ANSYS: Interpretation of Edge Loads

By Kieran Velon, Intern at RMA

Introduction

Prior to checking the stresses in a thin cylindrical sleeve which is to be fitted with an interference fit to a rotating shaft, verification studies of the ANSYS software [1] was undertaken. The purpose of these studies was to establish that the software could model accurately such a problem and to provide an indication as to the level of mesh refinement required. The sleeve is an axisymmetric structure which is thin in the axial direction and can, therefore, be reasonably idealised as a plane stress model. There are two such elements in ANSYS, namely, the two-dimensional continuum element and the shell element. Both elements can model plane stress or plate-membrane behaviour. The shell element does of course include plate-bending capability but this part of the complete functionality of the shell element is not required for the sleeve and will not be invoked by the in-plane force loading that will be applied to it.

Whilst the stresses in the actual sleeve might well be three-dimensional in nature particularly immediately adjacent to the shaft/sleeve interface, a good first approximation to the stresses can be obtained using a plane stress idealisation and, so, a solid model of one quadrant of the sleeve was constructed in ANSYS with boundary conditions applied to the solid model. This model could then be meshed either with PLANE183 or SHELL281 elements. Higher-order elements were chosen based on the complexity of the known theoretical solution to this problem as given by Lamé. When the results were compared it was observed that the stresses were different not only by a few percentage points but by orders of magnitude. Further exploration led to the conclusion that ANSYS interprets edges loads differently between the two element types. This short technical note explains this issue.

Benchmark Problem

Lamé presented the theoretical equations for structures with axisymmetric geometry, materials, supports and loading. Values for the hoop and radial stresses may be determined at a given radius, r, using the expressions presented in Figure 1.

$$\text{hoop stress } \sigma_H = A + \frac{B}{r^2}$$

$$\text{radial stress } \sigma_r = A - \frac{B}{r^2}$$

Figure 1: Lame's equations for the hoop and radial stresses occurring in a cylindrical sleeve [2].

The coefficients A and B are dependent on the boundary conditions applied to the model. For the example used in this technical note, the boundary conditions are both of the static type, i.e., applied pressures, and by consideration of equilibrium at the boundaries, the radial stresses at the boundaries will be equal and opposite to the pressure assuming a convention where a positive pressure acts onto the surface.

Using Lame's analytical approach, applying a 10MPa pressure, P_i, on the thin cylindrical sleeve of dimensions, R_i = 25mm and R_o = 50mm, yields a compressive radial stress of 10MPa at R_i – see Figure 2. Naturally, since P_o =0, the radial stress at R_o is null. Values obtained for the hoop stress at R_o and R_i were 6.68 MPa and 16.67 MPa respectively, both tensile in nature, as one would expect given that the average hoop stress must be tensile.

Constants	Value
A	3.33MPa
B	8.33kN

Figure 2: Constants derived from the Lamé equations.

Material Constants	Value
E	200GPa
ν	0.3

Figure 3: Youngs modulus and Poisson's ratio of cylindrical sleeve.

Finite Element Model

The model used is shown in Figure 2. It represents a symmetric quadrant of the complete sleeve and requires the symmetry constraints shown to be applied to the cut-faces.

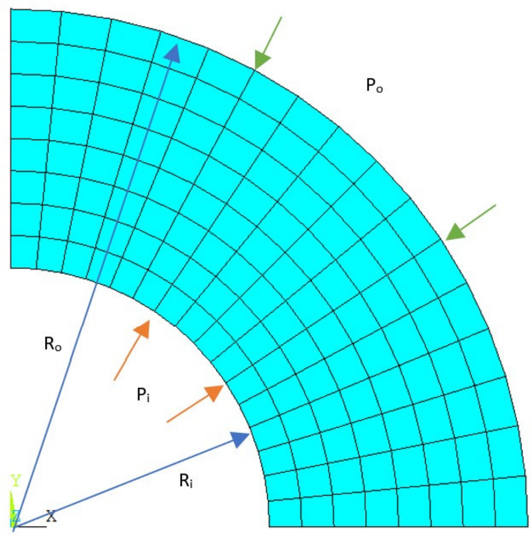


Figure 4: F.E. model of cylinder which makes use of symmetry of the geometry.

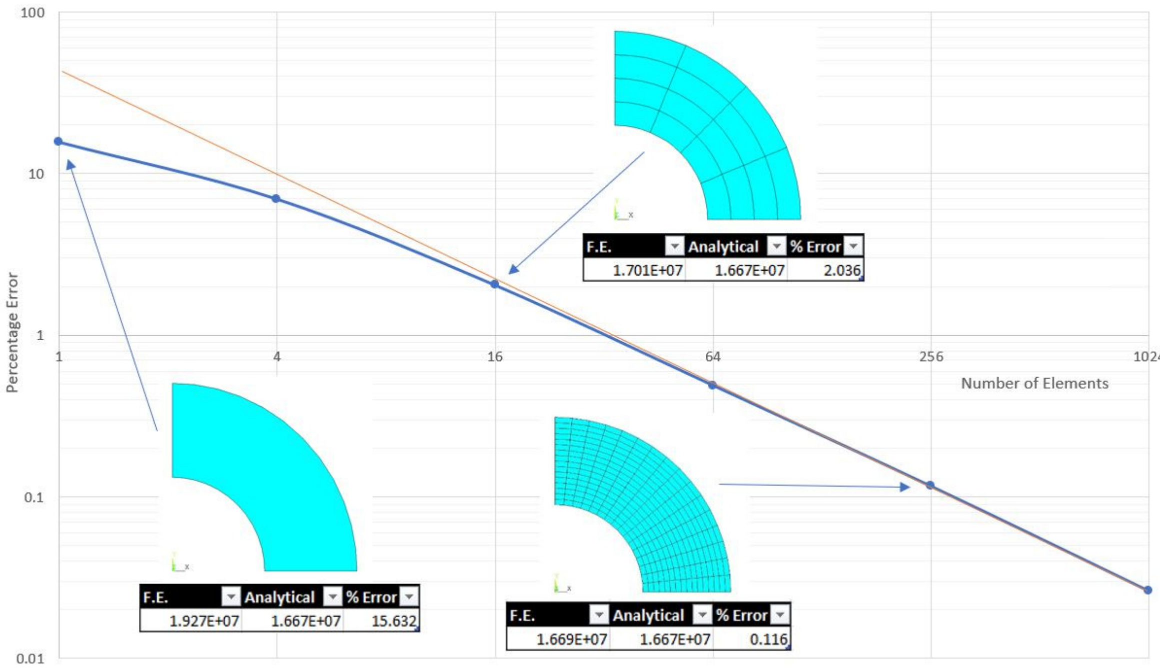
Both hoop and radial stresses are expected to vary with the inverse square of the radius across the cylinder wall, hence the use of second-order elements was felt to be preferable to the equivalent lower-order elements. The inherent properties of second order elements allows them to more closely model the hoop and radial stress distribution given by Lame's equations, compared to linear elements, and also capture the circular geometry of the sleeve.

The hoop stress at the inner radius is the maximum stress in the model and is likely to be the stress that an engineer would judge the viability of his/her design. For example, it might be the case that this stress should not exceed some fraction of the yield stress. The value of the fraction (which will generally be less than unity) will generally have been laid down in the design specification which in turn may have come from an appropriate code of practice, e.g., the Eurocode.

Mesh Convergence Study

Given that the FE model should converge to the theoretical solution with increasing mesh refinement the question the engineer must ask themselves is how accurate does my result need to be? Ask two different engineers and they will probably come up with two values, for example, 5% and 10%. These values will likely as not be based on their experience as to the accuracy of the material properties and loads used in the FE model. Whilst one might adopt the same level of accuracy for the FE model, it might be prudent not to add unnecessarily to the general degree of uncertainty by using an inaccurate FE model. As such, an accuracy level of 1% for the FE model will be adopted.

A mesh refinement study was conducted to determine if the mesh generated in Figure 2 contained the necessary number of elements required to achieve the desired 1% accuracy and the results are presented in Figure 5.



The values graphed in Figure 5 represent the percentage difference between the hoop stresses determined using the analytical and FE models (PLANE183) at r= 0.025m.

Figure 5: Percentage error between FE and analytical model.

Figure 5 indicates that an error of less than 1% is achieved using a mesh of 8x8=64 elements. The mesh used, see Figure 4, is comprised of 16x8=124 elements, making it more accurate than a mesh of 64 elements. Therefore, the mesh used is deemed satisfactory both for engineering purposes and for further exploration in this technical note.

Finite Element Results

The FE results for the model shown in Figure 4 using both planar and shell elements are shown in Figure 6 in the form of contour plots of the hoop and radial stresses.

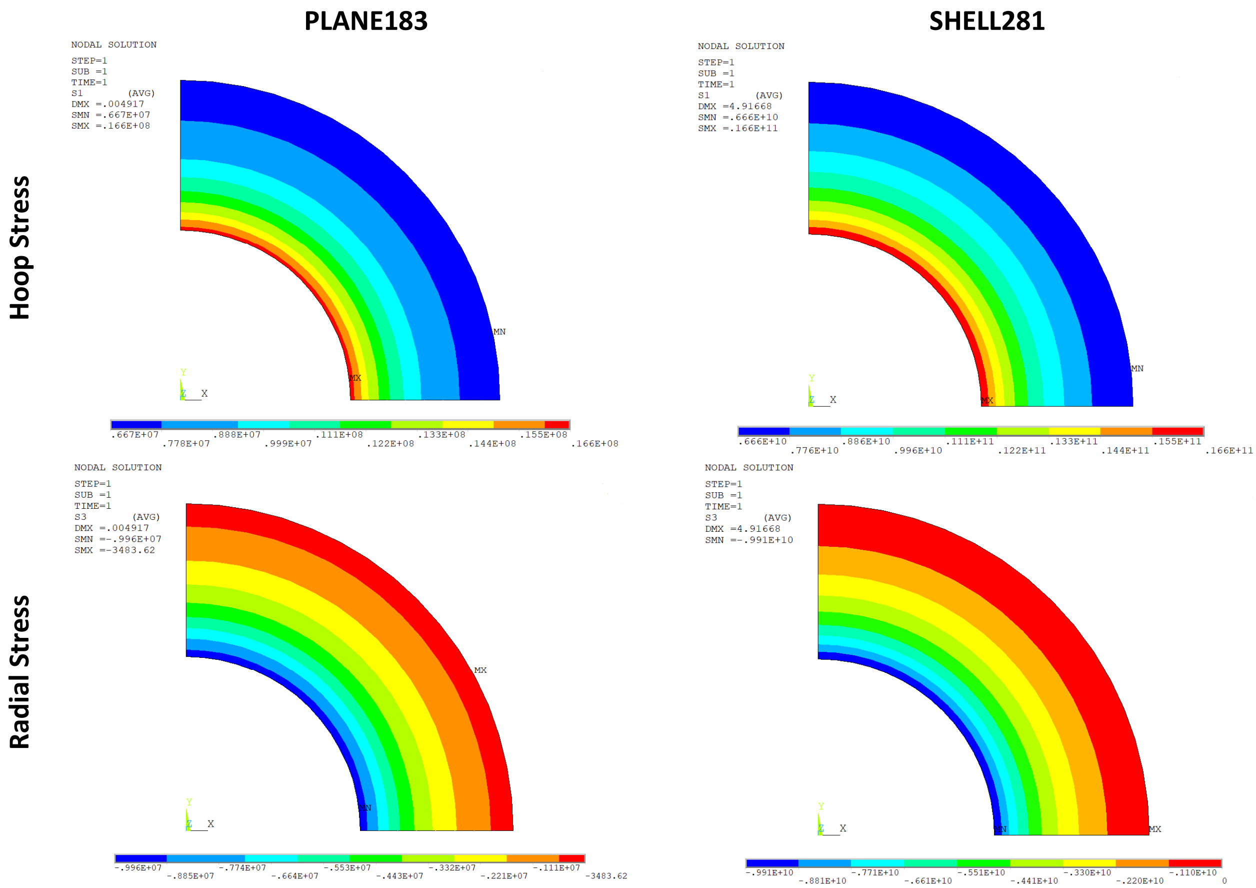


Figure 6: Stress contours for the two models

The hoop and radial stresses are plotted as a function of radius in Figure 7.

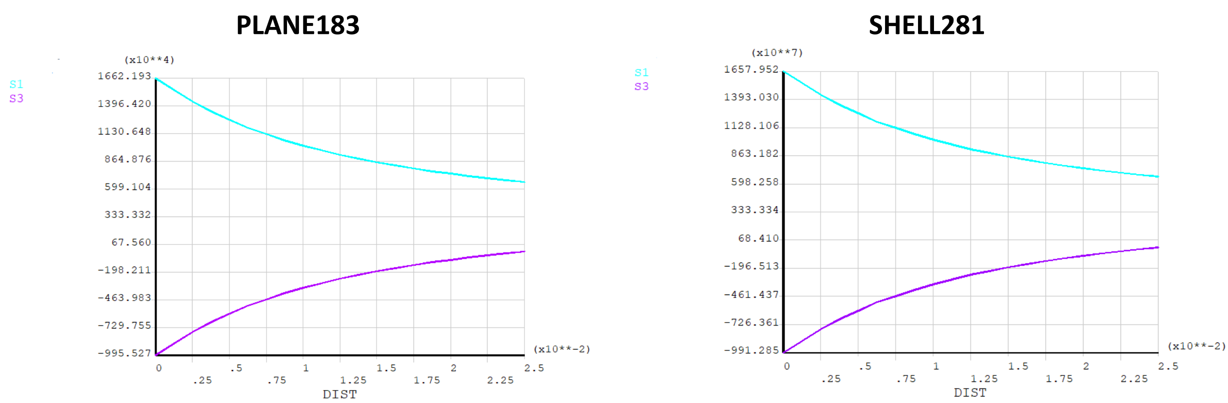


Figure 7: Hoop and radial stress variation.

Evident from Figures 6 and 7, the hoop and radial stresses obtained using PLANE183 reflected analytical values derived from Lame’s equations. However, hoop and radial stresses calculated using element SHELL281 were almost exactly three orders of magnitude too large.

Thus, whereas one would expect broadly similar results from the shell and membrane elements, this does not appear to be the case. To explore the reason for this difference, additional analyses were conducted using different axial thicknesses for the sleeve. The results are presented in Figure 8. It is seen in this figure that whilst the plane element recovers the expected stress, the shell element does not and that the stress is inversely proportional to the thickness. This unusual phenomenon concerning shell elements arises both in ANSYS Workbench and ANSYS Classic [1], and for both higher and lower-order elements.

Thickness (m)	PLANE183	SHELL281
1	-10	-1x10^1
0.1	-10	-1x10^2
0.01	-10	-1x10^3
0.001	-10	-1x10^4

Figure 8: Radial stress, [MPa], at inner radius for p=10MPa

An Explanation of the Issue

What appears to be happening here is that ANSYS interprets line pressures in a different way depending on the element type being used. Of course, a 'line pressure' is a misnomer since a line has no area. However, when using plane elements ANSYS appears use the membrane thickness to define the area whereas with the shell element it does not. So, for shell elements, the line pressure is interpreted as a line force, i.e., a force uniformly distributed along a line with N/m units. If the user wants the line pressure to represent a true pressure over the area identified by the line then he/she will need to multiply the value to be input to ANSYS by the shell thickness.

Closure

The purpose of this technical note was to outline possible boundary condition misinterpretations which may arise when applying pressure to the sides of planar or shell elements in ANSYS. As discussed, different interpretations are made by ANSYS depending on the element type used. For the planar element the line pressure is interpreted as a true pressure with units N/m^2, whereas for the shell element it is interpreted as a force per unit length with units N/m. One way in which such a misinterpretation could be avoided would be for the SFL command to have an option to define how the load should be interpreted by the software.

References

[1] ANSYS Mechanical 2019 R3, ANSYS Academic Teaching Introductory License.

[2] E.J. Hearn, "Thick Cylinders", in Mechanics of Materials 2nd Ed. City of Birmingham Polytechnic: Pergamon Press, 1989, section 10, pp. 215.