ELEMENT MIXED H8P1d commands

Synopsis

The ELEMENT MIXED H8P1d command is used to describe all mixed, 8-node tri-linear hexahedral continuum elements with a constant discontinuous pressure field.

Remarks

• The H8P1d is a mixed, hexahedral isoparametric “Lagrangian” continuum element [2]. 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 1). The pressure degrees of freedom are “condensed” out [2] 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. [1]

• 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 ] 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 numbering order of NODES associated with H8P1d elements, which must be specified sequentially from 1 to 8, is shown in Figure 1).

Figure 1: Typical Mixed H8P1d 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 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 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,

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

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.
Example of Command Usage
References
