	-1.037e-01
1.000e+06	3.038e-01	3.125e-05	-1.042e-01
1.000e+08	3.038e-01	3.125e-07	-1.042e-01
1.000e+10	3.038e-01	3.125e-09	-1.042e-01

$$\delta_{L/2} = \frac{QL^3}{24EI} = 1.389$$

Similarly, for a *propped* cantilever (i.e., one with a roller support at the right end)

$$\delta_{L/2} = \frac{7QL^3}{768EI} = 0.3038$$

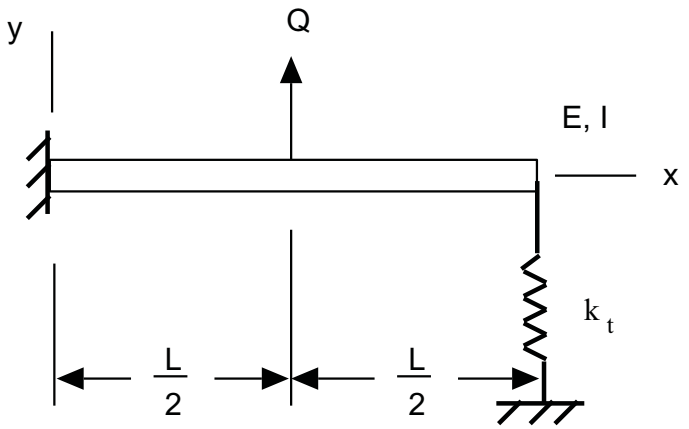

Figure 36.2: Cantilever beam with translational spring nodal specification.

Modeling this beam using a two-element mesh of C^1 elements, analyses were performed for increasing values of translational spring stiffness k^t (i.e., 2_Value). The results are summarized in Table 36.1.

It is evident that as k^t is increased, the transverse displacement at mid-span ($x = L/2$) decreases from that associated with a cantilever beam to the value associated with a propped cantilever. In addition, the displacement at $x = L$ decreases from the free condition to essentially zero. Finally, as k^t is increased the rotation (θ) at $x = L$ not only changes magnitude, but also sign. It is interesting to note that the transverse displacement at mid-span essentially does not change once k^t exceeds 1.000×10^4.

36.1.3 The SPECIFICATION CONCENTRATED ROTATIONAL Command

Synopsis

The **SPECIFICATION CONCENTRATED ROTATIONAL** command is used to specify rotational nodal specifications (i.e., moments, rotations and linear rotational spring stiffnesses) at boundary and interior nodes.

Syntax

The following syntax is associated with the **SPECIFICATION CONCENTRATED ROTATIONAL** command:

SPECIFICAtion CONcentrated ROTational NODes #:#:#
[11_Moment | 11_Rotation | 11_Spring] [11_History ##] [11_Value #.#]
[22_Moment | 22_Rotation | 22_Spring] [22_History ##] [22_Value #.#]
[33_Moment | 33_Rotation | 33_Spring] [33_History ##] [33_Value #.#]

Explanatory Notes

The **SPECIFICATION CONCENTRATED ROTATIONAL** command is used to specify rotational nodal specifications (i.e., moments, rotations and linear rotational spring stiffnesses) associated with a mechanical analysis. Rotational degrees of freedom only apply to beam and frame elements.

For rotation about each of the coordinate directions, the default rotational specification at a node is a **MOMENT** equal to a **VALUE** of zero (0.0), with an associated **HISTORY** function equal to -2 (refer to the **FUNCTION** commands, which are discussed in Chapter 12). In two-dimensional analyses, only the moments, rotations and linear rotational spring stiffness values associated with the x_3-axis are applicable.

If two or more contradictory rotational specifications are made for the same node, the *first* of these is arbitrarily assumed to be correct. If two or more concentrated moments are specified at the same node, their *sum* is used. If both a rotation and a moment are specified for a node, the moment value (since it has no effect on the solution) is *ignored*.

Example of Command Usage

In a three-dimensional analysis, it is desired to specify torsional springs, equal to 250.0, at nodes 66, 73, 80 and 85. The torsional rotation coincides with the global x_1 coordinate axis. The associated history function is equal to 3. The complete command is thus

```
specification concentrated  rotational nodes 66:80:7 85
   11_spring 11_val 250.0 11_hist 3
```

36.1.3.1 Extended Example: Insight into Rotational Spring Nodal Specifications for BE2_2D elements

In order to gain insight into the effect of rotational spring specifications on the response of C^1 beam elements, consider the simply supported beam shown in Figure 36.3. Assume $E = 3.0\,\mathrm{x}\,10^5$, $I = 0.010$ and $Q = 100$.

From Bernoulli-Euler beam theory, the transverse displacement at the mid-span of a simply supported beam is

$$\delta_{L/2} = \frac{QL^3}{48EI} = 0.6944$$

For a *propped* cantilever (i.e., one with a roller support at the right end)

$$\delta_{L/2} = \frac{7QL^3}{768EI} = 0.3038$$

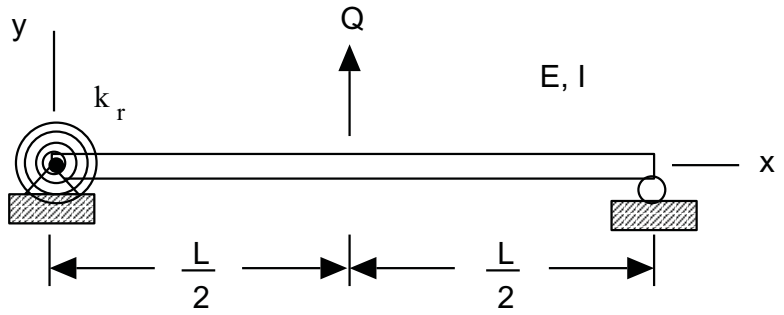

Figure 36.3: Simply supported beam with rotational spring nodal specification.

Modeling this beam using a two-element mesh of C^1 elements, analyses were performed for increasing values of rotational spring stiffness k^r (i.e., 33_Value). The results are summarized in Table 36.2.

It is evident that as k^r is increased, the transverse displacement at mid-span decreases from that associated with a simply supported beam to the value associated with a propped cantilever. In addition, the rotation (θ) at $x = 0$ tends to zero as k^r is increased, thus approaching a "fixed" (built-in) boundary. Initially, the value of θ at $x = L/2$ is zero due to the symmetric deflection of a simply supported beam. As k^r is increased, the rotation

Table 36.2: Simply Supported Beam with Rotational Spring Nodal Specification

k^r	Displacement at $x = L/2$	θ at $x = 0$	θ at $x = L/2$	θ at $x = L$
0.000e+00	6.944e-01	2.083e-01	0.000e+00	-2.083e-01
1.000e+02	6.554e-01	1.875e-01	2.604e-03	-1.979e-01
1.000e+04	3.361e-01	1.720e-02	2.389e-02	-1.128e-01
1.000e+06	3.042e-01	1.873e-04	2.602e-02	-1.043e-01
1.000e+08	3.038e-01	1.875e-06	2.604e-02	-1.042e-01
1.000e+10	3.038e-01	1.875e-08	2.604e-02	-1.042e-01

distribution ceases to be symmetric about the mid-span as it tends to that associated with a propped cantilever. Finally, it is interesting to note that the transverse displacement and rotation at $x = L/2$, as well as the rotation at $x = L$ change very little once k^r exceeds $1.000 \,\mathrm{x}\, 10^6$.

36.1.4 The SPECIFICATION CONCENTRATED SCALAR Command

Synopsis

The **SPECIFICATION CONCENTRATED SCALAR** command is used to specify scalar nodal specifications (i.e., heat flux per unit area, temperature, flow velocity and total hydraulic head) at boundary and interior nodes.

Syntax

The following syntax is associated with the **SPECIFICATION CONCENTRATED SCALAR** command:

SPECIFICAtion CONcentrated SCAlar NODes #:#:#
[FLUx | PHI] [HIStory ##] [VALue #.#]

Explanatory Notes

The **SPECIFICATION CONCENTRATED SCALAR** command is used to describe scalar nodal specifications. For **ANALYSIS TYPE MECHANICAL_THERMAL** and **ANALYSIS TYPE FLOW_MECHANICAL_THERMAL**[1] analyses, **FLUX** represents a heat flux per unit area (a *natural* boundary condition). For such analyses, **PHI** represents the temperature (an *essential* boundary condition). For **ANALYSIS TYPE SCALAR**, **FLUX** represents a flux, while **PHI** represents the primary dependent variable.

The default **SCALAR** specification is a **FLUX** equal to a **VALUE** of zero (0.0), with an associated **HISTORY** function equal to -2. Positive fluxes correspond to heat or material being *input* to an element.

If two or more contradictory specifications are made for **PHI** at the same node, the *first* of these is arbitrarily assumed to be correct. If two or more concentrated **FLUX** specifications are made at the same node, their *sum* is used. If both a **PHI** value and a **FLUX** value are specified for a node, the latter value (since it has no effect on the solution) is *ignored*.

[1] Refer to the **ANALYSIS TYPE** command, which is discussed in Section 5.9

Example of Command Usage

It is desired to specify temperatures of 105 degrees at nodes 21 to 34. The history function associated with this specifiction is equal to zero (0). In addition, temperatures equal to zero degrees are to be specified at nodes 98 to 112. The following commands are thus sufficient to describe the aforementioned specifications:

```
specification concentrated  scalar nodes 21:34 value 105.0 hist 0

specification concentrated  scalar nodes 98:112  phi  ! default temp. = 0.0
```

36.2 The SPECIFICATION LINE LINEAR Commands

The information supplied in conjunction with the **SPECIFICATION LINE LINEAR** commands allows the used to specify a *linearly varying* distributed fluid flow or heat flux normal to a straight or curved boundary. For two-node line elements, for three-node triangles, and for four-node quadrilateral elements, a linearly varying (including, of course, constant variation) a normal and/or tangential stress distribution can be applied along such a boundary. Nodal values equivalent to the distributed quantities are then automatically computed by the APES+ computer program. Figure 36.4 shows this process for the case of distributed tractions.

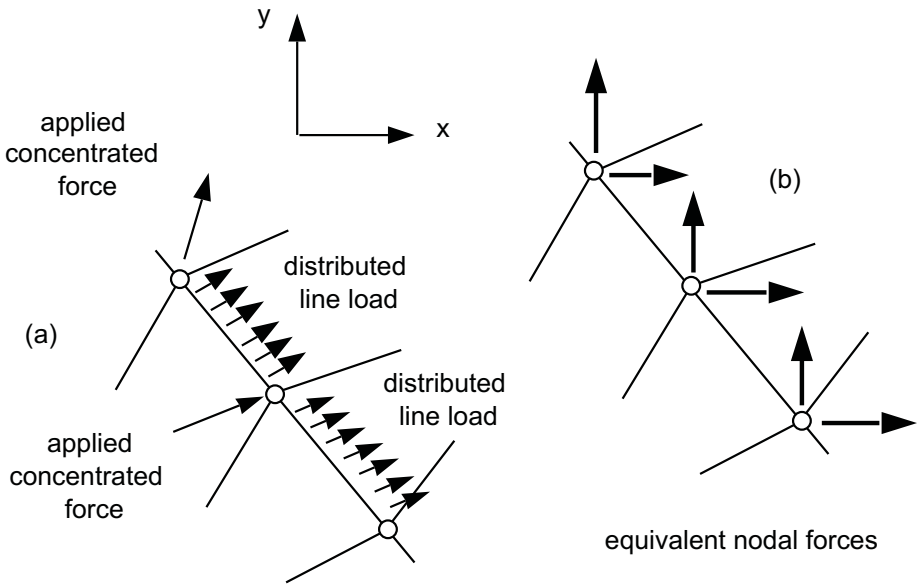

Figure 36.4: Schematic illustration of (a) Actual distributed and concentrated applied loads and (b) The concept of equivalent nodal forces.

If a distributed quantity is specified over several nodes in a sequence, then this specification can be generated with a single input record. This is achieved by supplying, on this record, a proper range (i.e., the beginning node number, ending node number, and numbering increment) of **NODE** numbers. In addition, suitable **HISTORY** function numbers must also be specified; these histories are used in applying the equivalent nodal quantities computed by APES+.

To be consistent with the order of node numbering employed in defining the element topology, during specification of a distributed quantity along a line, the nodes must likewise be specified in a counterclockwise sequence. A traction normal to an edge is assumed to be positive if, in conjunction with a counterclockwise node number specification, it is directed *into* the element. A tangential load is assumed to be positive if it acts in a counterclockwise

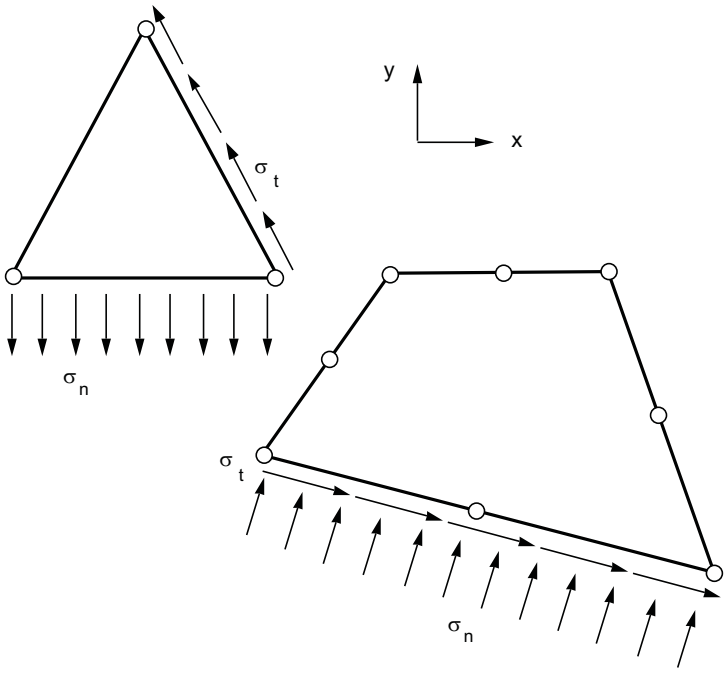

Figure 36.5: Normal and tangential distributed edge loads applied to a linear triangular element and to a quadratic serendipity element.

direction with respect to the loaded element (Fig. 36.5). Such definitions are necessary in order to avoid confusion when distributed loads are specified along the interface between two elements, such as that shown in Figure 36.6. Assuming a counterclockwise node number specification, if the applied tractions are assumed to act on element 1, they are considered to be positive. If they instead are taken as acting on element 2, these same tractions are negative.

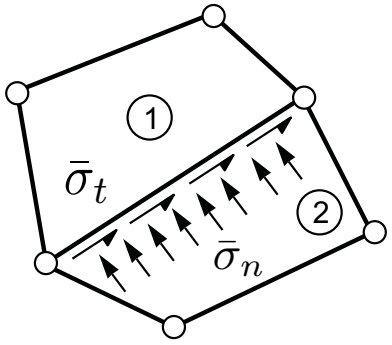

Figure 36.6: Edge loads specified along a typical element interface.

The following **SPECIFICATION LINE LINEAR** commands are available in the APES+ computer program:

- The **SPECIFICATION LINE LINEAR FLOW** command (see Section 36.2.1).
- The **SPECIFICATION LINE LINEAR MECHANICAL** command (see Section 36.2.2).
- The **SPECIFICATION LINE LINEAR SCALAR** command (see Section 36.2.3).

36.2.1 The SPECIFICATION LINE LINEAR FLOW Command

Synopsis

The **SPECIFICATION LINE LINEAR FLOW** command is used to specify a distributed flow normal to a given line associated with two-dimensional**ANALYSIS TYPE FLOW_MECHANICAL_THERMAL**[2] analyses.

Syntax

The following syntax is associated with the **SPECIFICATION LINE LINEAR FLOW** command:

SPECIFICAtion LINe LINear FLOw NODes #:#:#
[HIStory ##] [q_Begin ##] [q_End ##]

Explanatory Notes

The **LINEAR** keyword implies that the flow is distributed linearly. As such, the **SPECIFICATION LINE LINEAR FLOW** command should only be used in conjunction with *linear* continuum elements (e.g., Q4P4c, Q5P4c, T4P3c, etc.).

The values associated with the keywords **q_BEGIN** and **q_END** represent the values of distributed flow specified at the first (beginning) and last (end) nodes, respectively, in the line along which the flow is applied. The default value for all specified flows is equal to a value of zero (0.0), with associated **HISTORY** function equal to -2.

[2] Refer to the**ANALYSIS TYPE** command, which is discussed in Section 5.9

36.2.2 The SPECIFICATION LINE LINEAR MECHANICAL Command

Synopsis

The **SPECIFICATION LINE LINEAR MECHANICAL** command is used to specify a distributed normal and/or tangential stress distribution along a line associated with two-dimensional **ANALYSIS TYPE MECHANICAL**, **ANALYSIS TYPE MECHANICAL_THERMAL**, **ANALYSIS TYPE FLOW_MECHANICAL**, and **ANALYSIS TYPE FLOW_MECHANICAL_THERMAL**[3] analyses.

Syntax

The following syntax is associated with the **SPECIFICATION LINE LINEAR MECHANICAL** command:

SPECIFICAtion LINe LINear MEChanical NODes #:#:#
[1_HIStory ##] [2_HIStory ##]
[NP_Begin ##] [NP_End ##] [TP_Begin ##] [TP_End ##]

Explanatory Notes

The **LINEAR** keyword implies that the stress is distributed linearly. As such, the **SPECIFICATION LINE LINEAR MECHANICAL** command should only be used in conjunction with *linear* continuum elements (e.g., Q4P0, Q4P1d, Q4P4c, T3P0, T4P3c, etc.).

The "N" and "T" keyword prefixes refer to the directions normal and tangential, respectively, to the surface of the body being analyzed. The values associated with the keywords **NP_BEGIN** and **TP_BEGIN** represent the values of stress normal and tangential, respectively, to the body at the first (beginning) node in the line along which the stress is applied. The keywords **NP_END** and **TP_END** represent the values of stress normal and tangential, respectively, to the body at the last (end) node in the line along which the stress is applied. The sign convention associated with the specification of distributed stresses has been discussed in Section 36.2. The default values for all specified stresses are all equal to zero (0.0), with an associated **HISTORY** function equal to -2.

[3] Refer to the **ANALYSIS TYPE** command, which is discussed in Section 5.9

Example of Command Usage

To specify the normal stress loading along the edge shown in Figure 36.7, and varying according to the pre-set history function number zero (0), the user would specify either of the following commands:

```
specification line  linear mechanical  nodes  11:2:-3 np_begin 700.0 &
        np_end 350.0  1_history 0   2_history 0

specification line  linear mechanical  nodes  2:11:3 np_begin -350.0 &
        np_end -700.0  1_history 0   2_history 0
```

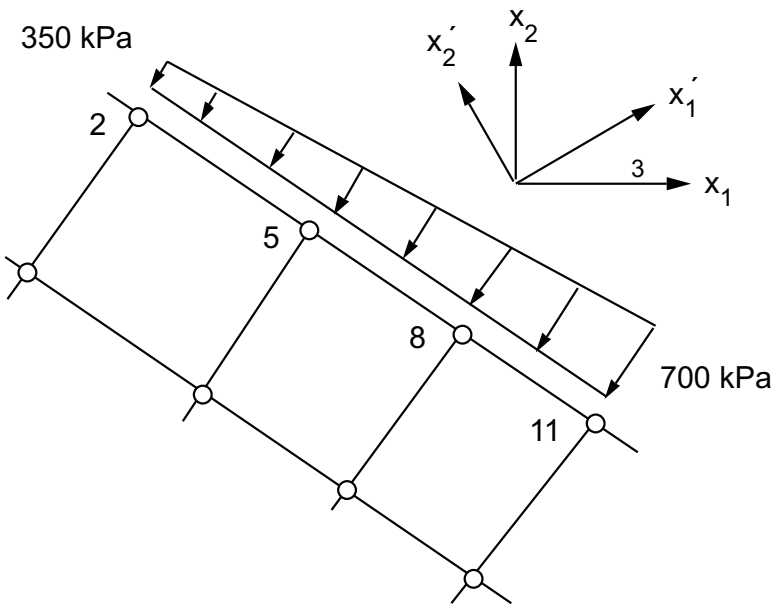

Figure 36.7: Boundary with normal traction (stress) applied.

36.2.3 The SPECIFICATION LINE LINEAR SCALAR Command

Synopsis

The **SPECIFICATION LINE LINEAR SCALAR** command is used to specify a distributed flux normal to a given line associated with two-dimensional **ANALYSIS TYPE SCALAR**[4] analyses.

Syntax

The following syntax is associated with the **SPECIFICATION LINE LINEAR SCALAR** command:

SPECIFICAtion LINe LINear SCAlar NODes #:#:#
[HIStory ##] [q_Begin ##] [q_End ##]

Explanatory Notes

The **LINEAR** keyword implies that the flux is distributed linearly. As such, the **SPECIFICATION LINE LINEAR SCALAR** command should only be used in conjunction with *linear* scalar continuum elements.

The values associated with the keywords **q_BEGIN** and **q_END** represent the values of distributed flux specified at the first (beginning) and last (end) nodes, respectively, in the line along which the flux is applied. The default values for all specified fluxes are equal to a value of zero (0.0), with associated **HISTORY** function equal to -2.

[4]Refer to the **ANALYSIS TYPE** command, which is discussed in Section 5.9

Example of Command Usage

To specify the flux acting along the edge shown in Figure 36.8, and varying according to the pre-set history function number zero (0), the user would specify either of the following commands:

```
specification line  linear scalar  nodes  14:2:-3 q_begin 0.010 &
         q_end 0.005  history 0

specification line  linear mechanical  nodes  2:14:3 q_begin -0.005 &
         q_end 0.010   history 0
```

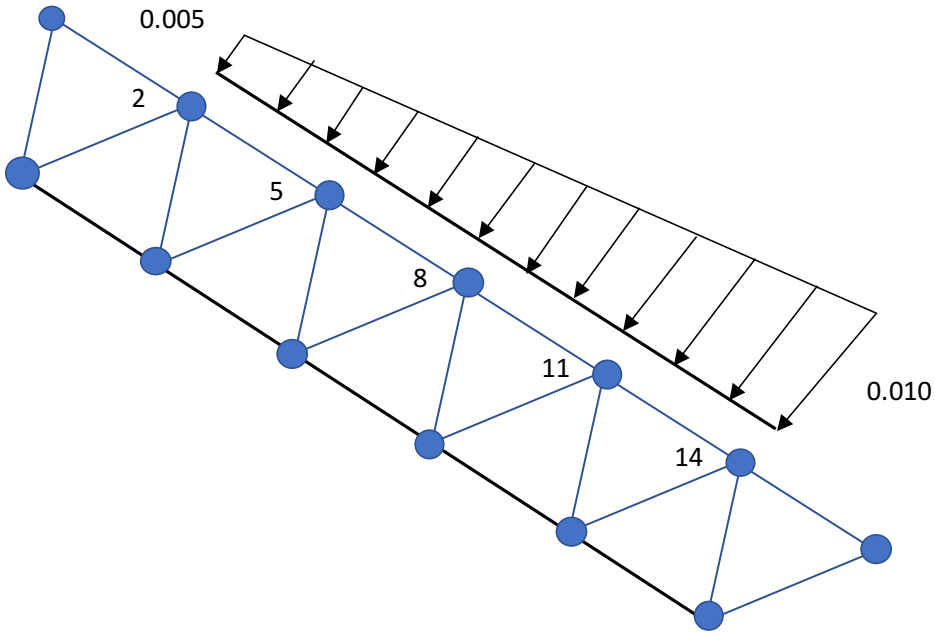

Figure 36.8: Boundary with normal flux applied.

36.3 The SPECIFICATION LINE QUADRATIC Commands

The information supplied in conjunction with **SPECIFICATION LINE QUADRATIC** commands allows the used to specify a *quadratically varying* distributed fluid flow or heat flux normal to a straight or curved boundary. For three-node line elements, for six-node triangles, and for eight- and nine-node quadrilateral elements, a quadratically varying (including, of course, trapezoidal, triangular, and constant variation) a normal and/or tangential stress distribution can be applied along such a boundary. Nodal values equivalent to the distributed quantities are then automatically computed by the APES+ computer program.

If a distributed quantity is specified over several nodes in a sequence, then this specification can be generated with a single input record. This is achieved by supplying, on this record, values for the beginning (**NODE_BEGIN**) and ending (**NODE_END**) node numbers, as well as the numbering increments **1_INCR** and **2_INCR**. The keyword **1_INCR** is used to specify the increment between corresponding node numbers; the keyword **2_INCR** is used to specify the increment between a corner (vertex) node number and the adjacent mid-side node number.

When specifying normal and/or tangential stress distributions, the "N" and "T" command prefixes refer to the directions normal and tangential to the surface of the body being analyzed. The keywords **NP_BEGIN** and **TP_BEGIN** represent the values of stress applied normal and tangential, respectively, to the body at the first (beginning) node in the line along which the loading is applied. The keywords **NP_END** and **TP_END** represent the values of stress normal and tangential, respectively to the body at the last (end) node in the line along which the loading is applied. The optional keywords **NP_MIDDLE** and **TP_MIDDLE** represent values of the stress at a node located somewhere between the beginning and end of the stress distribution.

In addition, suitable **HISTORY** function numbers must also be specified; these histories are used in applying the equivalent nodal quantities computed by APES+.

The following **SPECIFICATION LINE LINEAR** commands are available in the APES+ computer program:

- The **SPECIFICATION LINE QUADRATIC FLOW** command (see Section 36.3.1).
- The **SPECIFICATION LINE QUADRATIC MECHANICAL** command (see Section 36.3.2).
- The **SPECIFICATION LINE QUADRATIC SCALAR** command (see Section 36.3.3).

36.3.1 The SPECIFICATION LINE QUADRATIC FLOW Command

Synopsis

The **SPECIFICATION LINE QUADRATIC FLOW** command is used to specify a distributed flow normal to a given line associated with two-dimensional **ANALYSIS TYPE FLOW_MECHANICAL_THERMAL**[5] analyses.

Syntax

The following syntax is associated with the **SPECIFICATION LINE QUADRATIC FLOW** command:

SPECIFICAtion LINe QUAdratic FLOw
NODE_Begin ## NODE_End ##
[1_Incr ##] [2_Incr ##] [HIStory ##]
[q_Begin ##] [q_End ##] [X1_Middle #.#] [X2_Middle #.#] [q_Middle #.#]

Explanatory Notes

The **QUADRATIC** keyword implies that the flow is distributed quadratically. As such, the **SPECIFICATION LINE QUADRATIC FLOW** command should only be used in conjunction with *quadratic* continuum elements (e.g., Q8P4c, Q9P4c,T6P3c, etc.).

The values associated with the keywords **q_BEGIN** and **q_END** represent the values of distributed flow specified at the first (beginning) and last (end) nodes, respectively, in the line along which the flow is applied. If nodal specifications are to be generated automatically, the keyword **1_INCR** represents the increment between corresponding corner (vertex) node numbers. The keyword **2_INCR** represents the increment between a corner (vertex) node number and the adjacent mid-side node number.

For positive (i.e., fluid input to an element) values of flow specified along *exterior* boundaries, the nodes must be specified in a *counterclockwise* order; along an interior boundary (e.g., a hole) these nodes must be specified in a *clockwise* order.

The optional keyword **q_MIDDLE** specifies a flow that is known at the point have coordinates **X1_MIDDLE** and **X2_MIDDLE**.

The default value for all specified flows is equal to a value of zero (0.0), with associated **HISTORY** function equal to -2.

[5] Refer to the **ANALYSIS TYPE** command, which is discussed in Section 5.9

36.3.2 The SPECIFICATION LINE QUADRATIC MECHANICAL Command

Synopsis

The **SPECIFICATION LINE QUADRATIC MECHANICAL** command is used to specify a distributed normal and/or tangential stress distribution along along a line associated with the following analyses[6]:

- **ANALYSIS TYPE MECHANICAL**
- **ANALYSIS TYPE MECHANICAL_THERMAL**
- **ANALYSIS TYPE FLOW_MECHANICAL_THERMAL**

Syntax

The following syntax is associated with the **SPECIFICATION LINE QUADRATIC MECHANICAL** command:

SPECIFICAtion LINe QUAdratic MEChanical
NODE_Begin ## NODE_End ##
[1_Incr ##] [2_Incr ##]
[1_HIStory ##] [2_HIStory ##]
[NP_Begin ##] [NP_End ##] [TP_Begin ##] [TP_End ##]
[X1_Middle #.#] [X2_Middle #.#] [NP_Middle #.#] [TP_Middle #.#]

Explanatory Notes

The **SPECIFICATION LINE QUADRATIC MECHANICAL** command is used to specify a distributed normal and/or tangential stress distribution along a line associated with two-dimensional **ANALYSIS TYPE MECHANICAL**, **ANALYSIS TYPE MECHANICAL_THERMAL**, and **ANALYSIS TYPE FLOW_MECHANICAL_THERMAL** analyses. The **QUADRATIC** keyword implies that the stress is distributed quadratically. As such, the **SPECIFICATION LINE QUADRATIC MECHANICAL** command should only be used in conjunction with *quadratic* continuum elements (e.g., Q8P0, Q9P0, T6P0, Q8P4c, Q9P4c, T6P3c, etc.).

If nodal specifications are to be generated automatically, the keyword **1_INCR** represents the increment between corresponding corner (vertex) node numbers; the keyword **2_INCR**

[6]Refer to the **ANALYSIS TYPE** command, which is discussed in Section 5.9

represents the increment between a corner (vertex) node number and the adjacent mid-side node number.

The "N" and "T" keyword prefixes refer to the directions normal and tangential, respectively, to the surface of the body being analyzed. The values associated with the keywords **NP_BEGIN** and **TP_BEGIN** represent the values of stress normal and tangential, respectively, to the body at the first (beginning) node in the line along which the stress is applied. The keywords **NP_END** and **TP_END** represent the values of stress normal and tangential, respectively, to the body at the last (end) node in the line along which the stress is applied. The optional keywords **NP_MIDDLE** and **TP_MIDDLE** represent values of the stress at a node located somewhere between the beginning and he end of the stress distribution. The sign convention associated with the specification of distributed stresses has been discussed in Section 36.2.

The default values for all specified stresses are all equal to zero (0.0), with an associated **HISTORY** function equal to -2.

36.3.3 The SPECIFICATION LINE QUADRATIC SCALAR Command

Synopsis

The **SPECIFICATION LINE QUADRATIC SCALAR** command is used to specify a distributed flux normal to a given line associated with two-dimensional **ANALYSIS TYPE SCALAR**[7] analyses.

Syntax

The following syntax is associated with the **SPECIFICATION LINE QUADRATIC SCALAR** command:

SPECIFICAtion LINe QUAdratic SCAlar
NODE_Begin ## NODE_End ##
[1_Incr ##] [2_Incr ##] [HIStory ##]
[q_Begin ##] [q_End ##] [X1_Middle #.#] [X2_Middle #.#] [q_Middle #.#]

Explanatory Notes

The **QUADRATIC** keyword implies that the flow is distributed quadratically. As such, the **SPECIFICATION LINE QUADRATIC SCALAR** command should only be used in conjunction with *quadratic* scalar continuum elements.

The values associated with the keywords **q_BEGIN** and **q_END** represent the values of distributed flux specified at the first (beginning) and last (end) nodes, respectively, in the line along which the flux is applied. If nodal specifications are to be generated automatically, the keyword **1_INCR** represents the increment between corresponding corner (vertex) node numbers. The keyword **2_INCR** represents the increment between a corner (vertex) node number and the adjacent mid-side node number.

For positive (i.e., flux input to an element) values of flux specified along *exterior* boundaries, the nodes must be specified in a *counterclockwise* order; along an interior boundary (e.g., a hole) these nodes must be specified in a *clockwise* order.

The optional keyword **q_MIDDLE** specifies a flux that is known at the point have coordinates **X1_MIDDLE** and **X2_MIDDLE**.

The default value for all specified fluxes is equal to a value of zero (0.0), with associated **HISTORY** function equal to -2.

[7] Refer to the **ANALYSIS TYPE** command, which is discussed in Section 5.9

Chapter 37

The OUTPUT Commands

This chapter contains information related to commands that limit the amount of data written to an output file associated with finite element analyses using the APES$^+$ computer program. It consists of the following sections:

- General notes pertaining to **OUTPUT** commands (see Section 37.1).
- Information related to limiting the printing of element values of secondary dependent variables (e.g., strains and stresses): the **OUTPUT ELEMENTS PRINT** command (see Section 37.2).
- Information related to limiting the printing of element values of primary dependent variables: the **OUTPUT NODES PRINT** command (see Section 37.3).

37.1 OUTPUT commands: An Overview

Synopsis

Finite element analyses can generate a great deal of output, much of which is often not of interest. The **OUTPUT** commands collectively can be used to greatly restrict the amount of output to be printed. The user must, however, be aware that output which is not printed is lost and can only be regenerated by repeating the analysis. Thus if there is doubt as to whether some quantity will be of interest, it should be printed.

The purpose of the **OUTPUT ELEMENTS PRINT** commands is to eliminate unnecessary output generated by APES+. More precisely, if the time history of stresses, strains, pore pressure, heat fluxes, etc. is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of results by the user. Information associated with the elements specified in this section will be printed for every solution (time) step. In addition to the **OUTPUT ELEMENTS PRINT** commands, the user can control the type of element quantities printed (e.g., principal strains and/or stresses, average strains and/or stresses, etc.). This is done by specifying suitable keywords in conjunction with an **ELEMENT** command.

If the complete time history of nodal values of a primary dependent variable is desired only at a select few points, use of the **OUTPUT NODES PRINT** commands greatly speeds program output and facilitates inspection of the results by the user.

37.2 The OUTPUT ELEMENTS PRINT Command

Synopsis

The **OUTPUT ELEMENTS PRINT** command is used to specify all elements for which secondary dependent variables (e.g., strains, stresses, pore pressures, etc.) are to be printed. Use of this command greatly speeds program output and facilitates inspection of the results by the user. If no **OUTPUT ELEMENTS PRINT** commands are specified, and if no print control keywords have been issued in conjunction with an **ELEMENT** command, then secondary dependent variables shall be printed for *all* elements.

Syntax

The following syntax is associated with the **OUTPUT ELEMENTS PRINT** command:

OUTput ELEments PRInt #:#:#

Explanatory Notes

Finite element analyses can generate a great deal of output, much of which is often not of interest. The **OUTPUT ELEMENTS PRINT** command can be used to greatly restrict the amount of output to be printed. The user must, however, be aware that output which is not printed is lost and can only be regenerated by repeating the analysis. Thus if there is doubt as to whether some quantity will be of interest, it should be printed.

Example of Command Usage

To print primary dependent variables for elements 1 through 20, 25, 28, 31, 34, 37 and 40, specify either of the following commands:

```
out ele pri 1:20:1  25  28  31  34  37  40

output elements print 1:20, 25:40:3
```

37.3 The OUTPUT NODES PRINT Command

Synopsis

The **OUTPUT NODES PRINT** command is used to specify all nodes for which primary dependent variables (e.g., displacements, temperatures, etc.) are to be printed. Use of this command greatly speeds program output and facilitates inspection of the results by the user. If no **OUTPUT NODES PRINT** commands are specified, then primary dependent variables shall be printed for *all* nodes.

Syntax

The following syntax is associated with the **OUTPUT NODES PRINT** command:

OUTput NODes PRInt #:#:#

Explanatory Notes

Finite element analyses can generate a great deal of output, much of which is often not of interest. The **OUTPUT NODES PRINT** command can be used to greatly restrict the amount of output to be printed. The user must, however, be aware that output which is not printed is lost and can only be regenerated by repeating the analysis. Thus if there is doubt as to whether some quantity will be of interest, it should be printed.

Example of Command Usage

To print primary dependent variables for nodes 1 through 20, 25, 28, 31, 34, 37 and 40, specify either of the following commands:

```
out nod pri 1:20:1  25  28  31  34  37  40

output nodes print 1:20, 25:40:3
```

Chapter 38

The SOLUTION TIME Command

Synopsis

The **SOLUTION TIME** command is used to specify the solution time history. The associated keywords are repeated as many times as is necessary in order to specify all solution history information into which the analysis is subdivided. For all analyses at least one such record must be supplied.

Syntax

The following syntax is associated with the **SOLUTION TIME** command:

SOLution TIMe FINal #.# INCrements ## [RATio #.#] OUTput #:#:#

Explanatory Notes

Due to the time- and history dependence of inelastic soil constitutive models, and due to the nature of the consolidation process, a finite element analysis using the APES$^+$ computer program is, in general, time-dependent. For a problem in which the flow of pore fluid is not considered, and for which viscous effects are not considered in soil constitutive models, the actual rate at which time passes is not important - this is because, in such cases, all such models are *rate-independent*. For the purpose of modeling the history effects it is, however, still necessary to think in terms of time. In the case of a linear elastic, non-flow problem, the only role of time is to represent the loading history. If only final results are desired for these special circumstances, then only a single time step is required. Regardless of the analysis being performed, *at least one record* must be specified in this part of the input.

For convenience, the total solution history is broken into one or more "history segments". One **SOLUTION TIME** command is required for each such segment. The time at the end of a given history segment is specified in conjunction with the **FINAL** keyword. It is assumed that the first segment begins at time $t_{in} = 0.0$.

The number of solution (time) increments into which the history segment is to be subdivided is prescribed using the **INCREMENT** keyword. Within a given "history segment", the ratio of the lengths (in time) of two successive solution steps is equal to the prescribed spacing ratio specified in conjunction with the **RATIO** keyword[1]. The default spacing **RATIO** of 1.0 corresponds to *equally spaced* time steps.

The values specified in conjunction with the **OUTPUT ELEMENTS PRINT** and **OUTPUT NODES PRINT** commands control the extent of information printed in a given history segment[2].

In **ANALYSIS TYPE FLOW_MECHANICAL**, as well as in **ANALYSIS TYPE FLOW_MECHANICAL_THERMAL**[3] analyses involving time-dependent, inelastic (e.g., elastoplastic-viscoplastic) material idealizations, the selection of appropriate tie step lengths is complicated by the fact that two distinct processes are involved in the solution; i.e., the flow of pore fluid and the time-dependent nature of the material idealization. Thus, a certain amount of experimentation with successively smaller time steps will often be required. In this process of experimentation, several factors must be considered, namely:

- Abrupt changes in step size should be avoided; judicious use of the spacing **RATIO** facilitates smooth transitions from small to large time steps/
- Abrupt changes in applied loads or displacements will cause large flow gradients and will require small time steps.
- In proceeding from a nearly hydrostatic stress state to failure, a minimum of 10 to 25 soluton steps is typically required by the bounding surface elastoplastic-viscoplastic constitutive models.

Additional information concerning step sizes used for consolidation (i.e., coupled displacement-pore pressure) problems is given by Herrmann and Mish [41]. A certain amount of oscillation of the solution si to be expected and usually can be tolerated.

[1] Additional details concerning the **RATIO** keyword are given in the description of the **NODES** commands in Chapter 35. In particular, see the **NODES LINE** Command in Section 35.4.

[2] Addition information pertaining these commands is given in Sections 37.2 and 37.3, respectively.

[3] Refer to the **ANALYSIS TYPE** command, which is discussed in Section 5.9

Example of Command Usage

Either of the following commands can be used to specify a solution history ending at a time of 10.5, consisting of 15 progressively increasing increments with output at every second increment:

```
solution time final 10.5  increment 15  ratio 1.10

sol tim fin 10.5  inc 15  rat 1.10
```

Bibliography

[1] M. M. Al-Hussaini. Drained plane strain and triaxial compression tests on crushed napa basalt. Technical Report Technical Report S-71-2 (2), USAF Waterways Experiment Station, Vicksburg, MS, 1971.

[2] M. M. Al-Hussaini. Plane strain and triaxial compression tests on painted rock dam material. Technical Report Technical Report S-71-2 (3), USAF Waterways Experiment Station, Vicksburg, MS, 1972.

[3] M. M. Al-Hussaini and L. D. Johnson. Numerical analysis of a reinforced earth wall. In *Proceedings of the ASCE Symposium on Reinforced Earth*, pages 98–126, Pittsburgh, PA, 1978. ASCE.

[4] K. Z. Andrawes, A. McGown, R. F. Wilson-Fahmy, and M. M. Mashhour. The finite element method of analysis applied to soil-geotextile systems. In *Proceedings of the Second International Conference of Geotextiles*, pages 695–700, Las Vegas, NV, 1982.

[5] K. J. Bathe. *Finite Element Procedures*. Prentice Hall, Englewood Cliffs, NJ, 1996.

[6] R. J. Bathurst and R. G. Karpurapu. Large-scale reinforced-soil wall tests and numerical modelling of geosynthetic reinforced soil structures. Technical report, Royal Military College of Canada, Kingston, Ontario, CN, 1990. Report to the Chief of Construction and Properties, Department of National Defense.

[7] J. Bear. *Dynamics of Fluids in Porous Media*. American Elsevier Publishing Company, New York, NY, 1972.

[8] P. Bettess. Infinite elements. *International Journal for Numerical Methods in Engineering*, 11(1):53–64, 1977.

[9] M. A. Biot. General theory of three-dimensional consolidation. *Journal of Applied Physics*, 12(2):155–164, 1941.

[10] M. A. Biot. Theory of elasticity and consolidation for a porous anisotropic solid. *Journal of Applied Physics*, 26(2):182–185, 1955.

[11] M. A. Biot. General solutions of the equations of elasticity and consolidation for a porous material. *Journal of Applied Mechanics, Transactions of ASME*, 78:91–96, 1956.

[12] M. D. Boscardin, E. T. Selig, R-S. Lin, and G-R. Yang. Hyperbolic parameters for compacted clays. *Journal of Geotechnical Engineering, ASCE*, 116(1):88–104, 1990.

[13] P. C. Carman. *Flow of Gases Through Porous Media*. Academic, New York, NY, 1956.

[14] M. J. Chajes, V. N. Kaliakin, S. D. Holsinger, and A. J. Meyer. Experimental testing of composite wood beams for use in timber bridges. In *Fourth International Bridge Engineering Conference*, pages 371–380, Washington, DC, 1995. TRB, National Research Council.

[15] R. J. Chalaturnyk, J. D. Scott, and D. H. K. Chan. Stresses and deformations in a reinforced soil slope. *Canadian Geotechnical Journal*, 27(2):224–232, 1990.

[16] C-Y. Chang and J. M. Duncan. Analysis of soil movement around a deep excavation. *Journal of the Soil Mechanics and Foundations Division, ASCE*, 96(SM5):1655–1681, 1970.

[17] N. N. S. Chou. Performance predictions of the colorado geotextile test walls. In J. T. H. Wu, editor, *Geosynthetically-Reinforced Soil Retaining Walls*, pages 315–327, Rotterdam, 1992. Balkema.

[18] G. W. Clough and J. M. Duncan. Finite Element Analysis of Port Allen and Old River Locks. Technical Report TE69-3, University of California, Berkeley, Department of Civil Engineering, Institute of Transportation and Traffic Engineering, 1969.

[19] R. W. Clough and R. J. Woodward. Analysis of embankment stresses and deformations. *Journal of the Soil Mechanics and Foundations Division, ASCE*, 93(SM4):529–549, 1967.

[20] R. D. Cook, D. S. Malkus, M. E. Plesha, and R. J. Witt. *Concepts and Applications of Finite Element Analysis*. John Wiley and Sons, New York, NY, 4th edition, 2002.

[21] G. R. Cowper. The shear coefficient in timoshenko's beam theory. *Journal of Applied Mechanics, Transactions of ASME*, 33(2):335–340, 1966.

[22] L. Cui, A. H-D. Cheng, V. N. Kaliakin, Y. Abousleiman, and J.-C. Roegiers. Finite element analysis of anisotropic poroelasticity: A generalized mandel's problem and an inclined borehole problem. *International Journal for Numerical and Analytical Methods in Geomechanics*, 20:381–401, 1996.

[23] A. Curnier. A static infinite element. *International Journal for Numerical Methods in Engineering*, 19(10):1479–1488, 1983.

[24] Y. F. Dafalias and L. R. Herrmann. Bounding surface plasticity II: Application to isotropic cohesive soils. *Journal of Engineering Mechanics, ASCE*, 112(12):1263–1291, 1986.

[25] M. Dechasakulsom. *Modeling Time-Dependent Behavior of Geogrids and its Application to Geosynthetically Reinforced Walls*. PhD thesis, University of Delaware, Department of Civil and Environmental Engineering, 2000.

[26] C. S. Desai and H. J. Siriwardane. *Constitutive Laws for Engineering Materials with Emphasis on Geologic Materials*. Prentice Hall, Englewood Cliffs, NJ, 1984.

[27] E. Detouenay and A. H-D. Cheng. Fundamentals of poroelasticity. In J. A. Hudson, editor, *Comprehensive Rock Engineering: Principles, Practice and Projects*, chapter 5. Pergamon Press, Oxford, UK, 1993.

[28] J. M. Duncan. Hyperbolic stress-strain relationships. In R. K. Yong and H-Y. Ko, editors, *Proceedings of the Workshop on Limit Equilibrium, Plasticity and Generalized Stress-Strain in Geotechnical Engineering*, pages 443–460, New York, NY, 1980. ASCE.

[29] J. M. Duncan and C-Y. Chang. Nonlinear analysis of stress and strain in soils. *Journal of the Soil Mechanics and Foundations Division, ASCE*, 96(SM5):1629–1653, 1970.

[30] J. M. Duncan, P. Byrne, K. S. Wong, and P. Mabry. Strength, stress-strain and bulk modulus parameters for finite element analyses of stresses and movements in soil masses. Technical Report UCB/GT/80-01, University of California, Berkeley, Department of Civil Engineering, 1980.

[31] R. M. Ebeling, J. F. Peters, and R. E. Wahl. Prediction of reinforced sand wall performance. In J. T. H. Wu, editor, *Geosynthetically-Reinforced Soil Retaining Walls*, pages 243–253, Rotterdam, 1992. Balkema.

[32] B. Einarsson and Y. Shokin. Fortran 90 for the fortran 77 programmer. http://www.nsc.liu.se/f77tof90.html, 2005.

[33] M. Fortin. *Calcul numérique des écoulements par la méthode des éléments*. PhD thesis, Universite Paris, 1976.

[34] R. E. Goodman, R. L. Taylor, and T. L. Brekke. A model for the mechanics of jointed rock. *Journal of Soil Mechanics and Foundations Division, ASCE*, 94(SM3):637–659, 1968.

[35] A. E. Green. Hypoelasticity and plasticity. *Proceedings of the Royal Society*, 234: 46–59, 1956.

[36] L. R. Herrmann. Elasticity equations for nearly incompressible materials by a variational theorem. *AIAA Journal*, 3(10):1896–1900, 1965.

[37] L. R. Herrmann. Laplacian-isoparametric grid generation scheme. *Journal of the Engineering Mechanics Division, ASCE*, 102(EM5):749–756, 1976.

[38] L. R. Herrmann. User's manual for REA (general two dimensional soils and reinforced earth analysis program. Technical report, University of California, Davis, Department of Civil Engineering, 1978.

[39] L. R. Herrmann. Finite element analysis of contact problems. *Journal of Engineering Mechanics, ASCE*, 104(5):1043–1059, 1978.

[40] L. R. Herrmann and V. N. Kaliakin. User's manual for SAC-2, a two-dimensional nonlinear, time dependent, soil analysis code using the bounding surface elastoplasticity-viscoplasticity model. Technical report, University of California, Davis, Department of Civil Engineering, 1987. Volumes I and II.

[41] L. R. Herrmann and K. D. Mish. Finite element analysis for cohesive soil, stress and consolidation problems using bounding surface plasticity theory. Technical report, University of California, Davis, Department of Civil Engineering, 1983. CR84.006.

[42] L. R. Herrmann and K. D. Mish. User's manual for SAC-2, a two-dimensional nonlinear, time dependent soil analysis code using the bounding surface plasticity model. Technical report, University of California, Davis, Department of Civil Engineering, 1983. CR84.008.

[43] P. Hood and C. Taylor. Navier-stokes equation using mixed interpolation. In J. T. Oden, O. C. Zienkiewicz, R. H. Gallagher, and C. Taylor, editors, *Finite Elements in Flow Problems*, pages 121–132. University of Alabama at Huntsville Press, 1974.

[44] T. J. R. Hughes. *The Finite Element Method, Linear Static and Dynamic Finite Element Analysis*. Prentice Hall, Englewood Cliffs, NJ, 1987.

[45] E. Isaacson and H. B. Keller. *Analysis of Numerical Methods*. John Wiley and Sons, New York, NY, 1966.

[46] M. Jaber, G. R. Schmertmann, and J. G. Collin. Prediction of geosynthetic reinforced wall performance using finite element analysis. In J. T. H. Wu, editor, *Geosynthetically-Reinforced Soil Retaining Walls*, pages 305–313, Rotterdam, 1992. Balkema.

[47] N. Janbu. Soil compressibility as determined by oedometer and triaxial tests. In *Proceedings of the 3rd European Conference on Soil Mechanics and Foundation Engineering*, pages 19–25, Wissbaden, Germany, 1963.

[48] V. N. Kaliakin. An elastoplastic-viscoplastic bounding surface model for isotropic cohesive soils. In M. J. Keedwell, editor, *International Conference on Rheology and Soil Mechanics*, pages 147–163, Coventry, UK, 1988. Elsevier Applied Science.

[49] V. N. Kaliakin. A simple coordinate determination scheme for two-dimensional mesh generation. *Computers and Structures*, 43(3):505–516, 1992.

[50] V. N. Kaliakin. APES2D, Analysis Program for Earth Structures in Two Dimensions, Version 0.95. Technical Report 93-8, University of Delaware, Department of Civil Engineering, Newark, DE, 1993.

[51] V. N. Kaliakin. APES, Analysis Program for Earth Structures in Two and Three Dimensions, Version 2.0. Technical Report 95-1, University of Delaware, Department of Civil Engineering, Newark, DE, 1995.

[52] V. N. Kaliakin. *Approximate Solution Techniques, Numerical Modeling and Finite Element Methods*. Marcel Dekker, Inc., New York, NY, 2001.

[53] V. N. Kaliakin and Y. F. Dafalias. Simplifications to the bounding surface model for cohesive soils. *International Journal for Numerical and Analytical Methods in Geomechanics*, 13(1):91–100, 1989.

[54] V. N. Kaliakin and Y. F. Dafalias. Theoretical aspects of the elastoplastic-viscoplastic bounding surface model for cohesive soils. *Soils and Foundations*, 30(3):11–24, 1990. Japanese Society of Soil Mechanics and Foundation Engineering.

[55] V. N. Kaliakin and Y. F. Dafalias. Verification of the elastoplastic-viscoplastic bounding surface model for cohesive soils. *Soils and Foundations*, 30(3):25–36, 1990. Japanese Society of Soil Mechanics and Foundation Engineering.

[56] V. N. Kaliakin and M. Dechasakulsom. Development of a general time-dependent model for geogrids. *Geosynthetics International*, 9(4):319–344, 2002.

[57] V. N. Kaliakin and J. Li. Insight into deficiencies associated with commonly used zero-thickness interface elements. *Computers and Geotechnics*, 17(2):225–252, 1995.

[58] V. N. Kaliakin and K. J. Perano. Interp – a fortran callable data interpretation subroutine system. *Advances in Engineering Software*, 10(3):136–142, 1988.

[59] V. N. Kaliakin, A. N. Nieto-Leal, and M. Mashayekhi. Modeling the time- and temperature-dependent response of cohesive soils in a generalized bounding surface framework. *Transportation Infrastructure Geotechnology*, 5(3):250–286, 2018. URL DOI:10.1007/s40515-018-0060-3.

[60] R. Karpurapu and R. J. Bathurst. Finite element predictions for the colorado geosynthetic-reinforced soil retaining walls. In J. T. H. Wu, editor, *Geosynthetically-Reinforced Soil Retaining Walls*, pages 329–341, Rotterdam, 1992. Balkema.

[61] R. Karpurapu and R. J. Bathurst. Numerical investigation of controlled yielding of soil-retaining wall structures. *Geotextiles and Geomembranes*, 11(2):115–131, 1992.

[62] R. Karpurapu and R. J. Bathurst. Behaviour of geosynthetic reinforced soil retaining walls using the finite element method. *Computers and Geotechnics*, 17(3):279–299, 1995.

[63] A. D. Kerr. Elastic and viscoelastic foundation models. *Journal of Applied Mechanics, Transactions of ASME*, 31(3):491–498, 1964.

[64] A. Kèzdi. *Handbook of Soil Mechanics*. Elsevier, Amsterdam, The Netherlands, 1974.

[65] M. Khabbazian, V. N. Kaliakin, and C. L. Meehan. Performance of quasilinear elastic constitutive models in simulation of geosynthetic encased columns. *Computers and Geotechnics*, 38(8):998–1007, 2011.

[66] M. Khabbazian, V. N. Kaliakin, and C. L. Meehan. Modeling the behavior of geosynthetic encased columns: Influence of granular soil constitutive model. *International Journal of Geomechanics, ASCE*, 12(4):357–369, 2012.

[67] R. L. Konder. Hyperbolic stress-strain response: Cohesive soils. *Journal of the Soil Mechanics and Foundations Division, ASCE*, 89(SM1):115–143, 1963.

[68] R. L. Konder and J. M. Horner. Triaxial compression of a cohesive soil with effective octahedral stress control. *Canadian Geotechnical Journal*, 2(1):40–52, 1965.

[69] R. L. Konder and J. S. Zelasko. A hyperbolic stress-strain formulation for sands. In *Proceedings of the 2nd Pan American Conference on Soil Mechanics and Foundation Engineering*, pages 289–333, Sao Paulo, Brasil, 1963.

[70] R. L. Konder and J. S. Zelasko. Void ratio effects on the hyperbolic stress-strain response of a sand. In *Laboratory Shear Testing of Soils*, pages 250–257, Ottawa, 1963. ASTM.

[71] J. Koženy. Über kapillare leitung des wassers im boden. *Stizurgsberichte, Royal Academy of Science*, 136(1):271–306, 1927.

[72] F. H. Kulhawy and J. M. Duncan. Nonlinear finite element analysis of stresses and movements in oroville dam. Technical Report TE70-2, University of California, Berkeley, Department of Civil Engineering, Institute of Transportation and Traffic Engineering, 1970.

[73] F. H. Kulhawy and J. M. Duncan. Stresses and movements in oroville dam. *Journal of the Soil Mechanics and Foundations Division, ASCE*, 98(SM7):653–665, 1972.

[74] F. H. Kulhawy, J. M. Duncan, and H. B. Seed. Finite element analysis of stresses and movements in embankments during construction. Technical Report TE69-4, University of California, Berkeley, Department of Civil Engineering, Institute of Transportation and Traffic Engineering, 1969.

[75] P. V. Lade. *The Drained Stress-Strain and Strength Characteristics of Cohesionless Soils*. PhD thesis, University of California, Berkeley, Berkeley, CA, 1972.

[76] P. V. Lade. Calibration of the single hardening constitutive model for clays. In *Proceedings of the 11th International Conference of the Association for Computer Methods and Advances in Geomechanics (IACMAG)*, pages CD–ROM, Turin, Italy, 2005. Pàtron: Bologna.

[77] P. V. Lade and J. M. Duncan. Cubical triaxial tests on cohesionless soil. *Journal of the Soil Mechanics and Foundation Division, ASCE*, 99(SM10):793–812, 1973.

[78] P. V. Lade and M. K. Kim. Single hardening constitutive model for frictional materials, iii. comparisons with experimental data. *Computers and Geotechnics*, 6(1):31–47, 1988.

[79] P. V. Lade and R. B. Nelson. Modelling the elastic behaviour of granular materials. *International Journal for Numerical and Analytical Methods in Geomechanics*, 11(5): 521–542, 1987.

[80] K. L. Lee and H. B. Seed. Drained strength characteristics of sands. *Journal of the Soil Mechanics and Foundations Division, ASCE*, 93(6):117–141, 1967.

[81] S. G. Lekhnitskii. *Theory of Elasticity of an Anisotropic Body*. Mir Publishers, Moscow, Russia, 1981.

[82] H. I. Ling and F. Tatsuoka. Nonlinear analysis of reinforced soil structures by modified cande (m-cande). In J. T. H. Wu, editor, *Geosynthetically-Reinforced Soil Retaining Walls*, pages 279–291, Rotterdam, 1992. Balkema.

[83] A. E. H. Love. *A Treatise on the Mathematical Theory of Elasticity*. Cambridge Univ. Press, Cambridge, UK, 4th edition, 1927.

[84] J. Lubliner. *Plasticity Theory*. Macmillan Publishing Co., New York, NY, 1990.

[85] R. H. MacNeal. *Finite Elements: Their Design and Performance*. Marcel Dekker, New York, NY, 1994.

[86] G. F. Marcozzi. Bearing capacity of rigid and flexible foundations on dense sand: An experiment study. Master's thesis, University of Delaware, Department of Civil Engineering, 1988.

[87] G. Mesri and R. E. Olson. Mechanisms controlling the permeability of clays. *Clays and Clay Minerals*, 19(3):151–158, 1971.

[88] K. D. Mish and L. R. Herrmann. User's manual for SAC-3, a three-dimensional nonlinear, time dependent soil analysis code using the bounding surface plasticity model. Technical report, University of California, Davis, Department of Civil Engineering, 1983. CR84.009.

[89] T. Moan. Experiences with orthogonal polynomials and 'best' numerical integration formulas on a triangle; with particular reference to finite element approximations. *Zeitschrift für Angewandte Mathematik und Mechanik*, 54(8):501–508, 1974.

[90] P. E. Nielan, K. J. Perano, and W. E. Mason. Antipasto: An interactive mesh generator and preprocessor for two-dimensional analysis programs. Technical Report SAND90-8203, Sandia National Laboratory, 1990.

[91] U. S. Department of Agriculture. *The Encyclopedia of Wood*. Skyhorse Publishing, New York, NY, 2007.

[92] E. Ogisako, H. Ochiai, S. Hayashi, and A. Sakai. Fem analysis of polymer grid reinforced-soil retaining walls and its application to the design method. In T. Yamanouchi, N. Miura, and H. Ochiai, editors, *Proceedings of the International Geotechnical Symposium on Theory and Practice of Earth Reinforcement*, pages 559–564, Fukuoka, Japan, 1988. Balkema.

[93] D. R. J. Owen and E. Hinton. *Finite Elements in Plasticity–Theory and Practice*. Pineridge Press, Swansea, UK, 1980.

[94] K. J. Perano. Skyldu – simultaneous equation-solving subroutine system. Unpublished document, 1986.

[95] K. J. Perano. Compro – a subroutine system for syntactical analysis. Technical Report SAND89-8441, Sandia National Laboratory, 1989.

[96] K. J. Perano and V. N. Kaliakin. Interp – a fortran callable data interpretation subroutine system. Technical Report SAND87-8244, Sandia National Laboratory, 1989.

[97] J. Pitkäranta and R. Stenberg. Error bounds for the approximation of the stokes problem using bilinear/constant elements on irregular quadrilateral meshes. Technical Report MAT-A222, Helsinki University of Technology, Institute of Mathematics, 1984.

[98] C. J. Poran, V. N. Kaliakin, L. R. Herrmann, K. M. Romstad, D.-F. Lee, and C. K. Shen. Prediction of trial embankment behavior hertfordshire county councils - stansford abbotts. In *Proceedings of the Reinforced Earth Prediction Symposium*, King's College, London, UK, 1986.

[99] B. S. Qubain. *Stress-Deformation Behavior of Footing Foundations*. PhD thesis, Drexel University, Philadelphia, PA, 1999.

[100] B. S. Qubain, V. N. Kaliakin, and J. P. Martin. A hyperbolic constitutive model for sand behaviour with a variable bulk modulus. *Journal of Geotechnical Engineering, ASCE*, 129(2):158–162, 2003.

[101] W. Ramberg and W. R. Osgood. Description of stress-strain curves by three parameters. *National Advisory Committtee on Aeronautics*, (Technical Note 902), 1943. Washington, DC.

[102] A. M. Samarasinghe, Y. H. Huang, and V. P. Drnevich. Permeability and consolidation of normally consolidated soils. *Journal of the Geotechnical Engineering Division, ASCE*, 108(GT6):835–850, 1982.

[103] C. Taylor and P. Hood. A numerical solution of the navier stokes equations using the finite element technique. *Computers and Fluids*, 1(1):73–100, 1973.

[104] D. W. Taylor. *Fundamentals of Soil Mechanics*. John Wiley and Sons, New York, NY, 1st edition, 1948.

[105] R. L. Taylor, P. J. Beresford, and E. L. Wilson. A non-conforming element for stress analysis. *International Journal for Numerical Methods in Engineering*, 10(6):1211–1219, 1976.

[106] K. Terzaghi and R. B. Peck. *Soil Mechanics in Engineering Practice*. John Wiley and Sons, New York, NY, 2nd edition, 1967.

[107] M. Thompson and J. R. Willis. A reformation of the equations of anisotropic poroelasticity. *Journal of Applied Mechanics, Transactions of the ASME*, 58(3):612–616, 1991.

[108] S. P. Timoshenko. On the correction for shear of the differential equation for transverse vibrations of prismatic bars. *Philosophical Magazine*, 41:744–746, 1921.

[109] S. P. Timoshenko. On the transverse vibrations of bars of uniform cross-section. *Philosophical Magazine*, 43:125–131, 1922.

[110] S. P. Timoshenko. *History of Strength of Materials*. Dover Publications, Inc., New York, NY, 1983.

[111] C. Truesdell. Hypoelasticity. *Journal of Rational Mechanics*, 4(1):83–133, 1955.

[112] A. B. Vesic and G. W. Clough. Behavior of granular material under high stresses. *Journal of the Soil Mechanics and Foundations Division, ASCE*, 94(3):661–688, 1968.

[113] E. Winkler. *Die Lehre von der Elasticitset und Festigkeit*. Prag. Dominicus, 1867.

[114] K. S. Wong and J. M. Duncan. Hyperbolic stress-strain parameters for nonlinear finite element analysis of stresses and movements in soil masses. Technical Report TE 74-3, University of California, Berkeley, Department of Civil Engineering, Institute of Transportation and Traffic Engineering, 1974.

[115] J. T. H. Wu. Predicting performance of the denver walls: General report. In J. T. H. Wu, editor, *Geosynthetically-Reinforced Soil Retaining Walls*, pages 3–20, Rotterdam, 1992. Balkema.

[116] Z. Z. Yin and J. G. Zhu. Stress and deformation of reinforced soil retaining wall. In J. T. H. Wu, editor, *Geosynthetically-Reinforced Soil Retaining Walls*, pages 205–216, Rotterdam, 1992. Balkema.

[117] O. C. Zienkiewicz and R. L. Taylor. *The Finite Element Method*. Butterworth Heinemann, Oxford, UK, 5th edition, 2000.

[118] J. G. Zornberg and J. K. Mitchell. Finite element prediction of the performance of an instrumented geotextile-reinforced wall. In H. J. Siriwardane and M. M. Zaman, editors, *Proceedings of the Eighth International Conference of the Association for Computer Methods and Advances in Geomechanics*, pages 1433–1438, Morgantown, WV, 1994. Taylor and Francis Group.

[119] M. Zytynski, M. F. Randolph, R. Nova, and C. P. Wroth. On modeling the unloading-reloading behavior of soils. *International Journal for Numerical and Analytical Methods in Geomechanics*, 2(1):87–94, 1978.