ELEMENT IRREDUCIBLE T3P0 command

Synopsis

The ELEMENT IRREDUCIBLE T3P0 command is used to describe all irreducible 3-node liner 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 IRReduceble TYPe T3P0 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 numbering order of **NODES** associated with **T3P0** elements, which must be specified sequentially from 1 to 3, is shown in Figure 1.

**NOTE:** Presently APES does *not* possess the ability to generate **T3P0** 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 1: Node Numbering Associated with a Typical Irreducible 3-Node Linear Triangular Continuum Element](image)

- 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 **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 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,

\[
\varepsilon_{ratio} = \frac{|\varepsilon_{vol}^{(e)}|}{\sqrt{\left(\varepsilon_{11}^{(e)}\right)^2 + \left(\varepsilon_{22}^{(e)}\right)^2 + \left(\varepsilon_{33}^{(e)}\right)^2}}
\]

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 T3P0 element. Under the *default* condition the volumetric strain and the aforementioned ratio are *not* computed and printed.
Example of Command Usage

Element Performance in Simple “Patch” Test

In order to verify the implementation of the T3P0 elements, the simple “patch” of elements shown in Figure 2 is analyzed. Further details pertaining to the so-called engineering patch test are given in Appendix D of [1].

A plane stress idealization is assumed. Vertical displacement is constrained at nodes 1 and 2. Horizontal displacement is constrained at nodes 1 and 4. A tensile stress equal to 10.0 is applied along the edge 2-3. A compressive stress of 20.0 is applied along the edge 3-4.

Figure 2: Simple “Patch” of Linear (T3P0) Triangular Elements

The input data file associated with the analysis is given below.
ana tit "patch test C involving 8 linear T3P0 triangles"
ana tit "see Figure D.1(a) in Kaliakin (2001)."
!
analysis type mech
analysis idealization plane_stress
analysis temp transient
!
echo func off
echo grav off
echo ini off
echo warn off
!
integration time parameter 0.50
!
dim max material isotropic elastic 1
dim max nodes 7
dim max t3p0 8
!
finished settings
!
mat elastic isotropic number 1 desc " test 1 " mod = 1000.0 poisson 0.3
!
nodes line number 1
nodes line number 2 x1 1.0
nodes line number 3 x1 1.0 x2 1.0
nodes line number 4 x2 1.0
nodes line number 5 x1 0.70 x2 0.40
nodes line number 6 x1 0.50 x2 0.70
nodes line number 7 x1 0.30 x2 0.50
!
element irreducible typ "t3p0" nodes 1 2 5 mat 1
element irreducible typ "t3p0" nodes 2 3 5 mat 1
element irreducible typ "t3p0" nodes 3 4 6 mat 1
element irreducible typ "t3p0" nodes 4 1 7 mat 1
element irreducible typ "t3p0" nodes 1 5 7 mat 1
element irreducible typ "t3p0" nodes 5 3 6 mat 1
element irreducible typ "t3p0" nodes 7 6 4 mat 1
element irreducible typ "t3p0" nodes 5 6 7 mat 1
!
spec line linear mech nodes 2:3 1_his 0 2_his 0 np_beg -10.0 np_end -10.0
spec line linear mech nodes 3:4 1_his 0 2_his 0 np_beg -10.0 np_end -10.0
!
spec conc mech nodes 1 1_dis 2_dis
spec conc mech nodes 2 1_for 1_his 0 2_dis 2_his 0
The results shown below are obtained using the above data in conjunction with the APES computer program. For clarity, the “header” that is printed at the top of the file is omitted from this file.

patch test C involving 8 linear T3P0 triangles
see Figure D.1(a) in Kaliakin (2001).

======================================================================
| D Y N A M I C S T O R A G E A L L O C A T I O N |
======================================================================

Largest NODE number which can used in the mesh = 7

Max. no. of ISOTROPIC, LINEAR ELASTIC materials = 1

Max. no. of 3-node triangular (T3P0) elements = 8

======================================================================
= G E N E R A L A N A L Y S I S I N F O R M A T I O N =
======================================================================

--> 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 STRESS idealization)
--> Nodal coordinates will NOT be updated
--> solver type used: SKYLINE

--> storage type: SYMMETRIC
"Isoparametric" scheme used for native mesh generation (if applicable)

In approximating time derivatives, the value of "THETA" = 5.000E-01

--> LINEAR analysis

<<<< NONE >>>

---> Material number: 1

 type: isotropic linear elastic
 info.: test 1

 Modulus of Elasticity = 1.000E+03
 Poisson's ratio = 3.000E-01

 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 = 0.000E+00
=  N O D A L  C O O R D I N A T E S  =

node :  1  x1 =  0.000E+00  x2 =  0.000E+00
node :  2  x1 =  1.000E+00  x2 =  0.000E+00
node :  3  x1 =  1.000E+00  x2 =  1.000E+00
node :  4  x1 =  0.000E+00  x2 =  1.000E+00
node :  5  x1 =  7.000E-01  x2 =  4.000E-01
node :  6  x1 =  5.000E-01  x2 =  7.000E-01
node :  7  x1 =  3.000E-01  x2 =  5.000E-01

=  E L E M E N T  I N F O R M A T I O N  =

--> number:  1  (type: T3P0  ) (kind: IRREDUCIBLE  )
  nodes:
     1  2  5
  integration rule for primary variables: 1-point formula for triangles
  integration rule for secondary variables: 1-point formula for triangles
  material no.  = 1
    material type: isotropic linear elastic
    thickness = 1.000E+00

--> number:  2  (type: T3P0  ) (kind: IRREDUCIBLE  )
  nodes:
     2  3  5
  integration rule for primary variables: 1-point formula for triangles
  integration rule for secondary variables: 1-point formula for triangles
  material no.  = 1
    material type: isotropic linear elastic
    thickness = 1.000E+00

--> number:  3  (type: T3P0  ) (kind: IRREDUCIBLE  )
  nodes:
     3  4  6
  integration rule for primary variables: 1-point formula for triangles
  integration rule for secondary variables: 1-point formula for triangles
  material no.  = 1
    material type: isotropic linear elastic
    thickness = 1.000E+00
--> number: 4 (type: T3P0 ) (kind: IRREDUCIBLE )
nodes:
  4  1  7
integration rule for primary variables: 1-point formula for triangles
integration rule for secondary variables: 1-point formula for triangles
material no. = 1
material type: isotropic linear elastic
thickness = 1.000E+00

--> number: 5 (type: T3P0 ) (kind: IRREDUCIBLE )
nodes:
  1  5  7
integration rule for primary variables: 1-point formula for triangles
integration rule for secondary variables: 1-point formula for triangles
material no. = 1
material type: isotropic linear elastic
thickness = 1.000E+00

--> number: 6 (type: T3P0 ) (kind: IRREDUCIBLE )
nodes:
  5  3  6
integration rule for primary variables: 1-point formula for triangles
integration rule for secondary variables: 1-point formula for triangles
material no. = 1
material type: isotropic linear elastic
thickness = 1.000E+00

--> number: 7 (type: T3P0 ) (kind: IRREDUCIBLE )
nodes:
  7  6  4
integration rule for primary variables: 1-point formula for triangles
integration rule for secondary variables: 1-point formula for triangles
material no. = 1
material type: isotropic linear elastic
thickness = 1.000E+00

--> number: 8 (type: T3P0 ) (kind: IRREDUCIBLE )
nodes:
  5  6  7
integration rule for primary variables: 1-point formula for triangles
integration rule for secondary variables: 1-point formula for triangles
material no. = 1
material type: isotropic linear elastic
thickness = 1.000E+00
= NODE POINT SPECIFICATIONS =

Node 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 = 5.000E+00 ; history no. = 0
    displacement-2 = 0.000E+00 ; history no. = 0

3 : ( x1 = 1.000E+00, x2 = 1.000E+00 )
    force-1 = 5.000E+00 ; history no. = 0
    force-2 = -1.000E+01 ; history no. = 0

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

end of mathematical model data

At time 1.000E+00 (step no. 1): NO iteration was required

= ELEMENT STRAINS & STRESSES =
--> element  1 ( type = T3P0 ):
    .................................
    @(x1 = 5.667E-01, x2 = 1.333E-01):
    eps_{11} = 1.600E-02 ; eps_{22} = -2.300E-02 ; eps_{33} = 3.000E-03 ; gam_{12} = -6.268E-11
    sig_{11} = 1.000E+01 ; sig_{22} = -2.000E+01 ; sig_{33} = 0.000E+00 ; sig_{12} = -2.411E-08

--> element  2 ( type = T3P0 ):
    .................................
    @(x1 = 9.000E-01, x2 = 4.667E-01):
    eps_{11} = 1.600E-02 ; eps_{22} = -2.300E-02 ; eps_{33} = 3.000E-03 ; gam_{12} = 3.719E-11
    sig_{11} = 1.000E+01 ; sig_{22} = -2.000E+01 ; sig_{33} = 0.000E+00 ; sig_{12} = 1.430E-08

--> element  3 ( type = T3P0 ):
    .................................
    @(x1 = 5.000E-01, x2 = 9.000E-01):
    eps_{11} = 1.600E-02 ; eps_{22} = -2.300E-02 ; eps_{33} = 3.000E-03 ; gam_{12} = -4.728E-11
    sig_{11} = 1.000E+01 ; sig_{22} = -2.000E+01 ; sig_{33} = 0.000E+00 ; sig_{12} = -1.819E-08

--> element  4 ( type = T3P0 ):
    .................................
    @(x1 = 1.000E-01, x2 = 5.000E-01):
    eps_{11} = 1.600E-02 ; eps_{22} = -2.300E-02 ; eps_{33} = 3.000E-03 ; gam_{12} = -4.520E-11
    sig_{11} = 1.000E+01 ; sig_{22} = -2.000E+01 ; sig_{33} = 0.000E+00 ; sig_{12} = -1.738E-08

--> element  5 ( type = T3P0 ):
    .................................
    @(x1 = 3.333E-01, x2 = 3.000E-01):
    eps_{11} = 1.600E-02 ; eps_{22} = -2.300E-02 ; eps_{33} = 3.000E-03 ; gam_{12} = -1.231E-11
    sig_{11} = 1.000E+01 ; sig_{22} = -2.000E+01 ; sig_{33} = 0.000E+00 ; sig_{12} = -4.733E-09
---> element 6 ( type = T3P0 ):

............... 

@(x1 = 7.333E-01, x2 = 7.000E-01):

\[ \begin{align*}
\varepsilon_{11} &= 1.600E-02 ; \quad \varepsilon_{22} = -2.300E-02 ; \quad \varepsilon_{33} = 3.000E-03 ; \quad \gamma_{12} = -8.411E-11 \\
\sigma_{11} &= 1.000E+01 ; \quad \sigma_{22} = -2.000E+01 ; \quad \sigma_{33} = 0.000E+00 ; \quad \sigma_{12} = -3.235E-08
\end{align*} \]

---> element 7 ( type = T3P0 ):

............... 

@(x1 = 2.667E-01, x2 = 7.333E-01):

\[ \begin{align*}
\varepsilon_{11} &= 1.600E-02 ; \quad \varepsilon_{22} = -2.300E-02 ; \quad \varepsilon_{33} = 3.000E-03 ; \quad \gamma_{12} = -3.800E-12 \\
\sigma_{11} &= 1.000E+01 ; \quad \sigma_{22} = -2.000E+01 ; \quad \sigma_{33} = 0.000E+00 ; \quad \sigma_{12} = -1.462E-09
\end{align*} \]

---> element 8 ( type = T3P0 ):

............... 

@(x1 = 5.000E-01, x2 = 5.333E-01):

\[ \begin{align*}
\varepsilon_{11} &= 1.600E-02 ; \quad \varepsilon_{22} = -2.300E-02 ; \quad \varepsilon_{33} = 3.000E-03 ; \quad \gamma_{12} = -4.371E-11 \\
\sigma_{11} &= 1.000E+01 ; \quad \sigma_{22} = -2.000E+01 ; \quad \sigma_{33} = 0.000E+00 ; \quad \sigma_{12} = -1.681E-08
\end{align*} \]

At time 1.000E+00 (step no. 1):

======================================================================
= N O D A L Q U A N T I T I E S =
======================================================================

node: 1 ( x1 = 0.000E+00, x2 = 0.000E+00 )
\[ \begin{align*}
u_1 &= 1.901E-22, \quad u_2 = -4.986E-22
\end{align*} \]

node: 2 ( x1 = 1.000E+00, x2 = 0.000E+00 )
\[ \begin{align*}
u_1 &= 1.600E-02, \quad u_2 = -1.333E-22
\end{align*} \]

node: 3 ( x1 = 1.000E+00, x2 = 1.000E+00 )
\[ \begin{align*}
u_1 &= 1.600E-02, \quad u_2 = -2.300E-02
\end{align*} \]
node: 4 ( x1 = 0.000E+00, x2 = 1.000E+00 )
    u_1 = 1.538E-22, u_2 = -2.300E-02
node: 5 ( x1 = 7.000E-01, x2 = 4.000E-01 )
    u_1 = 1.120E-02, u_2 = -9.200E-03
node: 6 ( x1 = 5.000E-01, x2 = 7.000E-01 )
    u_1 = 8.000E-03, u_2 = -1.610E-02
node: 7 ( x1 = 3.000E-01, x2 = 5.000E-01 )
    u_1 = 4.800E-03, u_2 = -1.150E-02

apes -> end of analysis . . . . . . .
Bibliography