International Journal of Pure and Applied Mathematics
Volume 119 No. 14 2018, 161-166
ISSN: 1314-3395 (on-line version)
url: http://www.ijpam.eu
Special Issue
ijpam.eu

# On the Evaluation of Mixed Mode Stress Intensity Factors for Axial Through Crack in Cylindrical Shell with Tori-Spherical End Closure

**Gurusharan[1], Aditya P.[2], Manish S.[3], Anirud N.[4], A.C.S. Pavan[5]**

*[1] Assistant Professor, Department of Mechanical Engineering, Global Academy of Technology, Bengaluru*
*[2, 3, 4, 5] Student, Department of Mechanical Engineering, Global Academy of Technology, Bengaluru*
**Corresponding author and email:*
**1gurusharan@gat.ac.in*

**Abstract**

Critical assessment of structural integrity, viz. strength, stiffness and durability, is often based on fracture mechanics analysis. Fracture is a mode of failure due to unstable crack propagation. Fracture mechanics deals with methodology for prediction, prevention and control of fracture in materials, components and structures. Tori-spherical shells are used as end closures in pressure vessels and cylindrical container in aerospace structures, thermal and nuclear power plants. This paper presents a refined finite element model and use of a special purpose subprogram to determine mixed-mode membrane and bending stress intensity factors for an axial crack in a cylindrical shell with tori-spherical end closures. The proposed finite element model is implemented using commercial Finite Element Analysis software ANSYS. The stress intensity factors are evaluated using a special purpose subprogram 3MBSIF. The methodology is validated using benchmarks, a set of standard test problems with known target solutions. Parametric studies are carried out to quantify the effect of crack location and orientation on the stress intensity factor values. Further, mixed-mode fracture prediction in terms of direction of crack growth is carried out using strain energy density theory for a specified location of crack with varying crack lengths.

***Keywords:*** *Mixed-mode fracture, Tori-spherical shell, Finite Element Analysis, 3MBSIF, Membrane and Bending Stress Intensity Factors*

## 1. Introduction

A pressure vessel is a closed container designed to hold gases or liquids at pressure higher than that of atmospheric pressure. This pressure differential is very dangerous and many fatal accidents have occurred in the history of their development and operation. Pressure vessels may be of any shape, but most commonly employed shapes are spherical, cylindrical and conical sections. One of the most common designs is a cylinder with end caps called HEADS. Head shapes are usually hemispherical or dished (tori-spherical).

One of the key issues in design and analysis pressure vessel is to assess the structural integrity, viz. strength, stiffness, durability of the pressure vessel. Fracture Mechanics analysis provides a methodology to assess the structural integrity of pressure vessel. Fracture is a failure mode due to unstable propagation of crack. Fracture mechanics enables the design engineer to approach the problem of fracture safe design in a more rational manner [1, 2].

Tori-spherical shells are used as end closure in pressure vessels, cylindrical containers in aerospace structures, thermal and nuclear power plants. Critical assessment of the structural integrity of the pressure vessel with tori-spherical end closures is of prime importance. Damage Tolerance Design methodology based on fracture mechanics is the only design methodology to predict and avoid failure of the structure [3].

The main objective of this study is to develop a refined finite element model and determine the Mixed Mode Membrane and Bending Stress Intensity Factors for axial cracks in a cylindrical pressure vessel with tori-spherical end closure subjected to internal pressure loading. To accomplish this, finite element modeling using ANSYS, a commercial Finite Element Analysis (FEA) software and use of the post-processing subprogram 3MBSIF to compute Stress Intensity Factors posteriori is presented. The methodology is validated using benchmarks. Parametric study for the case of axial crack is presented and discussed.

## 2. Back Ground

Analytical solutions for cylindrical shells with elliptical cuts-outs and cracks was developed by Murthy et al. [4]. Perturbation solutions for arbitrary oriented cracks in cylindrical shell and numerical procedure for analysis of cylindrical shells with arbitrarily oriented cracks was developed by Lakshminarayana et al. [5, 6]. A post-processing subprogram 3MBSIF which calculates Mode I and Mode II membrane and bending stress intensity factor solutions using displacements of flagged nodes was developed by Shivashankar R. Srivatsa et al. [3, 7]. The subprogram was verified and validated for complex shell structures. Stress intensity factor solutions for circumferentially cracked cylindrical shell with tori-spherical end closure was evaluated by Guru Sharan et al. [1]. Mixed mode stress intensity factor solutions for varying crack length and crack angle in helical coil tube with arbitrarily oriented and located cracks was presented by Ajay Krishna et al. [8].

## 3. Finite Element Modeling

Finite Element Modeling is defined here as the analyst's choice of material models, finite elements, meshes, constraint equations, analysis procedures, governing matrix equations and their solution methods, specific pre- and post-processing options available in a chosen commercial FEA software for determination of mixed mode membrane and bending stress intensity factors for shell structures with arbitrarily located and oriented cracks under different types of applied loads and boundary conditions. In this study, ANSYS is used for FE modeling. A fine mesh of singular Iso-parametric curved shell elements (STRIA6), triangular in shape and quadratic in order with six nodes and six engineering degrees of freedom at each node with user specified number NS from one crack face to another and size Δa is created around each crack tip. The rest of the domain under consideration is discretized using a compatible mesh of 8-noded curved shell elements, quadrilateral in shape and quadratic in order (QUAD8) and 6-noded curved shell element of triangular shape (TRIA6). A brief description of these elements is given below.

The QUAD8 element is shown in figure 1. The TOP, BOTTOM and MIDDLE surfaces of the element are curved, whereas the sections across the thickness are generated by the straight lines. The QUAD8 element carries six engineering degrees of freedom ($U_i$, $V_i$, $W_i$, $\theta_{xi}$, $\theta_{yi}$, $\theta_{zi}$) at each of the eight mid surface nodes. The nodal degrees of freedom are illustrated in Figure 1.

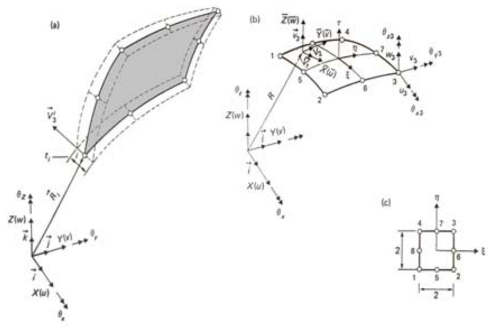

**Fig. 1 Iso-parametric quadrilateral shell element QUAD8**

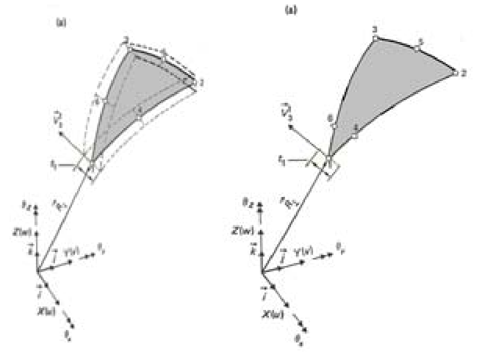

**Fig. 2(a) Iso-parametric triangular shell element TRIA6 (b) Singular Isoparametric triangular shell element STRIA6**

The TRIA6 element shown in figure 2(a) has six nodes and six engineering degrees of freedom at each node. The matrices and vectors for this element are computed as follows: The edge 1-4-8 of the QUAD8 element is collapsed and nodes 4 and 8 are co-located with node 1. Nodes 1, 4 and 8 are tied together to have the same degrees of freedom using multipoint constraint equations. The Singular Iso-parametric Triangular Shell element (STRIA6), shown in figure 2(b), has six nodes and six engineering degrees of freedom at each node. The matrices and vectors for this element are computed as follows: The nodes 4 and 6 which are normally located at mid side positions in the TRIA6 element are moved to the quarter point locations close to node 1. Node 1 in turn is located at a crack-tip. An analysis of the displacement, Strain and Stress field at any point within this element shows that the membrane and bending stress components exhibit the well-known $1/\sqrt{r}$ singularity. The number of STRIA6 elements used around a crack-tip (NS) can be progressively increased and their length reduced till accurate stress intensity factor solution is achieved. This demands a specific pre-processing capability. The pre-processing capability in ANSYS enables the creation of progressively refined mesh of STRIA6 element around each crack-tip with user specified *NS* and Δa. A compatible mesh of regular elements (QUAD8 and TRIA6) then completes the FE Model. Consistent with this FE Model, the stress intensity factors have to be calculated posteriori.

A post-processing subprogram 3MBSIF to calculate posteriori Stress Intensity Factors $K_I^m$, $K_{II}^m$, $K_I^b$ and $K_{II}^b$ and out put their normalized values is used for this purpose.

## 4. Benchmarks

A benchmark is a standard test problem with known target solution in the form of formulae/graphs/tables. The benchmark identified is a cylindrical shell with axial crack [1]. These are used to validate finite element models developed using ANSYS and stress intensity factors calculated using 3MBSIF.

### A. Test Problem

A cylindrical shell of radius R, length 2L, wall thickness t, with axial crack 2a is subjected to internal pressure P. R=1000mm, L=1000mm, t=10mm, P= 1MPa, E=200GPa and ν = 0.3 are used in the computation [1].

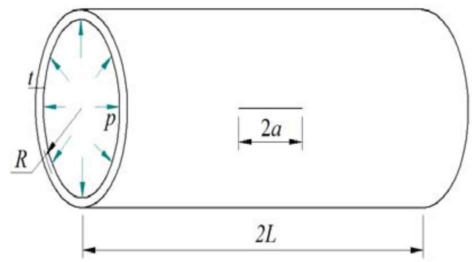


**Fig. 3: Pressure loaded cylinder with axial crack**

The target solutions shown in the figures 3 and 4 is taken from Murthy et.al. [1]. It is to be noted that the crack length parameter is given by

$$\beta^2 = \frac{a^2}{8Rt}[12(1-\vartheta^2)]^{1/2} \quad (1)$$

where β is the curvature parameter.

*B. Results comparison*

Mode-I membrane and bending stress intensity factors computed using 3MBSIF are presented graphically in the figure 5 and 6.

Comparing the present solution with target solution it can be observed that Normalized Mode I component of membrane and bending Stress Intensity Factors obtained from 3MBSIF for plane stress and plane strain assumption very closely matches with the target solution. Hence the finite element model stands validated.

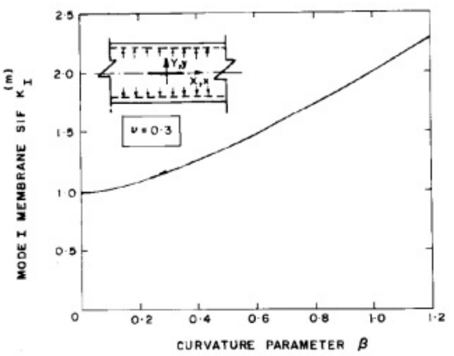


**Fig. 3 Mode I membrane SIF-Target solution [5]**

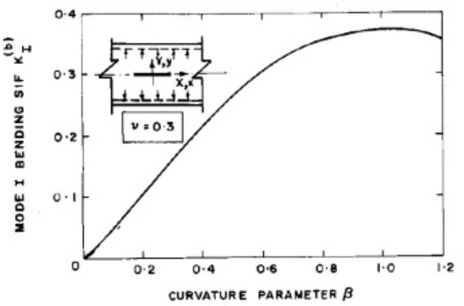


**Fig.4 Mode I bending SIF-Target solution [5]**

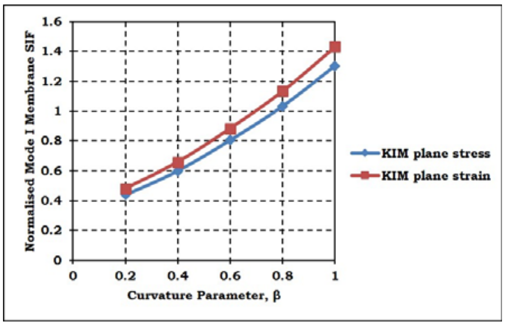


**Fig. 5 Mode I membrane SIF-Present Analysis**

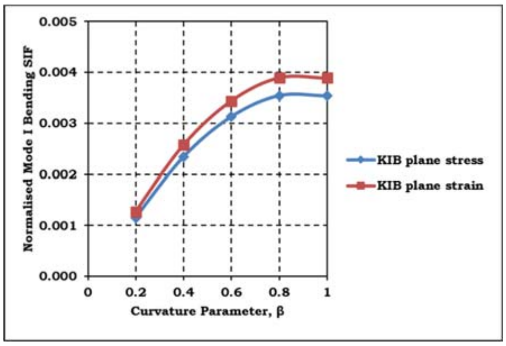


**Fig. 6 Mode I bending SIF-Present Analysis**

## 5. Case Study

A cylindrical shell of radius R, length 2L, wall thickness t, with tori-spherical end closure of Knuckle radius r, and crown radius $r_c$ having an axial crack of length 2a subjected to internal pressure P.
R = 1000mm, r = 500mm, $r_c$ = 1617.413mm, t = 10mm. P = 1MPa. E = 200GPa, ν = 0.3.

The finite element model was generated using ANSYS 14.5 [9]. The model was meshed suitably using Shell 281 element in ANSYS. The geometric model is shown in figure 5. Finite element modeling for circumferential crack is presented in figure 5b. A refined mesh of singular elements (STRIA6) with a compatible mesh of regular elements (QUAD8 and TRIA6) used in the present study is illustrated. The cylindrical shell with tori-spherical end closure is subjected to internal pressure.The graphical post processing capability in ANSYS is demonstrated. A refined mesh of 36 singular elements was generated around the crack tip.

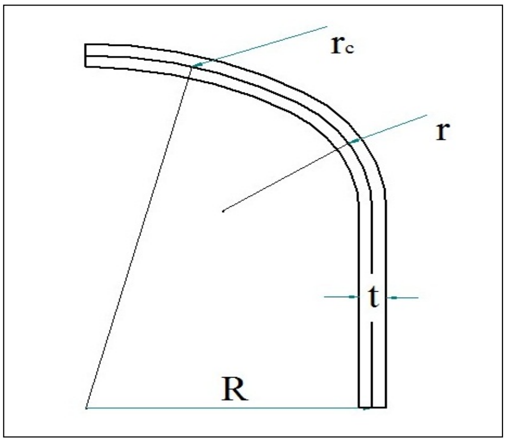


**Fig. 5 Cylindrical Shell with Tori-Spherical End Closure**

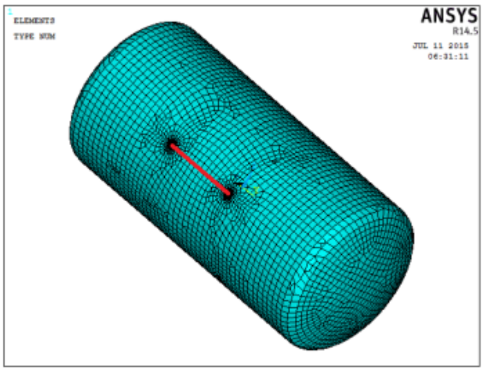


Fig 6 FE Model of Axial Crack in cylindrical region

The Finite Element model of axially cracked cylindrical shell with tori-spherical end closures is shown in figure 6. The crack length is varied by varying the non-dimensional crack length parameter β from 0 to 1 in steps of 0.2. The stress intensity factors for varying crack lengths are presented in figures (7) to (10). Table 1 shows the normalized mode I and mode II membrane and bending stress intensity factors for plane stress and plane strain assumption for axially cracked cylindrical shell with tori-spherical end closures.

**Table 1. Normalized membrane and bending stress intensity factor for axially cracked cylindrical shell with tori-spherical end closure**

| | Plane Stress Assumption | | | | Plane Strain Assumption | | | |
|---|---|---|---|---|---|---|---|---|
| β | $K_I^{(m)}$ | $K_{II}^{(m)}$ | $K_I^{(b)}$ | $K_{II}^{(b)}$ | $K_I^{(m)}$ | $K_{II}^{(m)}$ | $K_I^{(b)}$ | $K_{II}^{(b)}$ |
| 0 | 0 | 0 | 0 | 0 | 0 | 0 | 0 | 0 |
| 0.2 | 0.41 | 0 | 0.001 | 0.043 | 0.46 | 0 | 0.001 | 0.043 |
| 0.4 | 0.59 | 0.005 | 0.002 | 0.028 | 0.66 | 0.006 | 0.003 | 0.031 |
| 0.6 | 0.80 | 0 | 0.003 | 0.023 | 0.88 | 0 | 0.003 | 0.026 |
| 0.8 | 1.04 | 0.008 | 0.004 | 0.021 | 1.14 | 0.009 | 0.004 | 0.023 |
| 1 | 1.3 | 0.006 | 0.004 | 0.019 | 1.42 | 0.008 | 0.004 | 0.021 |

It can be observed that mode I membrane and bending stress intensity factors are gradually increasing with increase in crack length. Mode II membrane stress intensity factor is negligible. Mode II bending stress intensity factor gradually decreases with increase in crack length.

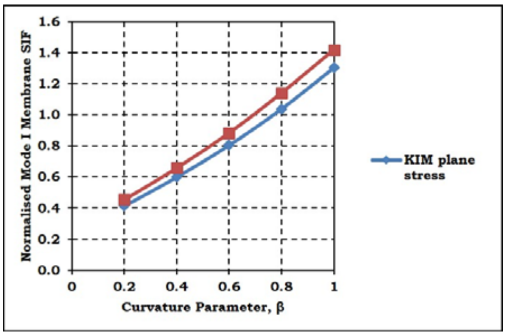


Fig. 7 Normalized Mode I Membrane SIF v/s Curvature Parameter β

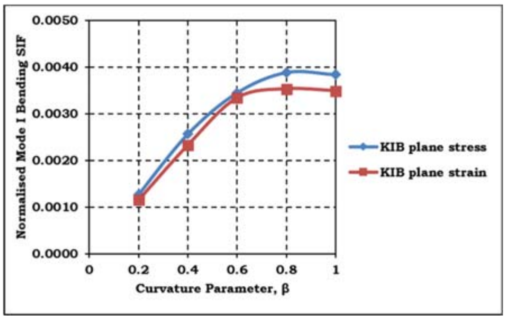


Fig. 8 Normalized Mode I bending SIF v/s Curvature Parameter β

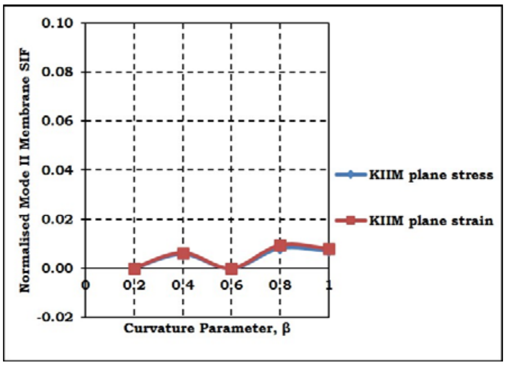


Fig. 9 Normalized Mode II Membrane SIF v/s Curvature Parameter β

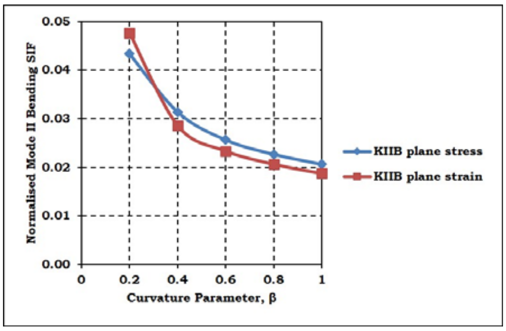


Fig. 10 Normalized Mode II bending SIF v/s Curvature Parameter β

## 6. Conclusion

The Mode I Membrane Stress Intensity Factor varies (increases) linearly with increase in β value, whereas mode I Bending Stress Intensity Factor is increases up to $\beta$=0.8 and then decreases. From the above plots it is clear that mode I Membrane Stress Intensity Factor is more significant compared to mode I Bending Stress Intensity Factor. By comparing the plots of mode I and mode II Membrane Stress Intensity Factors for a given value of $\beta \geq 0.2$ mode I Membrane Stress Intensity Factor is higher and hence it is significant among the two. Now by comparing the plots of mode I and mode II Bending Stress Intensity Factors for a given value of $\beta \geq 0.2$ mode II Bending Stress Intensity Factor is higher and hence it is significant among the two. Finally it can be concluded that the internal pressurized tori-spherical end closured cylindrical pressure vessel with axial crack fails by combination of mode I (membrane) and mode II (bending) fractures.

Finite element modeling using ANSYS software and the use of special purpose post processing sub program 3MBSIF to provide accurate mixed mode membrane and bending stress intensity factor solutions to the complex problem of a cylindrical pressure vessel with tori-spherical end closure under internal pressure with axial crack with of varying lengths is demonstrated in this investigation.

## References


[1] Gurusharan, H. V. Lakshminarayana, Ajay Krishna V. S., NarendraViswanath, *Stress Intensity Factors for Circumferential Through Wall Crack in Cylindrical Shell with Tori-spherical End Closure*, Proc. International Conference on Fatigue, Durability and Fracture Mechanics, Fatigue Durability India, J. N. Tata Auditorium, Indian Institute of Science, 28th -30th May, 2015.

[2] Ajay Krishna V. S., H. V. Lakshminarayana, *Mixed Mode fracture of through cracks in Nuclear reactor Steam generator helical Coil Tube*, Proc. International Conference on Fatigue, Durability and Fracture Mechanics, Fatigue Durability India, J. N. Tata Auditorium, Indian Institute of Science, 28th -30th May, 2015.

[3] Shivashankar R. Srivatsa, H.V. Lakshminarayana, Pramod R., *Stress intensity factors for arbitrarily oriented cracks in a cylindrical shell with tori-spherical end closures subjected to internal pressure*, International Journal of Engineering Research and Applications, Vol.2(3), 2012, pp.3057-3071.

[4] Murthy, M. V. V., Rao, K. P. and Rao A. K., *On the Stress Problem of Large Elliptical cut outs and cracks in circular cylindrical Shells*, International Journal of Solids and Structures, 10, 1974, 1243-1269.

[5] H.V. Lakshminarayana, M.V.V. Murthy, *On stresses around an arbitrarily oriented crack in a cylindrical shell*, International Journal of Fracture, Vol. 12(4), 1976, pp. 547-566.

[6] H.V. Lakshminarayana, M.V.V. Murthy, L.S. Srinath, *On an analytical-numerical procedure for the analysis of cylindrical shells with arbitrarily oriented cracks*, International Journal of Fracture, Vol. 19, 1982, pp. 257-275.

[7] Shivashankar R. Srivatsa, Lakshminarayana H. V., Sudhakar V. H. *Finite Element Models for Fracture Mechanics Analysis of Complex Shell Structures*, Proc. International Conference on Total Engineering and Manufacturing Technologies, Indian Institute of Science, 2008.

[8] Ajay Krishna V. S., Mahesh G. S., Lakshminarayana H. V., *Mixed Mode Fracture of a Nuclear Reactor Steam Generator Helical Coil Tube with Arbitrarily Oriented and Located Through wall crack*, Journal of Mechanical Engineering Research and Technology, Vol.2(1), 2014, pp. 59-66.

[9] *Structural analysis guide*, Release 14.5, ANSYS Inc. Canonsburg, PA. Nov 2014.

[10] Prashanth Kumar, *Elements of Fracture Mechanics*, McGraw Hill Education (India) Ltd., New Delhi, 2009.

[11] H. V. Lakshminarayana, *Finite Element Analysis: Procedures in Engineering*, Universities Press, Hyderabad, 1st edition, 2004.