ELEMENT MIXED FLOW Q8P4c commands

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 mixed, quadrilateral isoparametric “Serendipity” continuum element [2]. 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 (Figure 1).
  - 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 [1].

Syntax

The following syntax is used to describe a typical Q8P4c mixed-flow quadrilateral continuum element:

```
ELEMENT MIXed_Flow TYPE Q8P4c
NODes #:#:#: (MATERIAL #:) (PERmeability #:) (INITial #:)
(CONstruction #:) (EXCAvation #:) (THickness #:)
(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 numbering order of NODES associated with the Q8P4c element is shown in Figure 1. For this element the numbers \( n1 \) to \( n8 \), numbered sequentially, apply.

![Figure 1: Node Numbering Associated with a Typical Mixed 8-Node Quadratic (Q8P4c) Quadrilateral Continuum Element](image)

• The \textbf{MATERIAL} keyword is used to specify the number of the material idealization associated with the element. The default value for the \textbf{MATERIAL} number is one (1).

• The \textbf{PERMEABILITY} keyword is used to specify the number of the permeability (hydraulic conductivity) idealization associated with the element. In order to maximize flexibility, the \textbf{PERMEABILITY} number differs from the \textbf{MATERIAL} number. The default value for the \textbf{PERMEABILITY} number is one (1).

• The \textbf{INITIAL} keyword is used to specify the initial state number associated with the element. The \textit{default} value for the \textbf{INITIAL} is zero (0).

• The incremental \textbf{CONSTRUCTION} and \textbf{EXCAVATION} numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A \textbf{CONSTRUCTION} number equal to \textit{zero} corresponds to a material in existence at the beginning of the analysis. Since this is the \textit{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.

- 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 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,

\[
\frac{|\varepsilon_{\text{vol}}|}{|\min(\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33})|}; \min(\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33}) \neq 0
\]

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 four pressure (vertex) nodes.

• The keyword **PRINT VELOCITY** 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.
Example of Command Usage
References
