Elaboration of Degrees of Freedom in NASTRAN/PATRAN by comparing "Rod" and "Beam" elements

Cyrus Hagigat
College of Engineering
Engineering Technology Department
The University of Toledo

Abstract:

PATRAN is a software that can be a pre-processor for NASTRAN Finite Element software and it can also be used as a stand-alone Finite Element software that produces the same results as NASTRAN as illustrated in the PATRAN verification manual. In PATRAN, the geometry of line elements is defined as 2, 3, or 4 node Bar Elements. Various line elements can be assigned to the Bars. Two of such line elements are "Rod" and "Beam" elements. Each element type has various degrees of freedom and if forces and/or moments are applied to degrees of freedom that do not apply to the element, they are ignored. However, PATRAN issues no warning that forces and/or moments are present that do not apply to the element being used and consequently are ignored in the Finite Element computations. Consequently, if care is not exercised regarding the Degree of Freedom of the elements in a model, wrong results are obtained without any error messages generated by the software. Examples in this article illustrate cases where wrong results are obtained through improper use of the software.

Introduction:

Various one-dimensional (line elements) in NASTRAN/PATRAN have various degrees of freedom.

Rod elements have only displacement degrees of freedom along their axis and torsional degrees of freedom around their axial axis. Consequently, Rod elements are suitable for example for truss analysis but are not suitable where there is any type of bending load present.

Bar elements have displacement degrees of freedom along their axis and perpendicular to their axis. However, Bar elements do not have any torsional degrees of freedom. Therefore, Bar elements are suitable for modeling bending but not suitable for modeling twisting.

If care is not exercised in the use of the elements and their associated degrees of freedom, wrong results are produced by the software and the software will not produce any warning and/or error message.

Technical discussion:

NASTRAN/PATRAN offers a number of one-dimensional "line elements." Two of these elements are "Rod" and "Beam."

Discussion of “Rod” element

The Rod element shown in figure 1 has two Finite Element nodes and is capable of supporting an axial force along its length and a torsion around its axial axis [1].

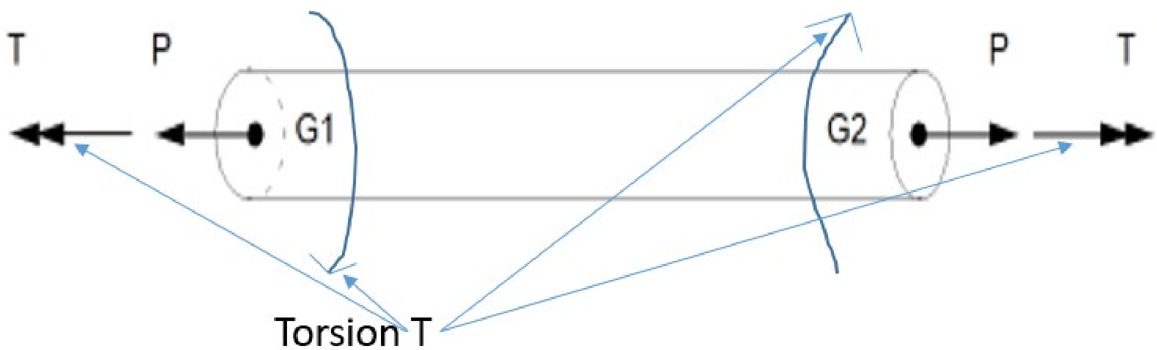


Figure 1: ROD element in NASTRAN/PATRAN

The minimum properties that must be defined for a “Rod” element are:

- Material properties (Modulus of Elasticity, Poisson’s ratio, Density is necessary for vibration analysis but not necessary for static stress analysis).
- Cross-sectional area.
- Polar moment of inertia (J) is necessary if torsional stresses are being calculated. This is called “Torsional Constant” in the PATRAN input dialog box shown in figure 2.
- Torsional Stress Coefficient is necessary for torsional stress but not necessary for axial stress. In the case of a circular shaft, this is the radius of the shaft. In the case of non-circular shafts, this has to be determined by other Mechanics of Materials techniques [2]. In the case of a circular shaft, the shear stress “T” is determined by

  T = (Shaft radius * Torque/ J)

  Therefore, the torsional constant for a circular Rod element is the shaft radius.

The polar moment of inertia formulas for the most common cross sections are shown in figure 3 [1].

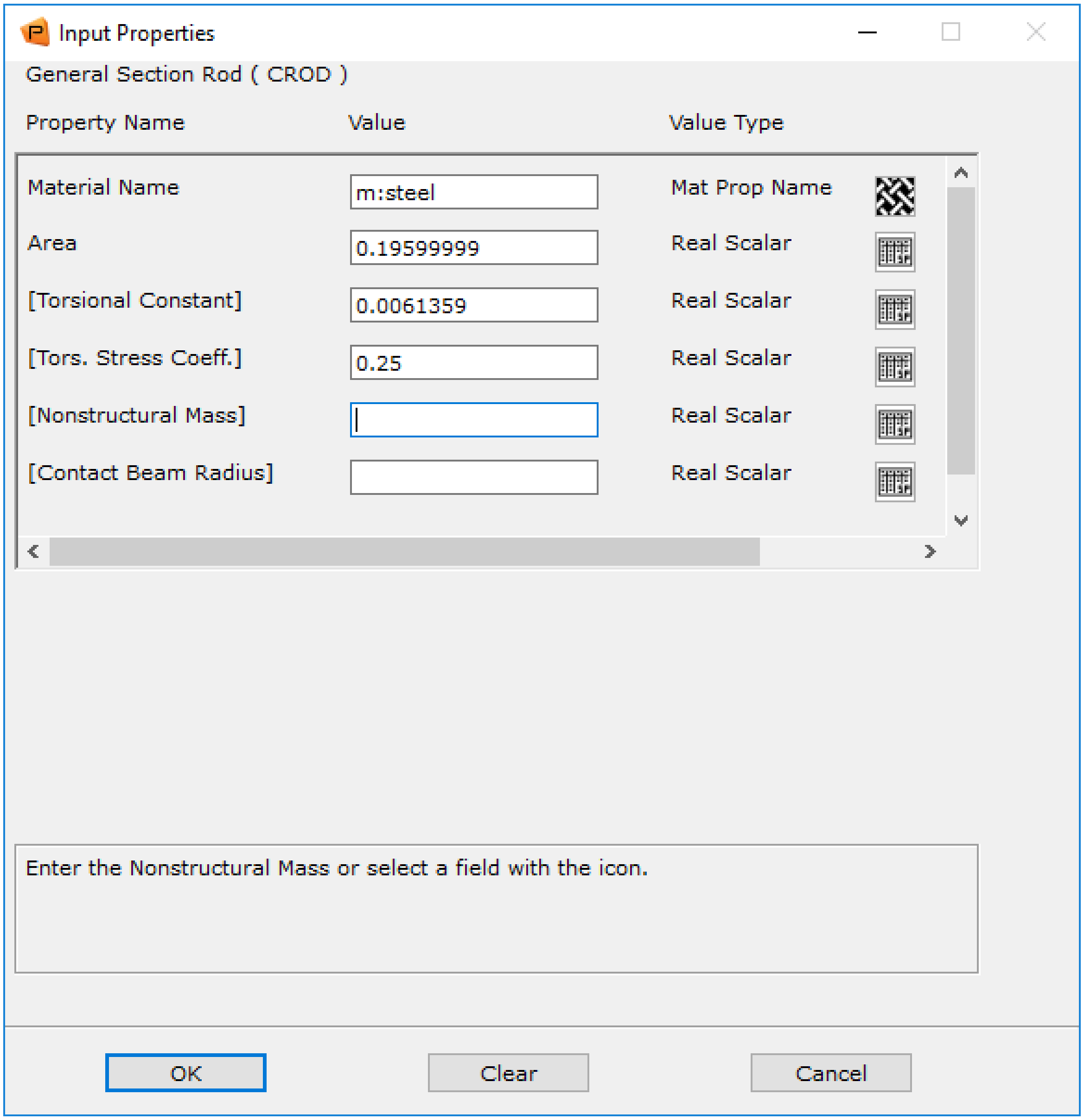


Figure 2: PATRAN input dialog box for a "Rod" element;
Numeric values shown are for the Finite Element models shown in figures 5, 7, 9 and 11

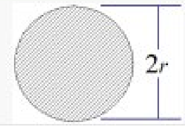
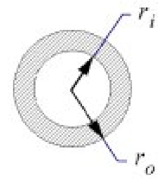
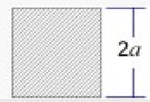
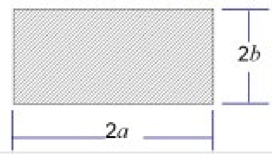
Formula for J	Cross Section
$J = \frac{1}{2}\pi r^4$	
$J = \frac{1}{2}\pi(r_o^4 - r_i^4)$	
$J = 2.25\ a^4$	
$J = ab^3\left[\frac{16}{3} - 3.36\frac{b}{a}\left(1 - \frac{b^4}{12a^4}\right)\right]$	

Figure 3: Formulas for calculation of polar moment of inertia (J)

The forces and moments (torques for “Rod” elements) are inputted under “Loads/BC” in PATRAN and the dialog box for inputting these parameters is shown in figure 4.

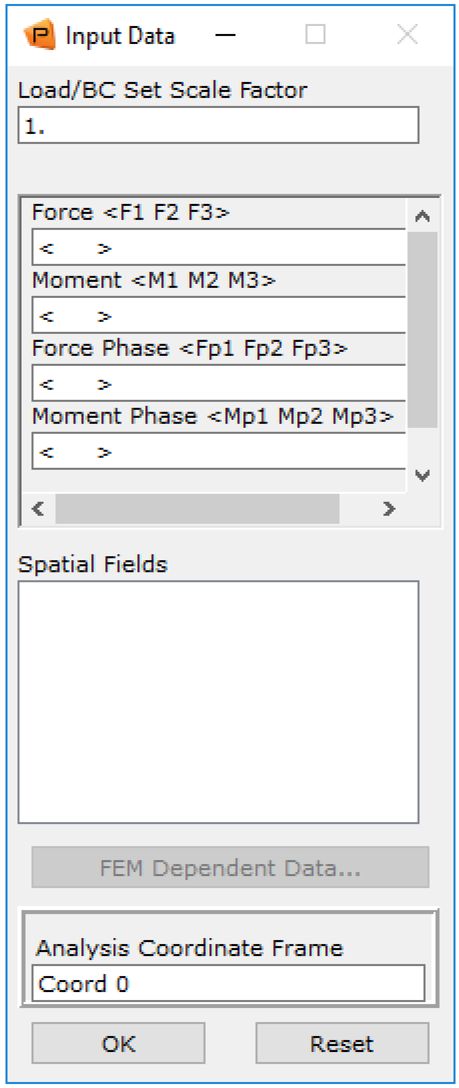


Figure 4: Typical PATRAN Input dialog box for forces and moments (torques)

Figure 5 is a Finite Element model in PATRAN. The finite element of figure 5 consists of 10 “Rod” elements that each are 1 inch long. The cross sectional area of all the “Rod” elements are circular with a radius of 0.25 inch. The material is assumed to be steel with its Modulus of Elasticity being $30x10^6$ PSI and its Poisson’s ratio as 0.3.

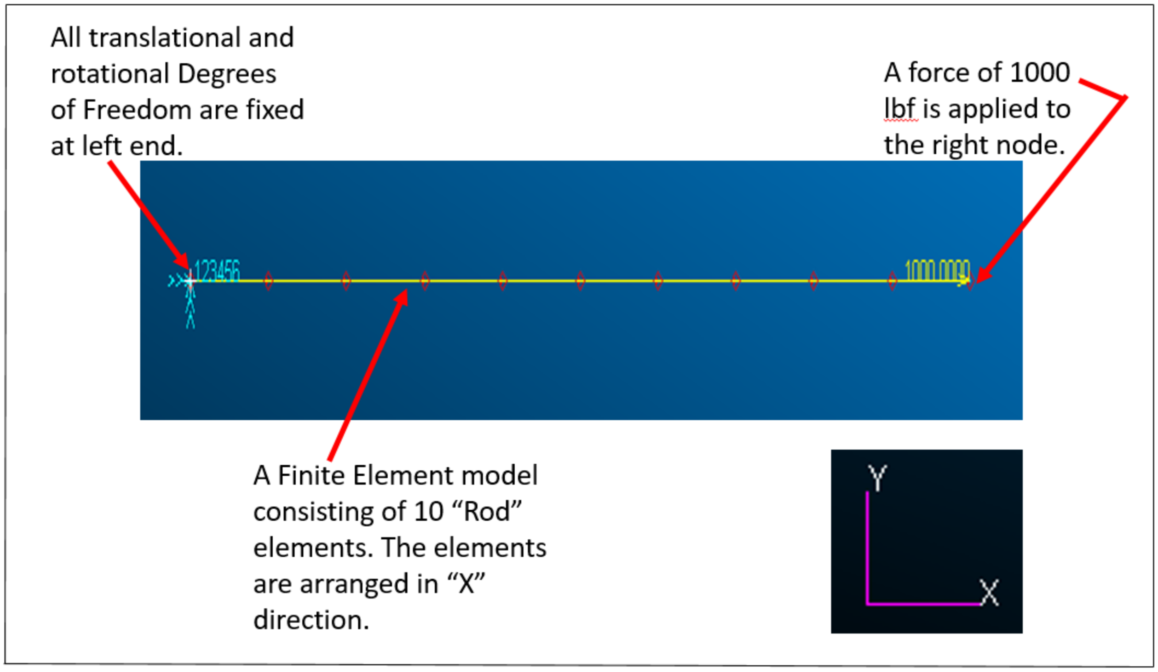


Figure 5: Finite Element Model of a rod in PATRAN

The stress results determined by NASTRAN/PATRAN for the model of figure 5 is shown in figure 6.

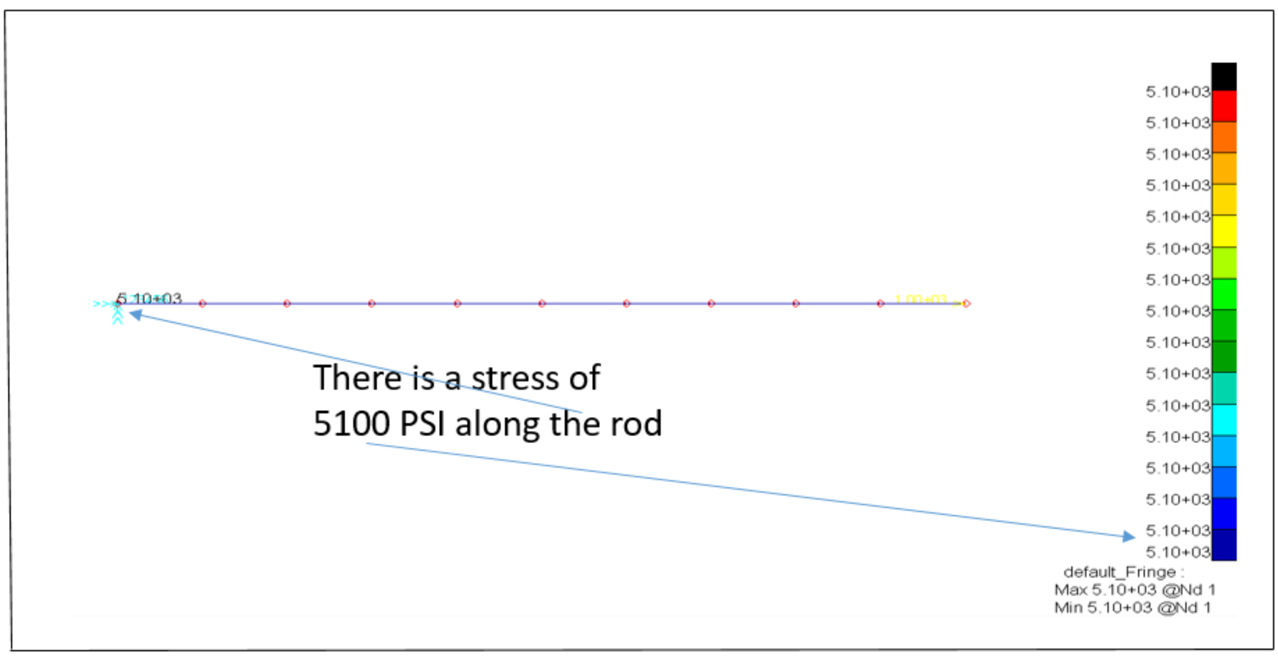


Figure 6: Stress obtained by the Finite Element technique

As shown in figure 6, the stress in the rod element as a result of the application of the force along its axis is equal to the force divided by the cross sectional area of the Rod.

Figure 7 is the same Finite Element model as figure 5, except that the force is being applied in a vertical direction which is not a Degree of Freedom for a “Rod” element. Figure 8 shows that the stresses for the model of figure 7 are “0”. The “0” stresses confirm that stresses are calculated only for the Degrees of Freedom of the element type used.

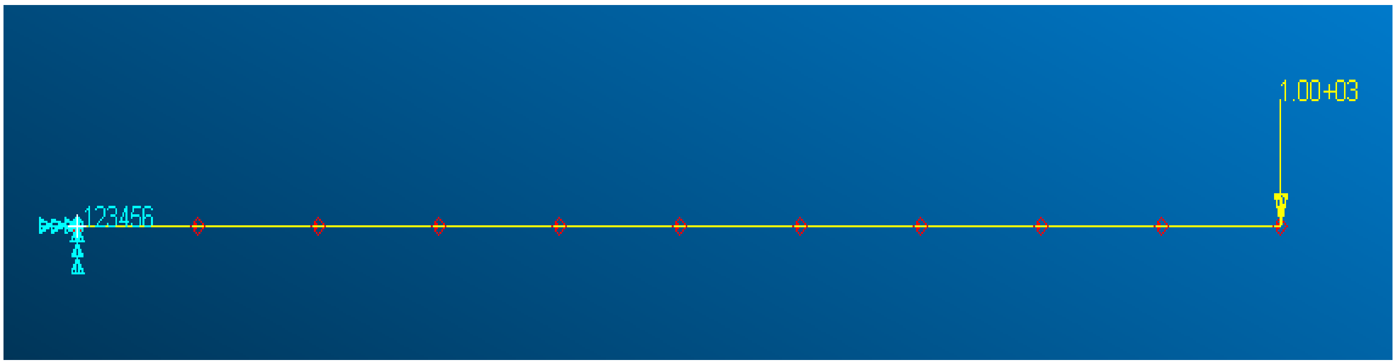


Figure 7: Finite element of “Rod” elements fixed at left end and vertical force applied at right end

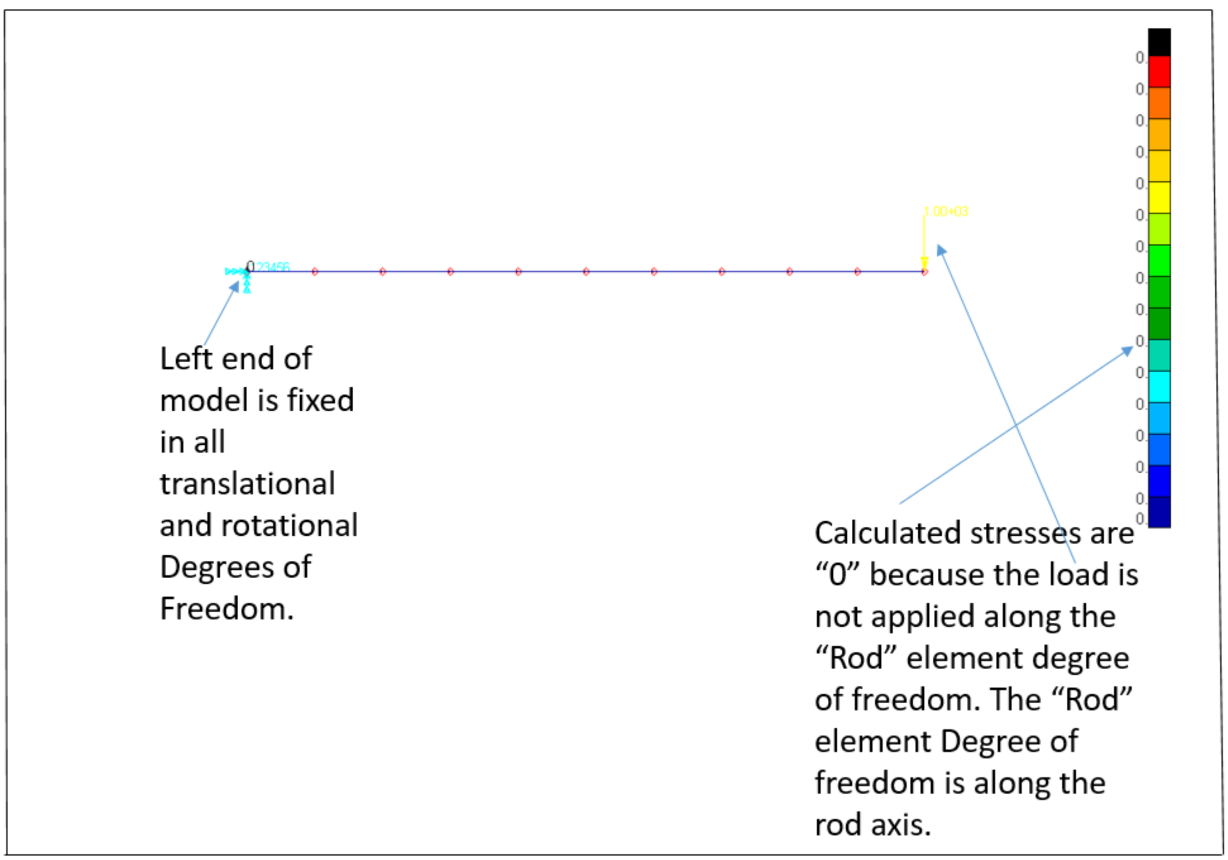


Figure 8: Stress results for the Finite Element model of figure 7

Figure 9 shows the Finite Element model where torque is applied to a "Rod" along its axial axis. Figure 10 is the resulting stresses calculated by NASTRAN/PATRAN for the model of figure 9.

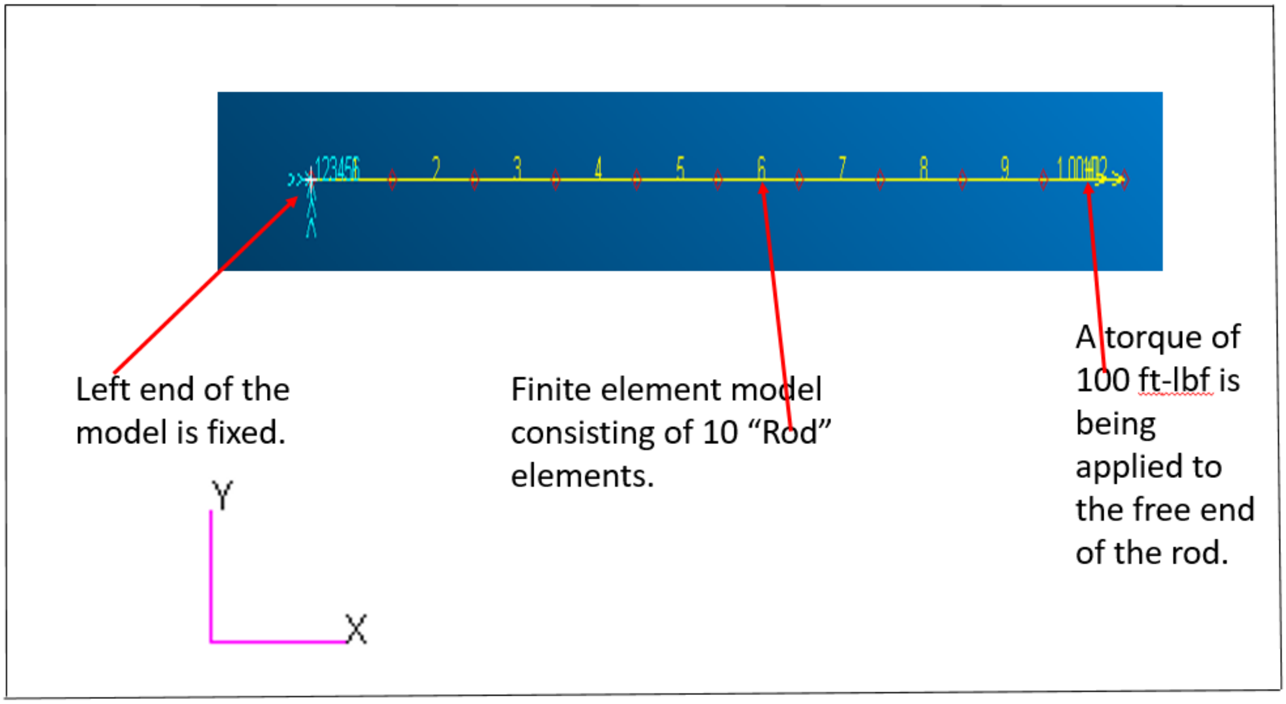


Figure 9: Finite element model of a "Rod" fixed at one end and subjected to a torque at the free end

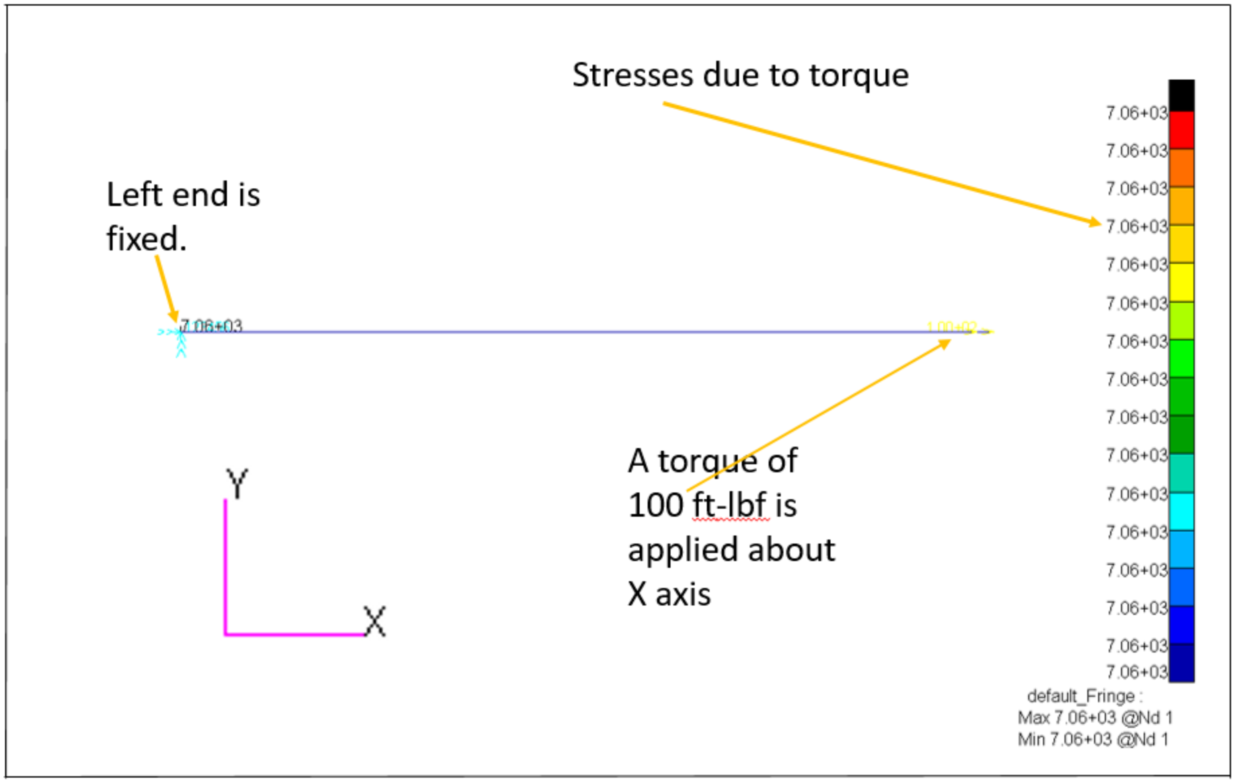


Figure 10: Stresses for the Finite Element model of figure 9

Figure 11 is the same Finite Element as figure 9, except that the torque is not applied around the axial axis and is instead applied around an axis that is perpendicular to the "Rod" element. Figure 12 is the resulting stresses for the model of figure 11, and as expected, the stress values are "0" because the Finite Element model only calculates stresses applied to the element degrees of freedom.

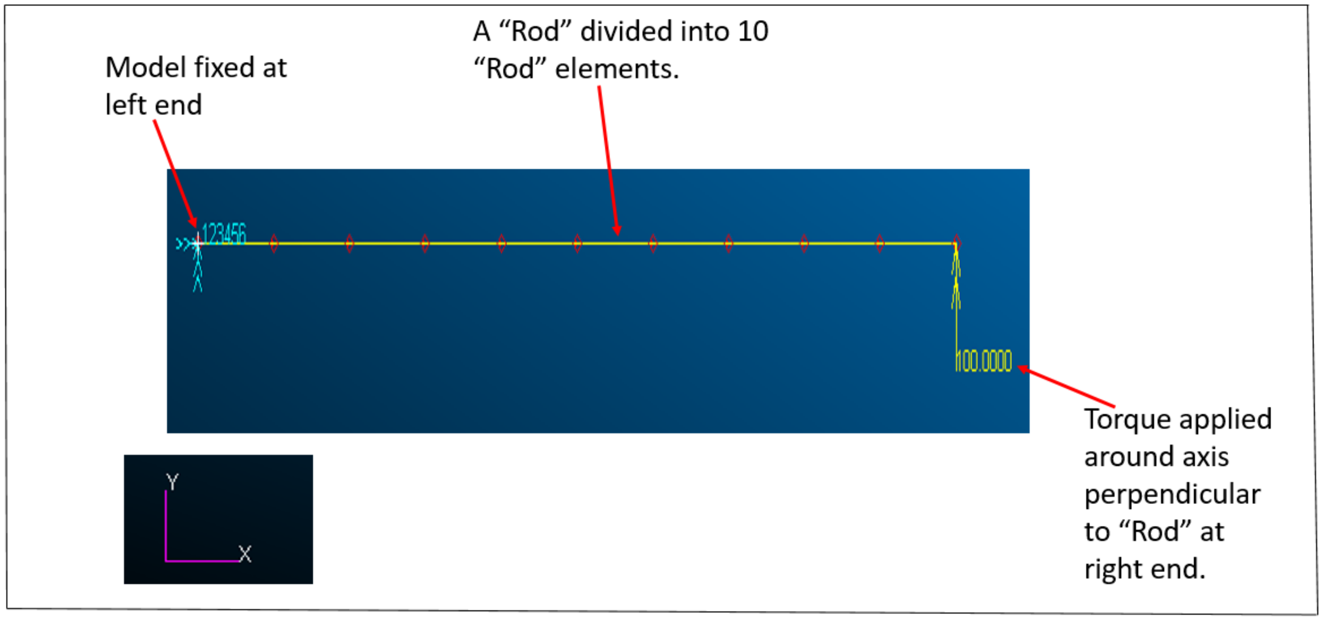


Figure 11: Finite element model of a Rod fixed at one end
and subjected to torque around an axis perpendicular to the rod at the free end

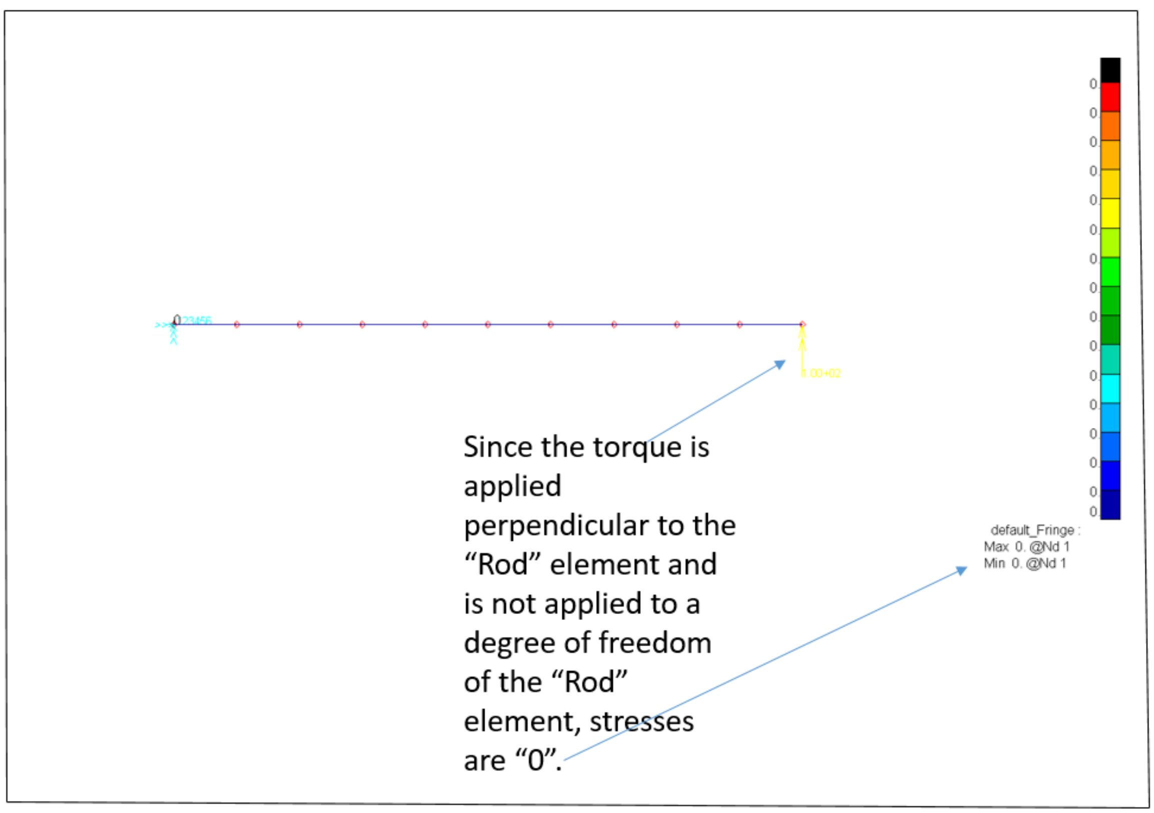


Figure 12: Stresses obtained for the model of figure 11

Discussion of "Beam" element

When defining a 'Beam" element, the dialog box of figure 13 appears in PATRAN. The initial input parameters are shown in figure 13,

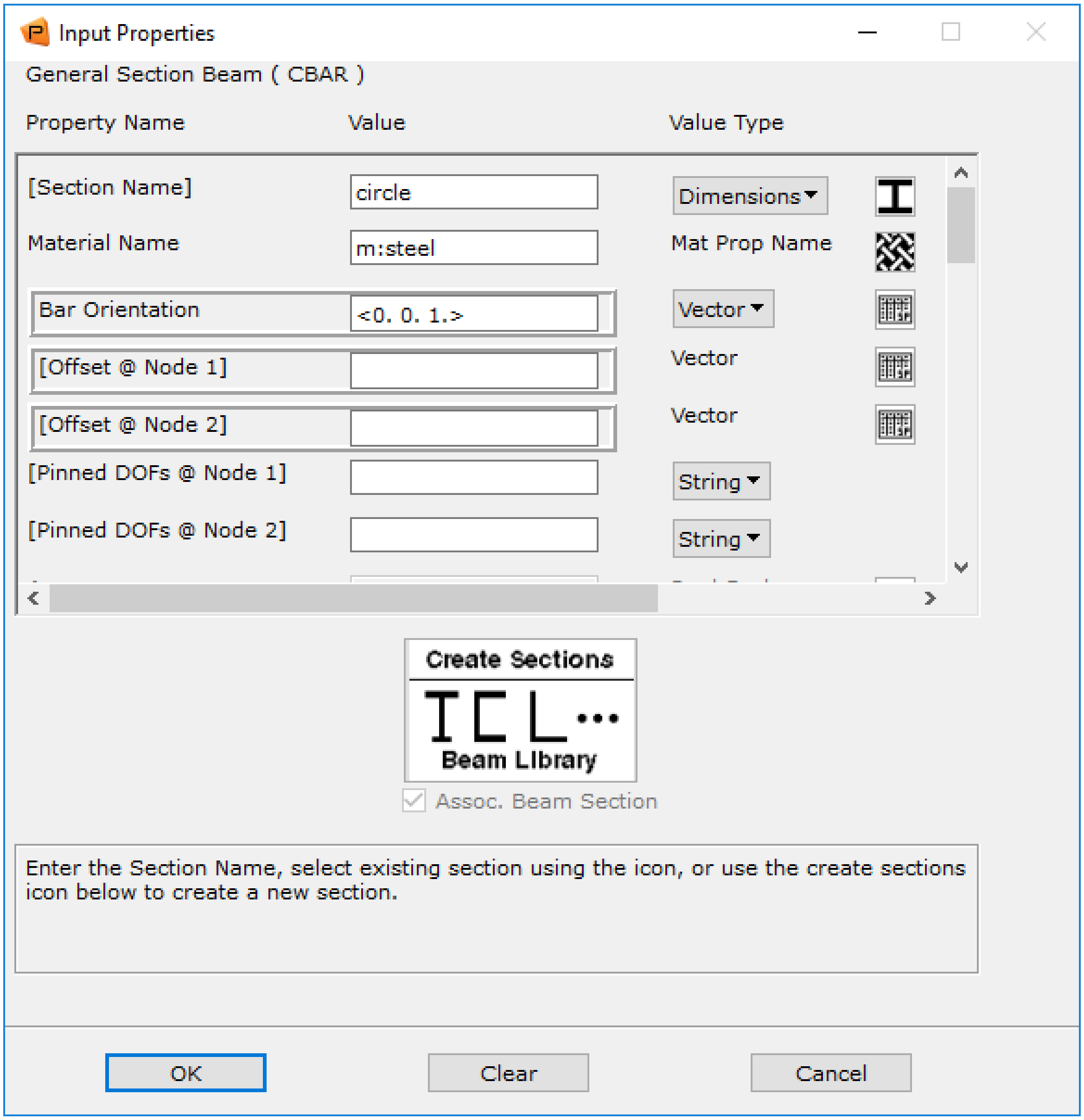


Figure 13: Dialog box for defining properties of Beam elements in PATRAN

The material input section is the same as the "Rod" element. The "Bar Orientation" section defines the element orientation. In the case of the example in this article, the goal is to model the geometry of the "Rod" element in the model of figure 5 as a "Beam" element. To achieve the desired orientation, <0 0 1> must be put in the "Bar Orientation" section. Doing so will align the minimum principal axis of the circular beam cross section with Z axis of the model. The model in effect becomes as shown in figure 14.

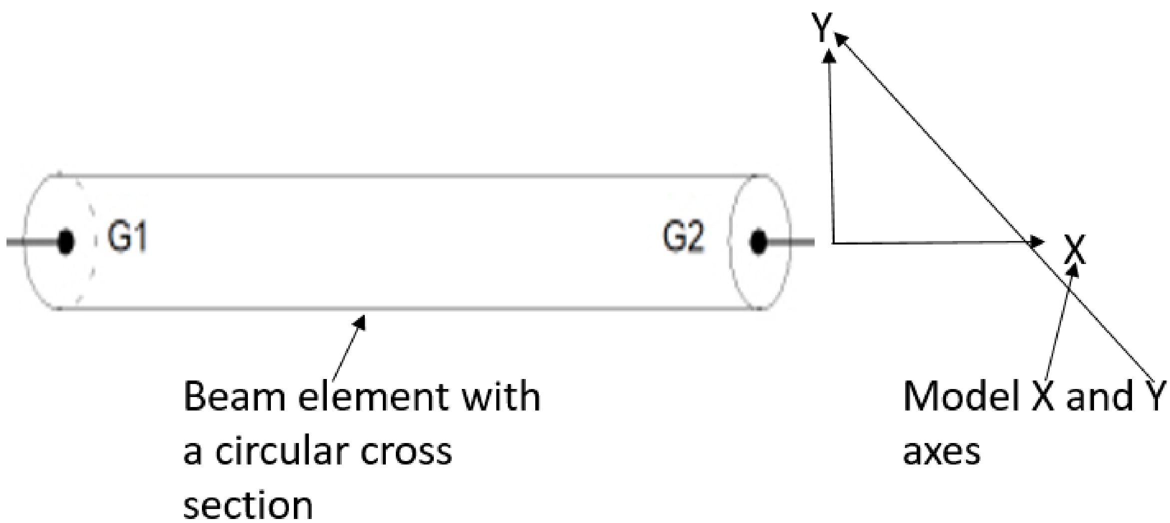


Figure 14: Illustration of a “Beam” element with a circular cross section

The cross sectional area of the beam element is defined by clicking on the “Beam Library” of the dialog box in figure 13. Since the cross sectional area of the beam Element is a circle of 0.25 inch, the input parameters are shown in figure 15.

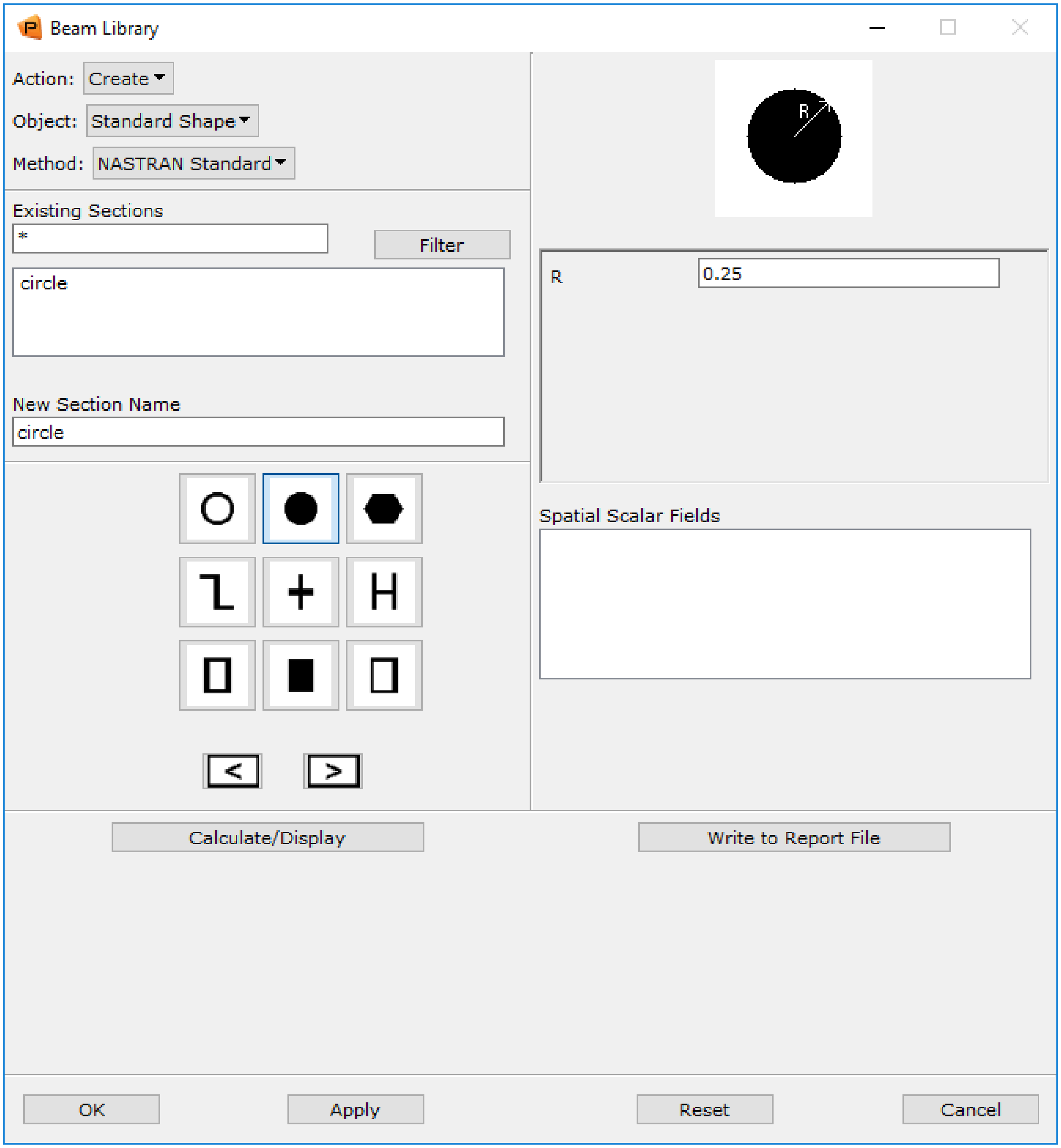


Figure 15: Inputting the geometry of the beam element

Figure 16 is the stress contour that is obtained for the model of figure 5 consisting of beam elements instead of Rod elements. As expected, because Beam elements and Rod elements have the same axial degrees of freedom, the results are the same as those shown in figure 6 within round-off errors.

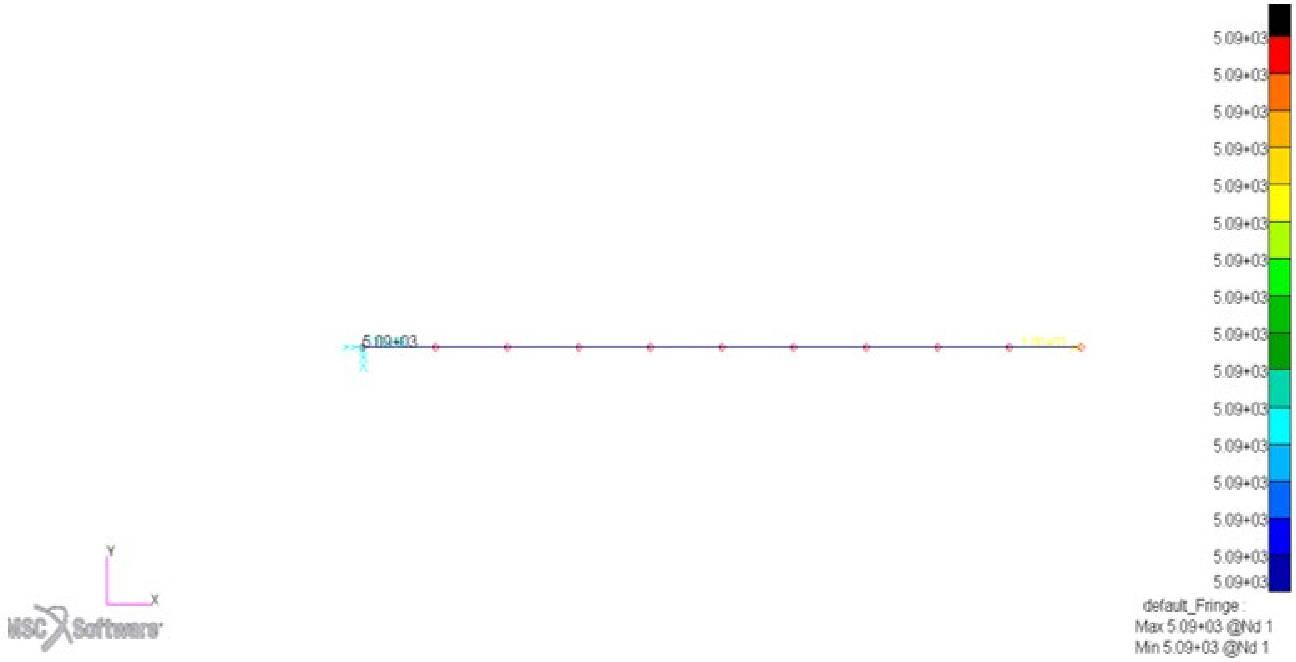


Figure 16: Stress contour of the structure with an axial force when the structure is modeled with Beam elements

Figure 17 is the stress contour for the model in figure 7 except that Beam elements are used in place of Rod elements. Since Beam elements have a degree of freedom perpendicular to the element, stresses are calculated (contrary to figure 8 which showed no stresses for the same exact conditions modeled with Rod elements).

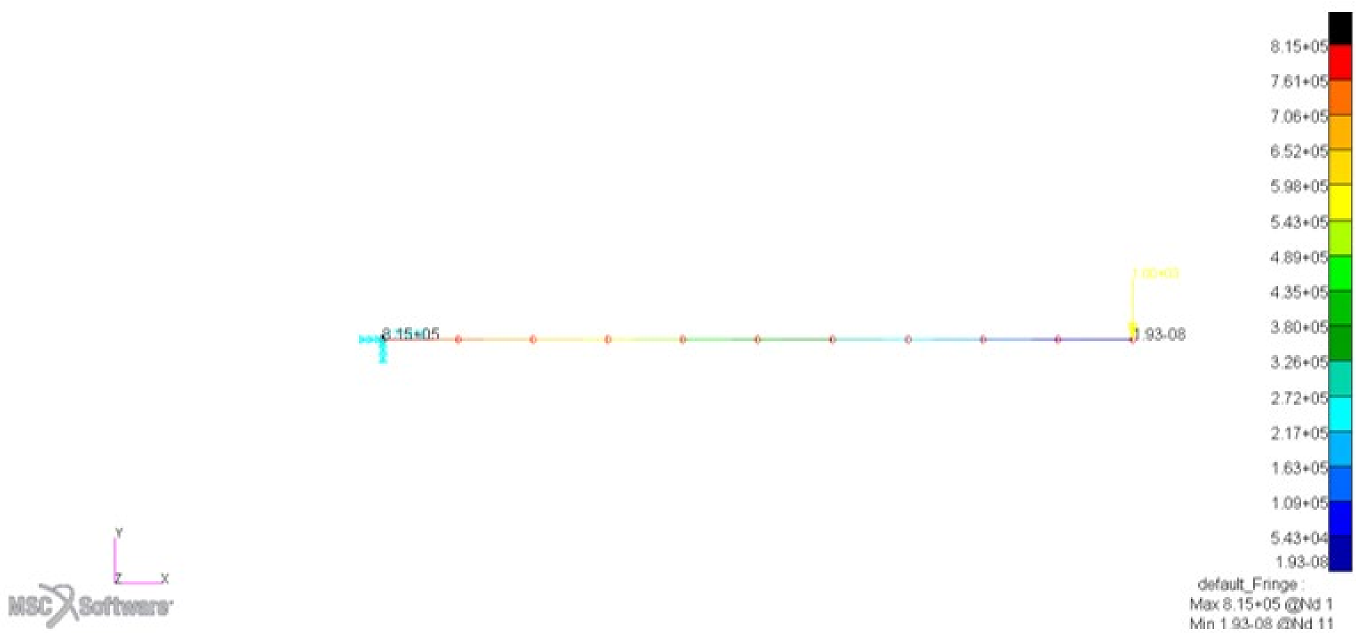


Figure 17: Stress results for Finite element model of figure 7 modeled with Beam elements

Figure 18 is the stress contour for the model in figure 9 except that Beam elements are used in place of Rod elements. Since Beam elements do not have a torsional degree of freedom, contrary to the model of figure 9, stresses are 0.

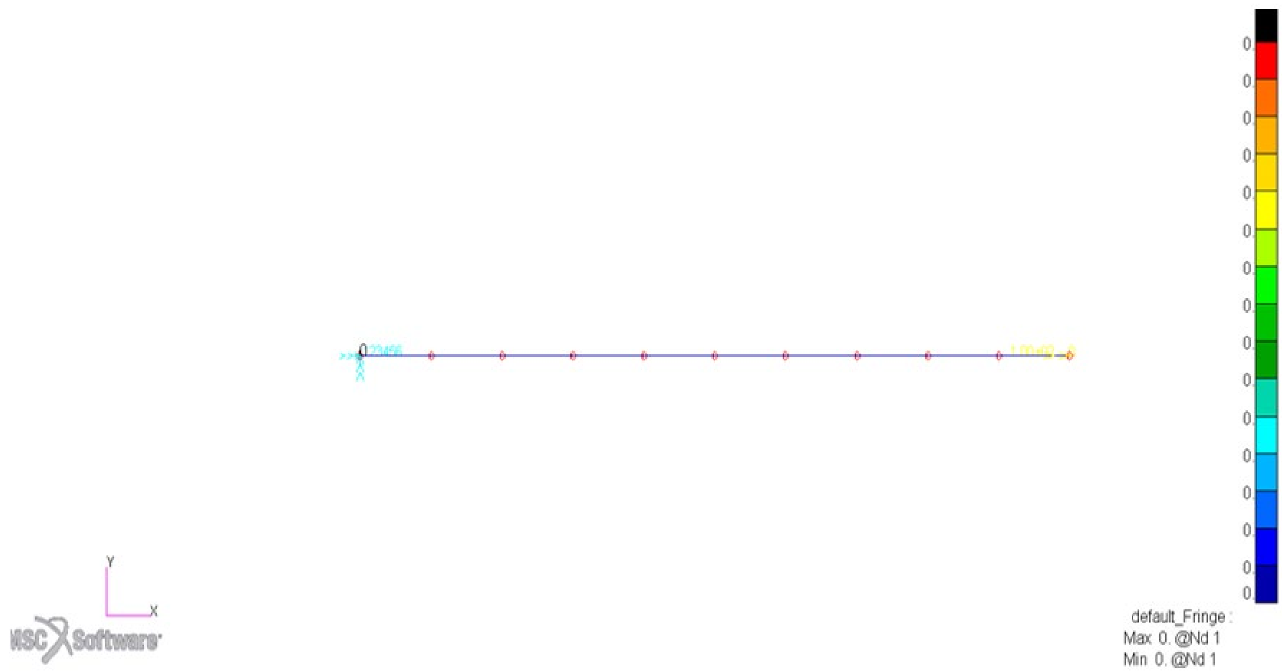


Figure 18: Stress results for Finite element model of figure 9 modeled with Beam elements

Summary and conclusion:

It is demonstrated through the use of examples that the Degree of Freedom of various elements must be considered before using the elements. It is shown that "Rod" elements have displacement degrees of freedom along their axis, but not perpendicular to their axis. It is also shown that "Rod" elements have torsional degrees of freedom around their axial axis but not around any other axis. It is shown that "Bar" elements have displacement degrees of freedom along their axis and perpendicular to their axis. However, "Bar" elements have no torsional degrees of freedom.

References:

[1]. MSC Nastran 2021.3 – Online Help section.
[2]. Statics and Strength of Materials, by Cheng.
[3]. PATRAN verification manual.