ELEMENT BAR_MECHANICAL 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 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 [1]. 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 numbering order of NODES associated with the L3P0 element is shown in Figure 1.

![Figure 1: Node Numbering Associated with Typical 3-Node Bar Element](image-url)
• 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 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.
Example of Command Usage
References