SPECIFICATION LINE LINEAR MECHANICAL command

Synopsis

The SPECIFICATION LINE LINEAR MECHANICAL command is used to describe specifications involving load (stress) distributed along a line associated with a two-dimensional mechanical, coupled flow-mechanical, or thermal-mechanical analysis.

Syntax

The following syntax is used to linearly varying tractions applied normal and tangential to a boundary:

```
SPECification LINE Linear MEChanical NODes #:## (1_History ##) (2_History ##)
(NP_Begin #:##) (NP_End #:##)
(TP_Begin #:##) (TP_End #:##)
(THIckness #:##)
```

Explanatory Notes

The SPECIFICATION LINE LINEAR MECHANICAL command can be used to apply a linearly varying normal and/or tangential distribution to the following linear elements:

- Two-node line (bar) elements (L2P0, L2FDP0).
- Three- and four-node triangles (T3P0, T4P3c).
- Four- and five-node quadrilateral elements (Q4P0, Q4P1d, Q4P4c, Q5P4c, QM6).

The “N” and “T” keyword prefixes refer to the directions normal and tangential to the surface of the body being analyzed, respectively. The values associated with the keywords NP_BEGIN and TP_BEGIN represent the values of pressure normal and tangential, respectively, to the body at the first (beginning) node in the line along which the pressure is applied.

The values associated with the keywords NP_END and TP_END represent the values of pressure normal and tangential, respectively, to the body at the last (end) node in the line along which the pressure is applied. The sign convention associated with the specification of distributed quantities is described at the beginning of this section.

1Possibly created as degenerate four-node quadrilaterals.
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 *default* values for all specified pressures are equal to zero \((0.0)\), with associated **HISTORY** function equal to -2.

**Example of Command Usage**

![Figure 1: Boundary with Normal Traction (Pressure) Applied](image)

To specify the normal pressure loading along the edge shown in Figure 1 (and varying according to the pre-set history function numbered zero), the user would specify either

```plaintext
spec line linear mech nodes 11:2:-3 np_begin 700.0 & np_end 350.0 1_history 0 2_history 0
```

or

```plaintext
spec line linear mech nodes 2:11:3 np_begin -350.0 & np_end -700.0 1_history 0 2_history 0
```

or

```plaintext
spec line linear mech nodes 2 5 8 11 np_begin -350.0 & np_end -700.0 1_history 0 2_history 0
```
References