ELEMENT MIXED T6P1d commands

Synopsis

The **ELEMENT MIXED 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 1). The pressure degrees of freedom are “condensed” out [3] 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* analyses. 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. 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 [1].
Syntax

The following syntax is used to describe a typical mixed or mixed-penalty T6P1d continuum element:

\[
\text{ELE} \text{ment MIXed TYPe [ T6P1d ] NODes \#:#:#: (MATerial \#) (INITial \#) (CONstruction \#) (EXCavation \#) (THICKness \#.\#. \#) (PENalty) (DONT_PRINT_Results) (DONT_PRINT_STRAins) (DONT_PRINT_STREsses) (PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses) (PRINT_VOLumetric_strain)}
\]

Explanatory Notes

- The numbering order of NODES associated with T6P1d elements, which must be specified sequentially from 1 to 6, is shown in Figure 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 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 DON'T 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 DON'T PRINT STRAINS keyword indicates that element strains are not to be printed. Under the default condition both strains are printed.

- Specification of the keyword DON'T 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,

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

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

Simulation of Incompressible Material Response under Plane Strain Conditions

Consider a simple four-element mesh of T6P1d elements. Although the solution domain is square with a side dimension of 2.0, the middle node is purposely not placed at the centroid of the solution domain Ω. As such, the elements are mildly distorted.

A distributed traction of 20.0, acting in the negative x₂-coordinate direction, is applied along the top boundary of Ω. Along the right boundary, a distributed traction of 10.0, acting in the negative x₁-coordinate direction, is applied.

The material is characterized using the isotropic constitutive model valid for both compressible and incompressible elasticity \([\mathcal{L}_s]\). It is assumed to be essentially incompressible, with a Poisson’s ratio (ν) equal to 0.499999. The elastic modulus (E) is taken equal to 7.50. Consequently, the Lamé parameters are

\[ \lambda = \frac{E\nu}{(1 + \nu)(1 - 2\nu)} = 1.250 \times 10^6 \]

and

\[ \mu = \frac{E}{2(1 + \nu)} = 2.50 \]

The resulting displacements along the right boundary of Ω are equal to 2.000. Along the top boundary, the displacement is equal to -2.000.

The associated strains are \(\varepsilon_{11} = 1.000\) and \(\varepsilon_{22} = -1.000\), with \(\varepsilon_{33} = 0\) by assumption of plane strain conditions. In addition, the engineering shear strain is a numeric zero (i.e., \(\gamma_{12} \approx 0\)).

Since \(\sigma_{11} = -10.0\) and \(\sigma_{22} = -20.0\) as a result of the specified distributed tractions, it follows that under plane strain conditions

\[ \sigma_{33} = \nu(\sigma_{11} + \sigma_{22}) = -15.0 \]

Finally, in the incompressible limit the pressure associated with the isotropic elastic material idealization approaches the mean pressure \(\bar{p}\); viz.,

\[ \bar{p} = \frac{1}{3}(\sigma_{11} + \sigma_{22} + \sigma_{33}) = -15.0 \]

The input data associated with this problem is given next.

```plaintext
ana title "simple 'patch' to test element behavior in the incompressible limit"
ana title "4-element mesh of T6P1d mixed triangles"
!
analysis action analyze
analysis type mechanical
analysis idealization plane_strain
analysis temp transient
!
echo func off
echo grav off
```
echo ini off
echo warn off
!
integration time parameter 0.50
!
dim max material incompressible elastic 1
dim max nodes = 9
dim max q4p1 = 4
!
finished settings
!
mat elastic incomp num 1 desc "a Herrmann type material idealization" &
   lambda 1.250e+6 shear 2.500
!
notes line number 1
notes line number 3 x1 2.0 incr 1
notes line number 13 x1 2.0 x2 2.0 incr 5
notes line number 11 x2 2.0 incr -1
notes line number -1 incr -5
!
notes line number 4 x1 0.5 x2 0.5
notes line number 5 x1 1.5 x2 0.5
!
  NOTE: middle node in mesh is not at the centroid

notes line number 7 x1 1.2 x2 1.2
!
notes line number 9 x1 0.5 x2 1.5
notes line number 10 x1 1.5 x2 1.5
!
element mixed type "t6p1" nodes 1 3 7 2 5 4 mat 1
element mixed type "t6p1" nodes 3 13 7 8 10 5 mat 1
element mixed type "t6p1" nodes 13 11 7 12 9 10 mat 1
element mixed type "t6p1" nodes 1 7 11 4 9 6 mat 1
!
spec conc mech nodes 1:3 2_disp
spec conc mech nodes 1:11:5 1_dis
!
specification line quad mech node_b 13 node_end 11 1_incr -2 2_incr -1 2_hist 0 &
   np_begin 20.0 np_end 20.0
!
specification line quad mech node_b 3 node_end 13 1_incr 10 2_incr 5 1_hist 0 2_hist 0 &
   np_begin 10.0 np_end 10.0
!
finished data
!
solution time final 1.0 increments 1 output 1:10:1
!
Using the above data in conjunction with the APES computer program, the results shown below are obtained. For clarity, the "header" that is printed at the top of the file is omitted from this file.

simple 'patch' to test element behavior in the incompressible limit
4-element mesh of T6P1d mixed triangles

<table>
<thead>
<tr>
<th>DYNAMIC STORAGE ALLOCATION</th>
</tr>
</thead>
</table>

Largest NODE number which can used in the mesh = 13

Max. no. of INCOMPRESSIBLE LINEAR ELASTIC materials = 1

Max. no. of 6-node tri. mixed (T6P1d) elements = 4

--- MECHANICAL analysis shall be performed
--- Fluid flow is NOT accounted for in the analysis
--- Thermal effects are NOT accounted for in analysis

--- TWO-DIMENSIONAL solution domain assumed
   (PLANE STRAIN idealization)

--- Nodal coordinates will NOT be updated

--- solver type used: SKYLINE
--- storage type: SYMMETRIC

--- "Isoparametric" mesh generation scheme used
In approximating time derivatives, the value of "THETA" = 5.000E-01

= NONLINEAR ANALYSIS INFORMATION =

--> LINEAR analysis

= MATERIAL IDEALIZATIONS =

--> Material number:  1

 type: incompressible (or nearly so) linear elastic
 info.: a Herrmann type material idealization

lambda (Lame' parameter) = 1.250E+06
Elastic shear modulus = 2.500E+00

Elastic bulk modulus of the solid phase = 0.000E+00
Material density of the solid phase = 0.000E+00
Combined bulk modulus for solid/fluid = 1.250E+06

= NODAL COORDINATES =

node :  1  x1 =  0.000E+00  x2 =  0.000E+00
node :  2  x1 =  1.000E+00  x2 =  0.000E+00
node :  3  x1 =  2.000E+00  x2 =  0.000E+00
node :  4  x1 =  5.000E-01  x2 =  5.000E-01
node :  5  x1 =  1.500E+00  x2 =  5.000E-01
node :  6  x1 =  0.000E+00  x2 =  1.000E+00
node :  7  x1 =  1.200E+00  x2 =  1.200E+00
node :  8  x1 =  2.000E+00  x2 =  1.000E+00
node :  9  x1 =  5.000E-01  x2 =  1.500E+00
node : 10  x1 =  1.500E+00  x2 =  1.500E+00
node : 11  x1 =  0.000E+00  x2 =  2.000E+00
node :  12  x1 = 1.000E+00  x2 = 2.000E+00
node :  13  x1 = 2.000E+00  x2 = 2.000E+00

node point specifications

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---

---
Node (coordinates)
Number specification:

1 : (x1 = 0.000E+00, x2 = 0.000E+00) 
displacement-1 = 0.000E+00 ; history no. = -2
displacement-2 = 0.000E+00 ; history no. = -2

2 : (x1 = 1.000E+00, x2 = 0.000E+00) 
force-1 = 0.000E+00 ; history no. = -2
displacement-2 = 0.000E+00 ; history no. = -2

3 : (x1 = 2.000E+00, x2 = 0.000E+00) 
force-1 = -3.333E+00 ; history no. = 0
displacement-2 = 0.000E+00 ; history no. = -2

6 : (x1 = 0.000E+00, x2 = 1.000E+00) 
force-1 = 0.000E+00 ; history no. = -2
force-2 = 0.000E+00 ; history no. = -2

8 : (x1 = 2.000E+00, x2 = 1.000E+00) 
force-1 = -1.333E+01 ; history no. = 0
force-2 = 0.000E+00 ; history no. = 0

11 : (x1 = 0.000E+00, x2 = 2.000E+00) 
force-1 = 0.000E+00 ; history no. = -2
force-2 = -6.667E+00 ; history no. = 0

12 : (x1 = 1.000E+00, x2 = 2.000E+00) 
force-1 = 0.000E+00 ; history no. = -2
force-2 = -2.667E+01 ; history no. = 0

13 : (x1 = 2.000E+00, x2 = 2.000E+00) 
force-1 = -3.333E+00 ; history no. = 0
force-2 = -6.667E+00 ; history no. = 0

end of mathematical model data

At time 1.000E+00 (step no. 1) NO iteration was required
--> element 1 ( type = T6P1d ):
....................................
@(x1 = 9.778E-01, x2 = 3.111E-01):
eps_11 = 1.000E+00 ; eps_22 = -1.000E+00 ; eps_33 = 0.000E+00 ; gam_12 = 7.896E-07
sig_11 = -1.000E+01 ; sig_22 = -2.000E+01 ; sig_33 = -1.500E+01 ; sig_12 = 1.974E-06 ; p = 1.500E+01

--> element 2 ( type = T6P1d ):
....................................
@(x1 = 1.644E+00, x2 = 9.778E-01):
eps_11 = 1.000E+00 ; eps_22 = -1.000E+00 ; eps_33 = 0.000E+00 ; gam_12 = 2.683E-06
sig_11 = -1.000E+01 ; sig_22 = -2.000E+01 ; sig_33 = -1.500E+01 ; sig_12 = 6.706E-06 ; p = 1.500E+01

--> element 3 ( type = T6P1d ):
....................................
@(x1 = 9.778E-01, x2 = 1.644E+00):
eps_11 = 1.000E+00 ; eps_22 = -1.000E+00 ; eps_33 = 0.000E+00 ; gam_12 = 9.951E-07
sig_11 = -1.000E+01 ; sig_22 = -2.000E+01 ; sig_33 = -1.500E+01 ; sig_12 = 2.488E-06 ; p = 1.500E+01

--> element 4 ( type = T6P1d ):
....................................
@(x1 = 3.111E-01, x2 = 9.778E-01):
eps_11 = 1.000E+00 ; eps_22 = -1.000E+00 ; eps_33 = 0.000E+00 ; gam_12 = -1.055E-06
sig_11 = -1.000E+01 ; sig_22 = -2.000E+01 ; sig_33 = -1.500E+01 ; sig_12 = -2.639E-06 ; p = 1.500E+01

node: 1 ( x1 = 0.000E+00, x2 = 0.000E+00 )
u_1 = -2.666E-24, u_2 = -2.370E-24
node:  2 ( x1 = 1.000E+00, x2 = 0.000E+00 )  
u_1 = 1.000E+00, u_2 = -2.133E-25  
node:  3 ( x1 = 2.000E+00, x2 = 0.000E+00 )  
u_1 = 2.000E+00, u_2 = -5.333E-24  
node:  4 ( x1 = 5.000E-01, x2 = 5.000E-01 )  
u_1 = 5.000E-01, u_2 = -5.000E-01  
node:  5 ( x1 = 1.500E+00, x2 = 5.000E-01 )  
u_1 = 1.500E+00, u_2 = -5.000E-01  
node:  6 ( x1 = 0.000E+00, x2 = 1.000E+00 )  
u_1 = -7.467E-26, u_2 = -1.000E+00  
node:  7 ( x1 = 1.200E+00, x2 = 1.200E+00 )  
u_1 = 1.200E+00, u_2 = -1.200E+00  
node:  8 ( x1 = 2.000E+00, x2 = 1.000E+00 )  
u_1 = 2.000E+00, u_2 = -1.000E+00  
node:  9 ( x1 = 5.000E-01, x2 = 1.500E+00 )  
u_1 = 5.000E-01, u_2 = -1.500E+00  
node: 10 ( x1 = 1.500E+00, x2 = 1.500E+00 )  
u_1 = 1.500E+00, u_2 = -1.500E+00  
node: 11 ( x1 = 0.000E+00, x2 = 2.000E+00 )  
u_1 = -5.926E-26, u_2 = -2.000E+00  
node: 12 ( x1 = 1.000E+00, x2 = 2.000E+00 )  
u_1 = 1.000E+00, u_2 = -2.000E+00  
node: 13 ( x1 = 2.000E+00, x2 = 2.000E+00 )  
u_1 = 2.000E+00, u_2 = -2.000E+00  

apes -> end of analysis . . . . . . . .
References

