International Journal for Ignited Minds (IJIMIINDS)

Finite Element Modeling for Engineering Analysis-Solid Mechanics Applications

Archana shinde[a], Dr H. V. Lakshminarayana[b] & Thejeswini M[c]

[a]PG Scholar, Dayananda Sagar College of Engineering, India.
archana.pawar138@gmail.com
[b]Prof, Department of mechanical engineering, Dayananda Sagar College of Engineering, India.
hvllakshminarayana@gmail.com
[c]Asst prof, Dayananda Sagar College of Engineering, India.
thej.dsce@gmail.com

ABSTRACT

Stress plays important role in the design of structures/machines. Prior stress estimation and behavior helps in designing durable structures. Due to the advances in the computer based finite element software the design simulation is fast replacing the old type of prototype built up and testing which is costly and time consuming. This paper presents the results of four common engineering problems, a circular plate with an elliptical cutout subjected to external pressure, a annular disc subjected to thermal loading, aero engine compressor disk subjected to pressure load and a curved cantilever beam subjected to moment.

Keywords *– Elasto-Plastic, Compressor, Reinforcement, Circular Annulus.*

1. INTRODUCTION

The most common engineering problems are considered for finding the stress behavior in the structures. The problems include a circulate plate with elliptical cutout subjected to external load, an annular disc subjected to harmonic type thermal load, an aero engine compressor disc subjected to external load, a curved beam subjected to a structural load. All the geometries are built using Ansys pre-processor. Albert Kaufman[1] correlated theoretical stresses and strains in a pressure vessel having a reinforced opening with experimental data obtained for progressive stages of elastic-plastic deformation up to the point of failure. Elastic-plastic stresses are used with the true ultimate tensile strength of the shell material to give a reasonably accurate prediction of the burst pressure.

C. R. Calladine [2] applied plastic design approach for the design of reinforcement for openings in thin spherical pressure vessels. The essence of the approach is to adjust the thickness and shape of the vessel in the vicinity of the opening so that the full limit pressure of the vessel may be carried with relatively little bending action. It is concluded that pad-reinforced nozzle represents major advantage over other forms of reinforcement is that its general form enables the forces on the nozzle and its surround due to pressure loading to be carried almost entirely by membrane stresses; and this in turn results in low stress concentration effects.

R. Kitching [3] compared the experimental limit pressures for cylindrical shells with unreinforced openings for different shapes and sizes. It was concluded that the limit pressures similar for openings of the same overall dimensions and is not dependent of the shape.

M. N. Bapurao [4] studied stresses around a small elliptic hole in an infinitely long circular cylindrical shell subjected to torsion. It was concluded that the effect of curvature for a given β is maximum if the hole is circular. As the hole is made more and more slender keeping β constant, the perturbation stress tend to decrease except for the crack case ($\lambda = 1$) when they become singular at the crack tip.

A. J. Durelii [5] presented work on the stresses concentration in a ribbed cylindrical shell with a reinforced circular hole subjected to internal pressure, by several experimental methods and the results obtained were compared with those corresponding to a non-reinforced hole in a ribbed and un-ribbed shell and also to a reinforced hole in an un-ribbed shell. From the result it was found that the maximum value of hoop stress, and longitudinal stress, in shells always occurred at the points $\theta = 0°$ and $\theta = 90°$, respectively, along the edge of the hole, θ being the angle measured clockwise from the longitudinal axis of the hole.

This paper presents significant results of stress analysis of
A circular plate with an elliptical cutout subjected to external pressure,

An aero engine compressor disk subjected to pressure load and spinning
Annular Disc subject to harmonic type thermal loading and
A curved cantilever beam subjected to moment.

2. FINITE ELEMENT MODELLING

A Circular plate with an elliptical cutout subjected to external pressure load.

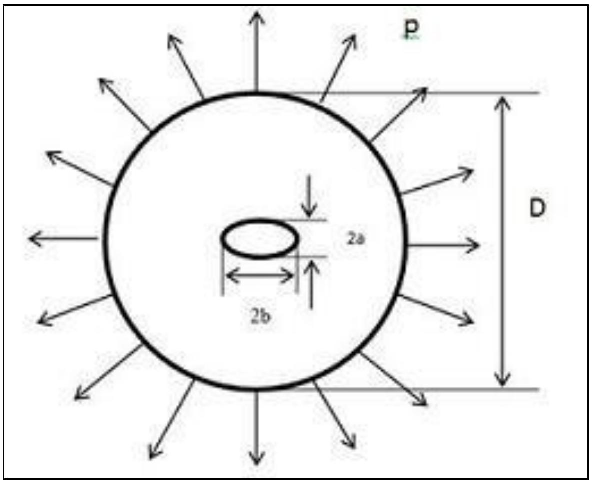

Fig 1: Circular plate with an elliptical cut-out subjected to external pressure load.

The problem is illustrated in fig 1 is a steel circular plate with an elliptical cutout subjected to external pressure load. The physical parameters defining the problem are D=200mm, t=1mm, a=10mm, b=20mm.

The finite element model shown in fig 2 was developed using PLANE42 element in ANSYS software. PLANE42 is an iso-parametric plane element, quadrilateral in shape, quadratic in order with four nodes and two engineering degrees of freedom at each node. The graded mesh consists of 400 elements.

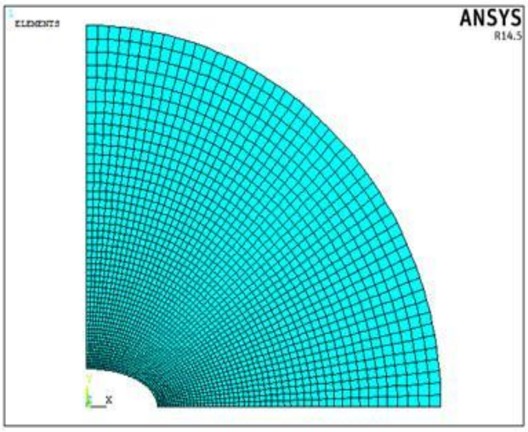

Fig 2: Finite element model of circular plate with an elliptical cutout subjected to external pressure load.

An aero engine compressor disk subjected to pressure load and spinning

The problem is illustrated in fig 3 and fig 4 shows half symmetrical problem with symmetric boundary conditions. A pressure load of 85000 PSI is applied on the outer geometry which is applied on the outer line. The problem is modeled using steel properties.

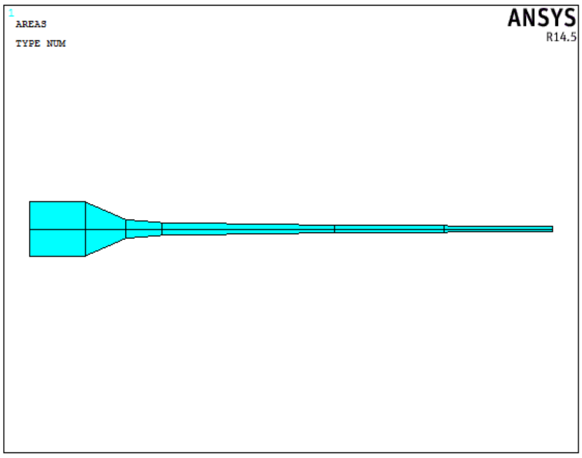

Fig 3: An aero engine compressor disk subjected to pressure load.

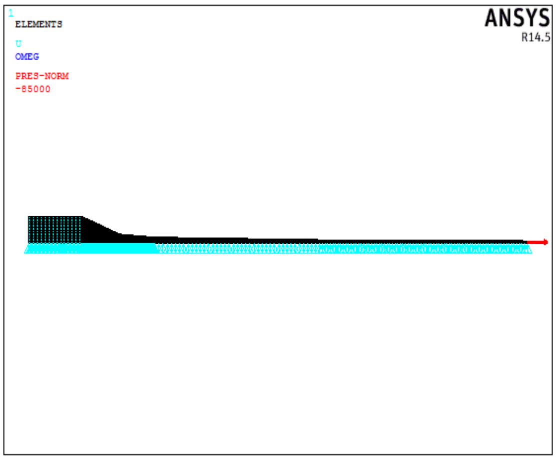

Fig 4: Finite element model of an aero engine compressor disk.

Annular Disc with Thermal Load:

Analysing the stress condition of the annular disc under harmonic type thermal load across the geometry which varies from zero at the inner and outer boundaries, maximum at the mean diameter. The dimensions of the disc are 200mm (2a) outer diameter with 40mm (2b) inner diameter.

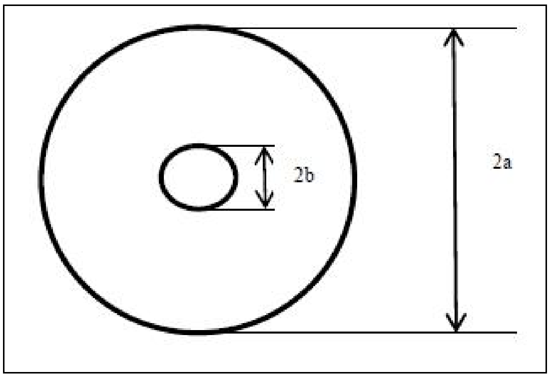

Fig 5: Annular Disc under Thermal Loading

A curved cantilever beam subjected to moment

The problem shown in fig 6 is a curved cantilever beam subjected to moment. The parameters defining the problem are Outer radius, r1=100mm, Inner radius, r2=50mm, width, w= 20mm. The finite element model developed using ANSYS is displayed in fig 7. The finite element model shown in fig 6 is developed using solid185 element which is hexahedral in shape, quadratic in order with eight nodes and three degrees of freedom at each node. The properties of steel are used in the analysis.

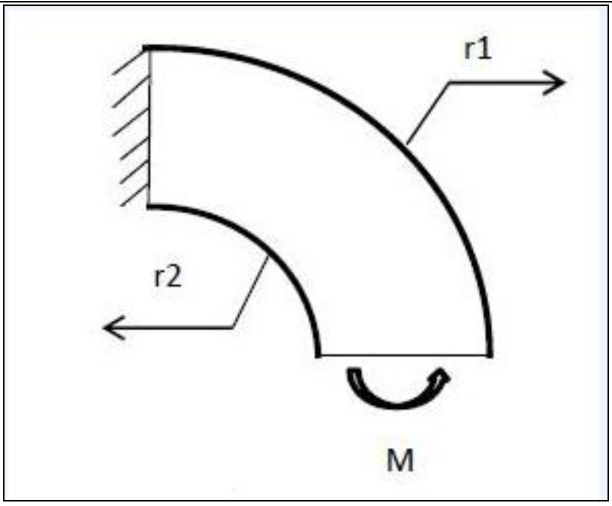

Fig 6: A curved cantilever beam subjected to moment load.

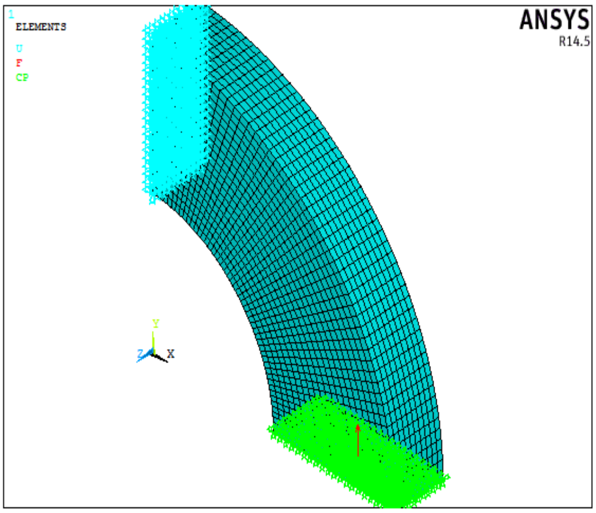

Fig. 7 Finite element model of a curved cantilever beam subjected to moment.

3. RESULTS AND DISCUSSION

A Circular plate with an elliptical cut-out subjected to external pressure load.

A benchmark problem circular plate with an elliptical cut out subjected to external pressure load is considered. The stress distribution in the circular plate with an elliptical cutout has been analysed here. ANSYS capability is been exploited here to Plot the graphs for stress variation along the boundary of the elliptical hole and across the geometry for finding the stress behavior.

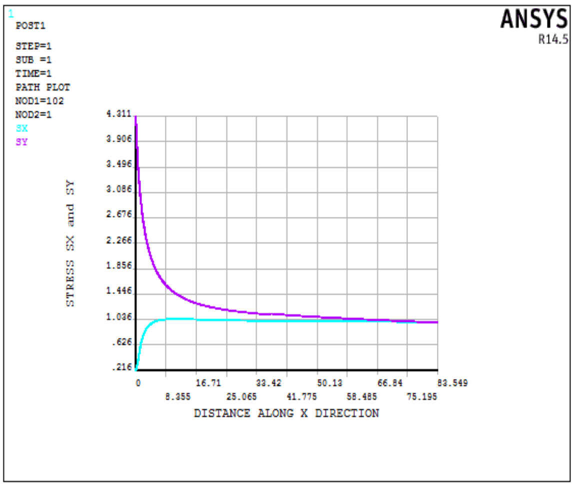

Fig 8: Stress SX and SY varying along x direction.

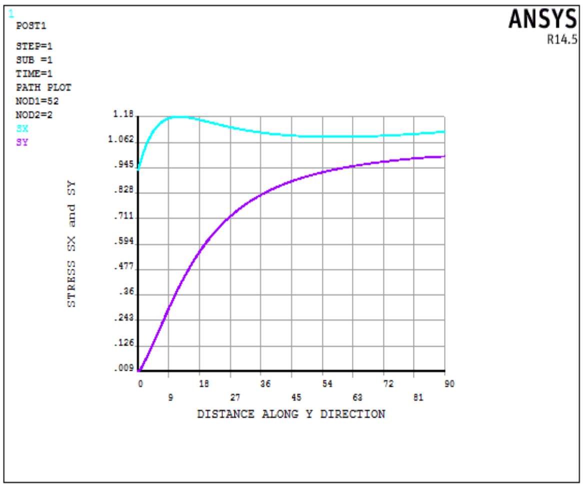

Fig 9: Stress SX and SY varying along y direction.

An aero engine compressor disk subjected to pressure load and spinning.

Stress analysis of an aero engine compressor disk subjected to pressure load and spinning is presented here. Due to the symmetry of the problem, the geometry is built in the axisymmetric domain. The geometry is split to ease map meshing of the geometry. Ansys mixed approach is used for meshing the problem. The FE Model presented above is refined enough to assure engineering accuracy. The analysis results are represented for displacement radial stress and hoop stress. Typical results shown represents maximum stress level of 93190.1PSI or 642.5Mpa. The graph showing the variation of SX and SY along the horizontal axis is shown in the fig 13.

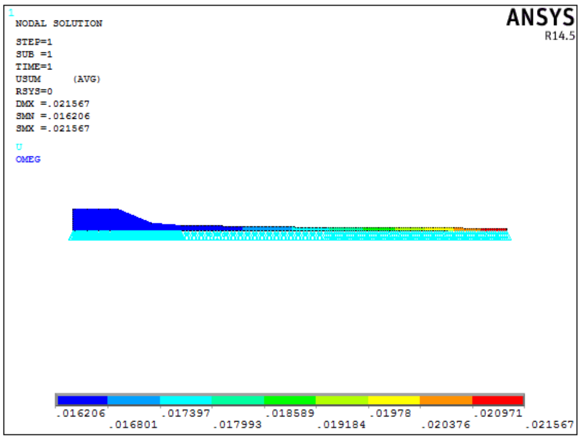

Fig 10: Plot showing the displacement vector sum.

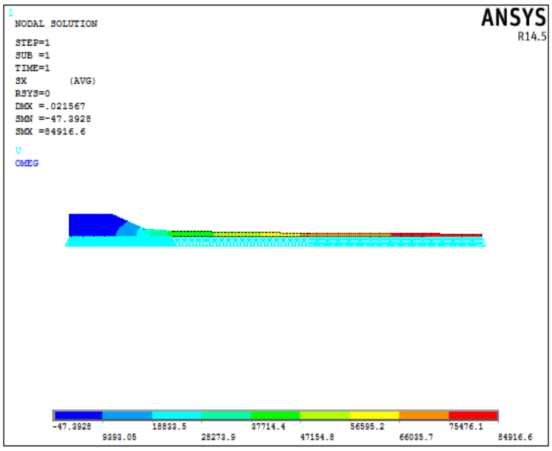

Fig 11: Radial Stress Plot

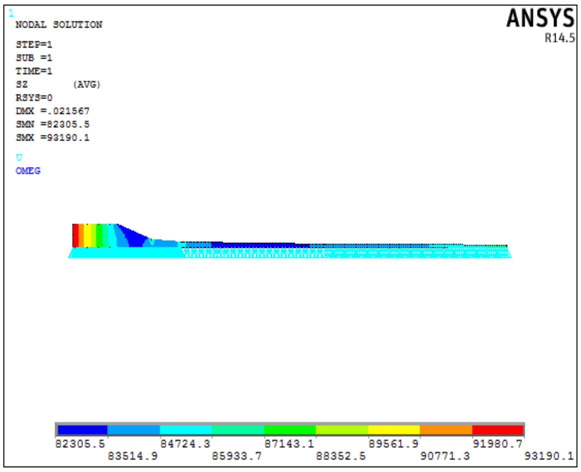

Fig 12: Hoop Stress Plot.

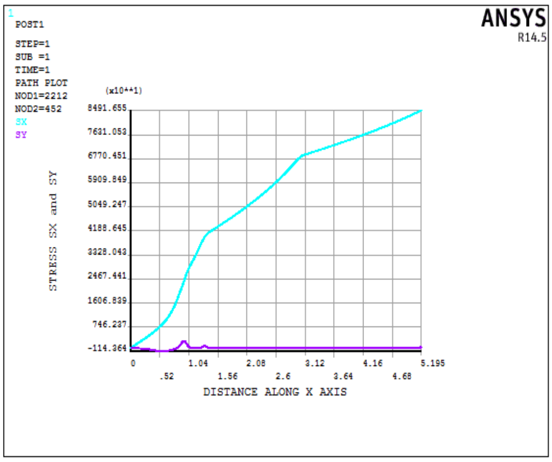

Fig 13: Stress SX and SY along the horizontal axis.

Annular Disc Subjected to Thermal Load

The results for Annular Disc Subjected to thermal load shows maximum temperature of 100 degrees at the center and Maximum vonmises stress of 16Mpa due to thermal load. Also the graphical variation of stress is also represented.

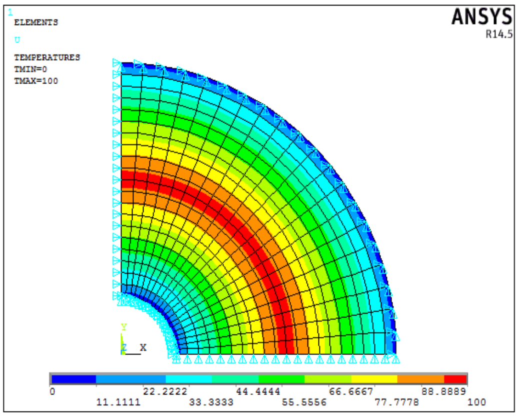

Fig 14: Temperature Plot.

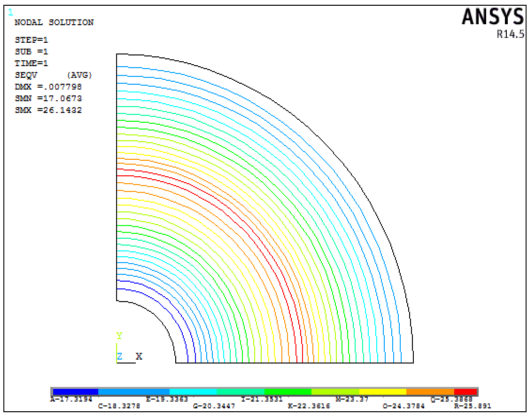

Fig 15: Von-Mises Stress Plot.

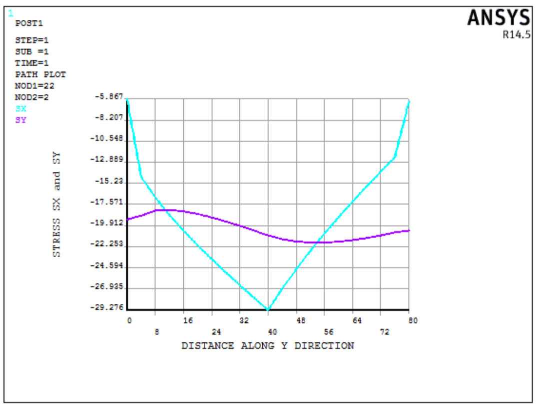

Fig 16: Hoop and Radial stress plot along the boundary.

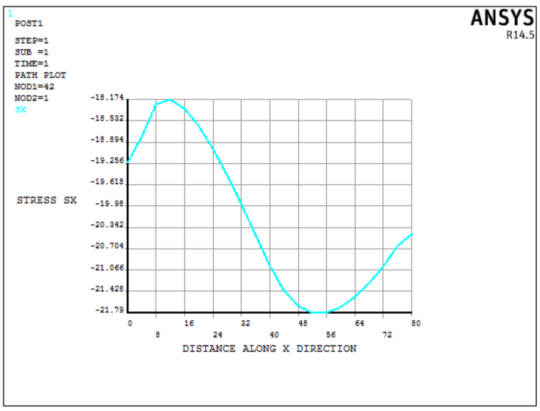

Fig 17: Radial Stress plot across the section.

A curved cantilever beam subjected to moment

Analysing the curved beam under bending moment is the main objective of the problem to find stress distribution in the structure. The problem is intractable using analytical methods. Experimental investigations are highly complex and expensive. However, FE Modeling using ANSYS is able to provide accurate numerical solutions. The stress analysis results are graphically post-processed as contour plots of displacement, radial stress, shear stress plot along AB.

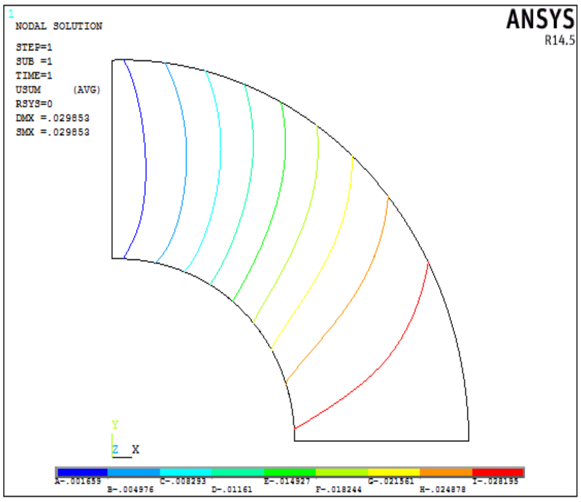

Fig 18: Plot showing the displacement vector sum.

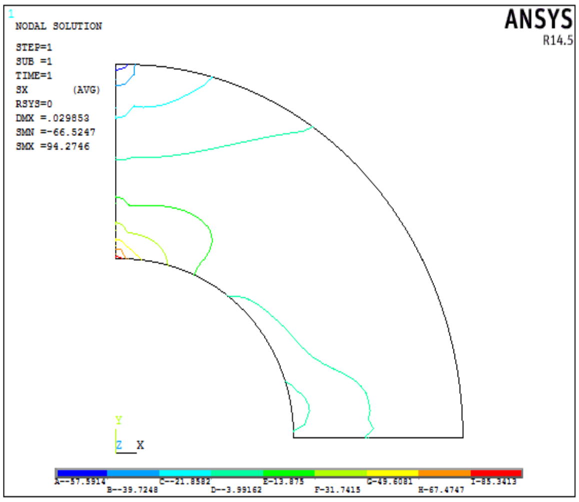

Fig 19: Contour plot showing the stress SX.

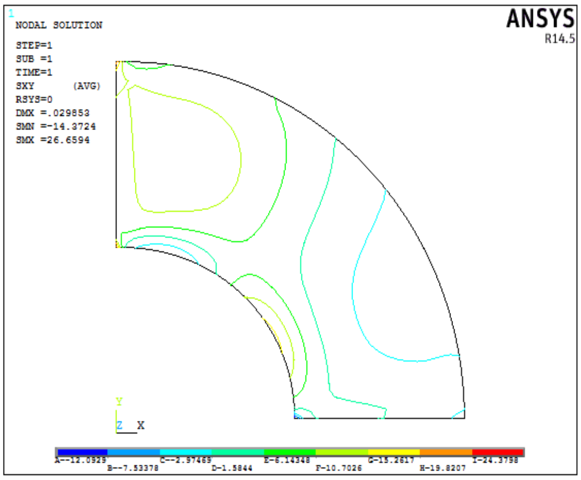

Fig 20: Plot showing the shear stress SXY.

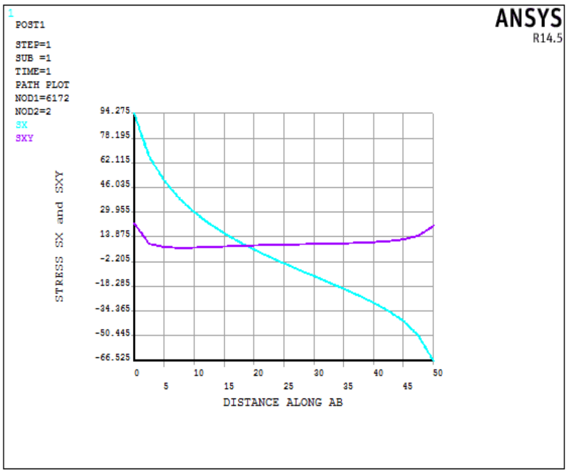

Fig 21: X-Directional stress and shear stress plot.

4. CONCLUSION

In this study the results for the circular plate with an elliptical cut out subjected to external pressure load shows the stresses are maximum at the inner boundary and stress is dropping to external surface for hoop stress. The result shows less variation of hoop stress along the elliptical boundary and shows higher variation for radial stress. The problem is modeled using the plane stress condition. The result of an aero-engine compressor disc subjected to large pressure load at the end of the blade shows maximum stress at the inner boundary and maximum displacement at the outer boundary. The results are found to match with the existing research data. The results for thermal loading shows, maximum temperature at the center and stress also in the same region. The results for a curved cantilever beam subjected to an external loading (moment) shows maximum stress is taking place at the bottom corner region along the constrained region.

The powerful post processing capability in ANSYS software is exploited graphically to present contour plots of the stresses. Based on the present study it can be concluded that the FEM in general and commercial FEA software ANSYS in particular is a unified approach for stress analysis.

In this study correlation of numerical results of FEA was restricted to limited experimental results reported for the problems on hand. However there is a real need for sophisticated experimental investigations to verify all the results presented here. This is identified as future work.

References

1. Albert Kaufman, David Spera, *"Investigation of the elastic- plastic stress state around reinforced opening in a spherical shell",* NASA Scientific and technical publications, Washington, D. C., Feb 1965, PP 1-27.
2. C. R. Calladine, *"On the design of reinforcement for openings and nozzles in thin spherical pressure vessels",* Journal Mechanical Engineering Science, 1966, Vol8 No 1, PP 1-14.
3. R. Kitching, J. K. Davise, *"Limit pressures for cylindrical shells with unreinforced openings of various shapes",* Journal Mechanical Engineering Science, 1970, Vol12 No 5, PP 313-330.
4. M. N. Bapurao, M. V. V. Murthy, *"On the stresses in the vicinity of an elliptic hole in a cylindrical shell under torsional loading",* Nuclear Engineering and Design, Vol. 16, 1971, PP309-321.
5. A. J. Durelii& V. J. Parks, "stresses in a pressurized ribbed cylindrical shell with a reinforced hole", The Journal of Strain Analysis for Engineering Design, 1973, Vol. 8, PP-140-150.