Journal of Materials Science & Surface Engineering Vol. 3 (4), 2015, pp 308-311

Contents lists available at http://www.jmsse.org/

Journal of Materials Science & Surface Engineering

Stress Intensity Factors for Circumferential Through Wall Crack In A Cylindrical Shell With Tori-Spherical End Closures

Guru Sharan[1*], H. V. Lakshminarayana[2], Ajay Krishna V. S.[3] and Narendra Viswanath[4]

[1]*Assistant Professor, Department of Mechanical Engineering, Global Academy of Technology, Bangalore, India*
[2]*Professor, Department of Mechanical Engineering, Dayananda Sagar College of Engineering, Bangalore, India.*
[3]*Lecturer, Department of Mechanical Engineering, A. P. S. College of Engineering, Bangalore, India.*
[4]*Principal, Global Academy of Technology, Bangalore, India.*

Article history

Received: 17-Aug-2015
Revised: 21-Aug-2015
Available online: 01-Oct-2015

Keywords:
Mixed Mode Fracture,
Tori-Spherical shell,
Finite Element Analysis,
3MBSIF,
Membrane and Bending SIF,

Abstract

A critical assessment of structural integrity: 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 a special purpose subprogram to determine mixed mode membrane and bending stress intensity factors for a cirucumferential crack in a cylindrical shell with tori-spherical end closures. The proposed finite element model is implemented using commercial FEA software ANSYS V15.0. The stress intensity factors are evaluated using 3MBSIF. The methodology is validated using benchmarks, a set of standard test problems with known target solutions. Parametric studies are carried out to study the effect of crack location and orientation on the stress intensity factor values.

The work had been presented at an international conference ***Fatigue Durability India 2015****, 28-30th May 2015, JN TATA AUDITORIUM, Indian Institute of Science, Bangalore.*

Introduction

Tori-Spherical shells are used as end closures in pressure vessels and cylindrical container in aerospace structures, thermal and nuclear power plants. A critical assessment of the structural integrity (stiffness, strength and durability) of the cylindrical shell with tori spherical end closure is of prime importance. Damage tolerance design methodology based on fracture mechanics is the only design methodology to predict and avoid the failure of the structure[1-3].

Cracks are unavoidable in structures. The fracture of a component or structure begins at crack tips. A crack can trigger a local failure at lower load level[2,4]. Fracture mechanics enables design engineers to approach the problem of fracture safe design in a more rational manner. In design, consideration is now given to likelihood that a new structure contains flaws introduced during the processing of the basic material, during fabrication of components and during assembly process. Fracture mechanics analysis involves the determination of stress intensity factors at the crack tip. Therefore, an accurate determination of crack tip stress intensity factors in a given structure and application of strain energy density theory to predict fracture and critical pressure at which the structure fail for a particular location and orientation of crack is essential to the development of safe and reliable designs[5-7].

The main objective of this study is to develop a refined finite element model and new post processing subprogram to determine mixed mode membrane and bending stress intensity factors for circumferential cracks in a cylindrical pressure vessel with tori-spherical end closure subjected to internal pressure loading. To accomplish this, finite element modelling using ANSYS, a commercial FEA software and development of new post processing sub program 3MBSIF to compute the stress intensity factors posteriori is presented. The methodology is validated using benchmarks. Parametric study for circumferential cracks are presented and discussed.

Finite Element Modelling

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 Isoparametric 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 element, 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(a). The TOP, BOTTOM and MIDDLE surfaces of the element are curved,

Corresponding author. Tel./fax: +918904507308
E-mail address: sai_gurusharan@yahoo.co.in

whereas the sections across the thickness are generated by the straight lines. The geometric modeling requires specification of two vectors at each of the eight mid surface nodes. One is the position vector R_i of the node, with the three global Cartesian components X_i, Y_i, Z_i where the subscript identifies the node number. The other is the unit normal vector along with the wall thickness of the same nodes. The QUAD8 element carries six engineering degrees of freedom (U_i, V_i, W_i, θ_{xi}, θ_{yi}, θ_{zi}) at each of the eight mid surface nodes. The nodal degrees of freedom are illustrated in Figure 1(a).

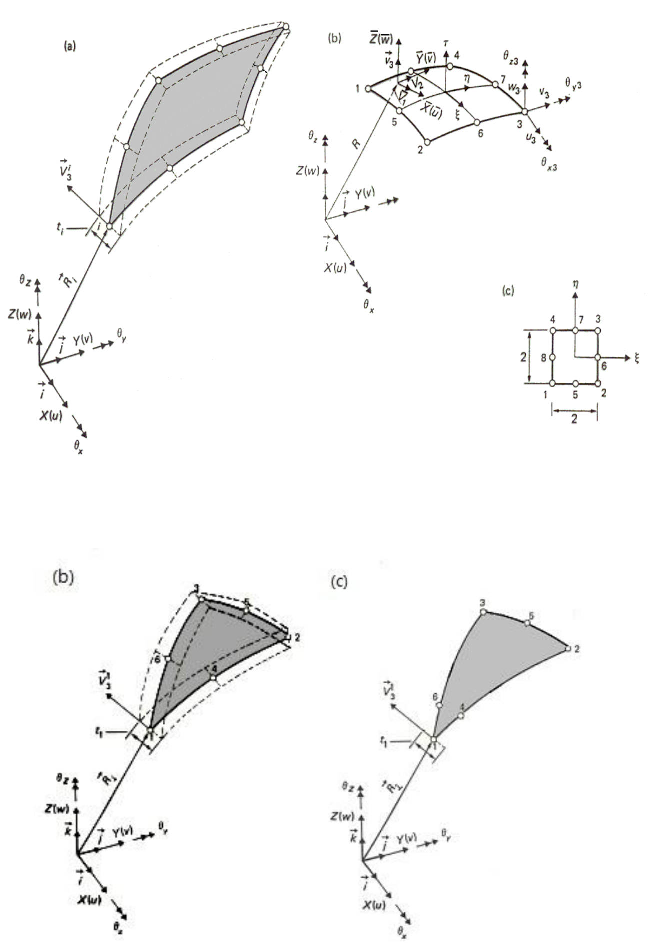


Figure 1: (a) Isoparametric quadrilateral shell element QUAD8; (b) Isoparametric triangular shell element TRIA6 and (c) singular Isoparametric triangular shell element STRIA6

The TRIA6 element shown in Figure. 1(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 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 Isoparametric Triangular Shell element (STRIA6), shown in Figure. 1(c), 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 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 critical assessment of post-processing options for Computational Fracture Mechanics in ANSYS identified the need for development and validation of a special purpose post-processing sub-program for computation of mixed mode membrane and bending stress intensity factors. This program is called 3MBSIF and an overview of this is given in next section.

A Post Processing sub program 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 developed in this study. 3MBSIF is a post-processing subprogram to compute crack-tip Stress Intensity Factors for shell type structures. It can output Mode-I and Mode-II components of MEMBRANE and BENDING Stress Intensity factors individually. It can be used with any commercial general-purpose Finite Element Analysis program that has the modeling capability. ANSYS has the required capability. ANSYS with 3MBSIF therefore is an efficient Computational Fracture Mechanics Tool. The nodal displacements and rotations will be extracted from flagged nodes located on properly flagged SINGULAR elements at the crack tip. Figure 2 shows the singular element around the crack tip, local coordinate system and the flagged nodes.

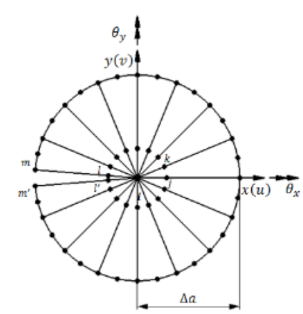


Figure 2: Crack tip coordinate system

There is a need to automate the orientation of the crack tip coordinates x, y, z. x is along the crack plane, z is along the crack front and y is perpendicular to both x and z. This is done to automate the calculation of direction cosines to compute the transformation matrix λ.

$$[\lambda] = \begin{bmatrix} \cos(x,X) & \cos(x,Y) & \cos(x.Z) \\ \cos(y,X) & \cos(y,Y) & \cos(y,Z) \\ \cos(z,X) & \cos(z,Y) & \cos(z,Z) \end{bmatrix}$$

Where X, Y, Z denote the Global Cartesian coordinate system in the solver. The displacements at the crack tip coordinate system (x, y, z) are calculated as

$$\begin{bmatrix} u_i \\ v_i \\ w_i \end{bmatrix} = [\lambda] \begin{bmatrix} U_i \\ V_i \\ W_i \end{bmatrix}$$

$$\begin{bmatrix} \theta_{xi} \\ \theta_{yi} \\ \theta_{zi} \end{bmatrix} = [\lambda] \begin{bmatrix} \theta_{Xi} \\ \theta_{Yi} \\ \theta_{Zi} \end{bmatrix}$$

Appropriate 3MBSIF-evaluation formulae computes, normalizes and outputs Stress Intensity Factor at points along the crack front. These points are located at TOP, MIDDLE and BOTTOM surfaces of the shell along the crack front. Explicit formulae to compute $K_I^{(m)}$, $K_{II}^{(m)}$, $K_I^{(b)}$, and $K_{II}^{(b)}$ are derived and used. These formulae enables user of FEM system to compute surface Stress Intensity Factor (TOP/ MIDDLE/ BOTTOM) using standard outputs namely nodal displacements and rotations.

Benchmarks

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

Test Problem:

A cylindrical shell of radius R, length 2L, wall thickness t, with a circumferential crack of varying length is subjected to an axial force. The crack length is defined by introducing a non-dimensional crack length parameter β. The non-dimensional parameter is given by the relation

$$\beta^2 = \frac{a^2}{8\times R\times t}\left[12\times\left(1-\nu^2\right)\right]^{1/2}$$

R=1000mm, L=1000mm, t=10mm, P= 1MPa, E=200GPa and ν = 0.3 are used in the computation.

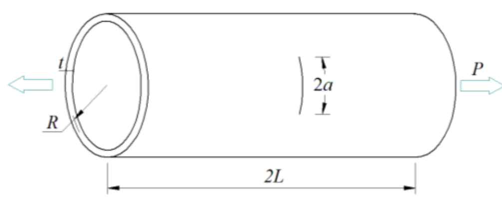


Figure 3: Axial loaded cylinder with a circumferential crack

Table 1: Input Parameters

Material Properties	Geometric Details	Applied Load
Young's modulus $E = 2\times10^5$ N/mm ^{2} Poisson's Ratio ν =0.3	R = 1000 mm L = 1000 mm β = 0 to 1.0 t = 10 mm	P = 1000 N Reference Stress $\sigma_0 = \frac{P}{2\pi Rt}$

Result Comparison:

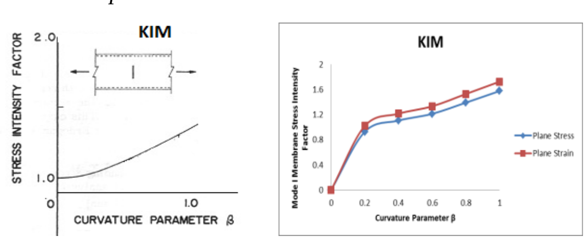


Figure 4: Mode I membrane SIF-Target solution and Present Analysis

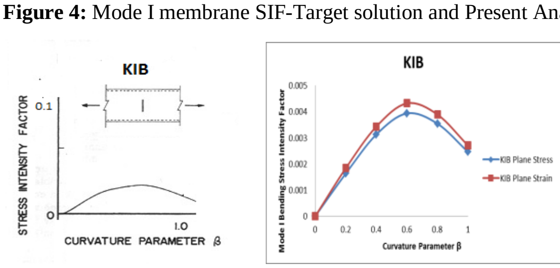


Figure 5: Mode I bending SIF-Target solution and Present Analysis

From Figures 4 and 5 it can be observed that Normalized Mode I components of Membrane and Bending Stress Intensity Factors obtained from 3MBSIF for plane stress and plane strain assumptions very closely matches with the target solution.

Case Study

The geometric modeling of the cylinder with tori spherical end closures was done using ANSYS 15.0. The geometric dimensions used in the computation are: Cylinder Radius= 1000mm, knuckle radius=500mm, knuckle angle=60^0 crown angle=30^0, wall thickness=10mm. The applied internal pressure P= 1 MPa. The material properties are E= 200GPa and Poisson's ratio ν= 0.3.

The finite element model was generated using ANSYS 15.0. The model was meshed suitably using Shell 281 element in ANSYS. The geometric model of the tube is shown in Figure 6 (a). Finite element modeling for circumferential crack is presented in Figure 6 (b). 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 graphical post processing capability in ANSYS is demonstrated. A refined mesh of 36 singular elements was generated around the crack tip.

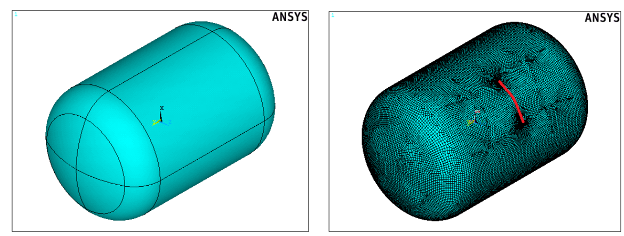
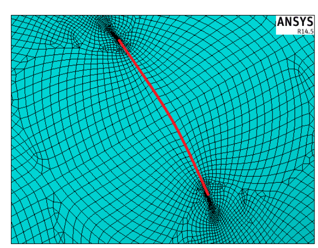


Figure 6: a) Geometric Model of cylinder with tori-spherical end closures
b) FE Model of Circumferential Crack in cylindrical region

Circumferentially cracked cylindrical pressure vessel with tori-spherical end closure is analyzed for varying crack lengths. The crack length is defined by introducing non-dimensional crack length parameter β from 0 to .



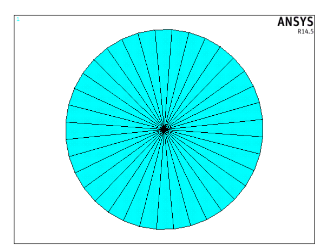


Figure 7: Circumferential crack **Figure 8:** Crack tip singular elements

The Stress Intensity Factors for varying crack lengths are presented in Figures 9, 10, 11 and 12.

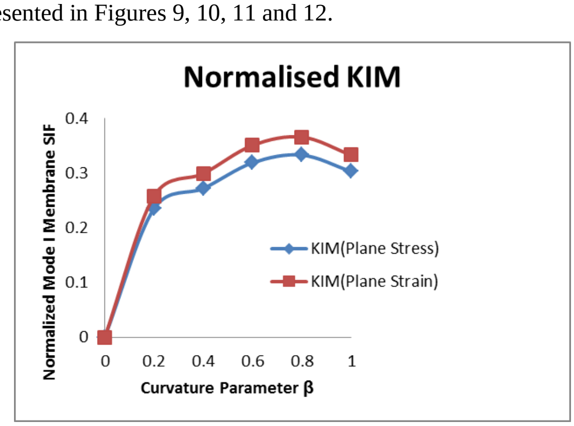


Figure 9: Normalized Mode I Membrane SIF v/s Curvature Parameter β

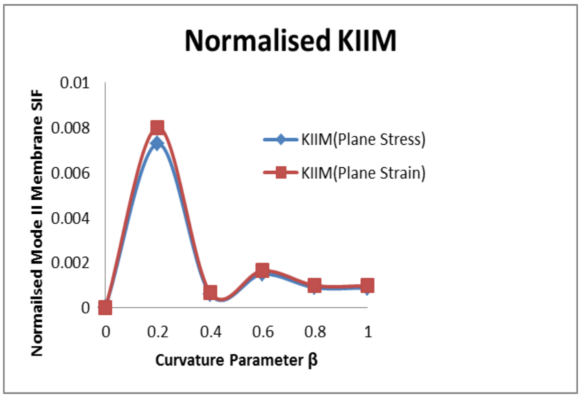


Figure 10 : Normalized Mode II Membrane SIF v/s Curvature Parameter β

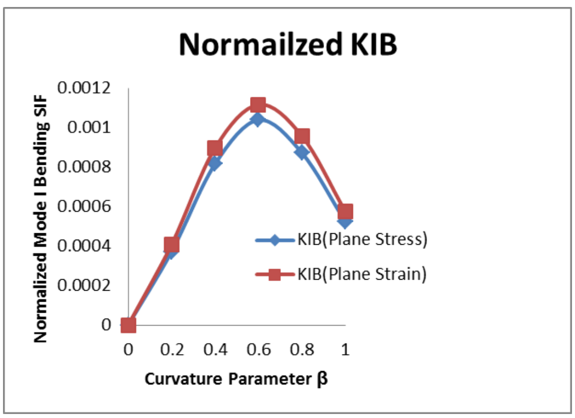


Figure 11: Normalized Mode I bending SIF v/s Curvature Parameter β

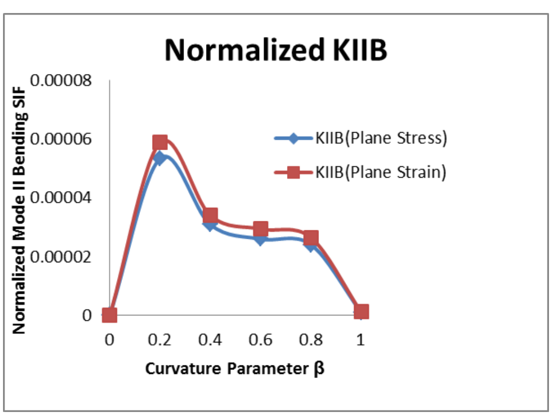


Figure 12: Normalized Mode II Bending SIF v/s Curvature Parameter β

Conclusions

Finite Element Modelling using ANSYS software and the use of special purpose post processing sub program 3MBSIF has provided accurate mixed mode Membrane and Bending Stress Intensity Factor solutions. The results obtained for the Benchmark are in good agreement with the target solutions as seen from the graphs. Thus, the proposed case study is validated for the determination of SIF for varying crack lengths.

References

1. 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.
2. 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.
3. 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.
4. Shivashankar R. Srivatsa, Lakshminarayana H. V., Sudhakar V. H. Finite Element Models for Fracture Mechanics Analysis of Complex Shell Structures, International Conference on Total Engineering and Manufacturing Technologies, Indian Institute of Science, 2008.
5. Structural analysis guide, Release 15.0, ANSYS Inc. Canonsburg, PA. Nov 2014.
6. Prashanth Kumar, Elements of Fracture Mechanics, McGraw Hill Education (India) Ltd., New Delhi, 2009.
7. H. V. Lakshminarayana, Finite Element Analysis: Procedures in Engineering, Universities Press, Hyderabad, 1st edition, 2004.