

A Peer Reviewed Open Access International Journal

# Fracture Analysis of Cracks in Composite Cylindrical Panel

Kumara Swamy P

PG Student Department of Mechanical Engineering Malla Reddy College of Engineering and Technology (MRCET), JNTU-Hyderabad, India.

#### Abstract

Fracture mechanics is mechanics of solids containing planes of displacement discontinuities (cracks) with special attention to their growth. In this paper, the fracture mechanics analysis done in ANSYS for cylindrical panels with semi elliptical non-through surface cracks will be investigated by determining the stress intensity factors, deformation and compared for different materials.

Theoretical calculations will also be done to compare the stress intensity factors, Strain energy release rates and J - Integral.

3D modeling will be done in Pro/Engineer and fracture analysis will be done in Ansys.

Keywords—Stress Intensity Factor, Strain Energy Release Rate, J-Integral, Pro/E, FEA etc.

#### 1. Introduction

#### **1.Fracture mechanics:**

Fracture mechanics is the field of mechanics concerned with the study of the propagation of cracks in materials. It uses methods of analytical solid mechanics[8] to calculate the driving force on a crack and those of experimental solid mechanics to characterize the material's resistance to fracture.

In modern materials science, fracture mechanics is an important tool in improving the mechanical performance of mechanical components. It applies the physics of stress and strain, in particular the theories of elasticity and plasticity, to themicroscopic K.Rajashekar Reddy

Professor Department of Mechanical Engineering Malla Reddy College of Engineering and Technology (MRCET), JNTU-Hyderabad, India.

crystallographic defects found in real materials in order to predict the macroscopic mechanical failure of bodies. Fractography is widely used with fracture mechanics to understand the causes of failures and also verify the theoretical failure predictions with real life failures. The prediction of crack growth is at the heart of the damage tolerance discipline.

There are three ways of applying a force to enable a crack to propagate:

Mode I – Opening mode

A tensile stress normal to the plane of the crack.

Mode II– Sliding mode

a shear stress acting parallel to the plane of the crack and perpendicular to the crack front.

Mode III- Tearing mode

Shear stress acting parallel to the plane of the crack and parallel to the crack front.



Fig -1 Modes of failure

Volume No: 4 (2017), Issue No: 1 (January) www.ijmetmr.com



A Peer Reviewed Open Access International Journal

#### 2. Stress Intensity Factor

The stress intensity factor is used in fracture mechanics topredict the stress state ("stress intensity") near the tip of acrack caused by a remote load or residual stresses.

Stress intensity factor is important parameter linear elastic fracture mechanics for the structure contains crack and singular stress fields. By using the SIF the stress intensity at the crack tip is measured. The stress intensity factor calculated using the stress and strain analysis or using the parameter of the strain energy release rate[6] during the crack growth. The Stress Intensity Factor performed in analytical or numerical techniques.

# Elastic Stress Intensity Factor by empirical equation

The form of the elastic stress distribution near a crack tip that contains the stress-intensity factor  $K_I$  and the square-root singularity iswell known. The determination of  $K_I$  is the basis for linearelastic fracture mechanics. The stress-intensity factor is a function ofload, structural configuration, and the size, shape and location of thecrack. In general, the elastic stress-intensity factor can be expressed as

$$K_{\rm I} = F_0 \sigma_0 \sqrt{\pi a} - - - (1)$$

Where

 $K_I$ -is the Stress Intensity Factor for Mode-I  $\sigma_o$ -is the Nominal stress a-is the Crack length  $F_0$ -is the boundary-correction factor

#### **Trough Crack in a Thin Pressurized Cylinder**

For the through crack (axial) in a cylindrical shell subjected to internal pressure(fig.),the elastic stress intensity factor at failure is given by equation(1) where the nominal stress is

$$\sigma_0 = \frac{PD}{2t} \qquad \dots \qquad (2)$$

And

The boundary-correction factor

**F**<sub>0</sub>= [ 1+0.52λ +1.29λ<sup>2</sup> -0.074 λ<sup>3</sup>]<sup>1/2</sup> ---(3) For 0≤λ≤0 whereλ =  $\frac{a}{\sqrt{Rt}}$ 

# Stress Intensity Factor by virtual crack closure technique

The approach for evaluating the energy-release rate is based on the virtual crack-closure technique (VCCT). The energy-release rate calculation occurs during thesolution phase of the analysis and the results are saved for post processing

Using VCCT for Energy-Release Rate Calculation[6] VCCT is based on the assumption that the energy needed to separate a surface is the same as the energy needed to close the same surface. The implementation described here uses the modified crack-closure method (a VCCT-based method) and assumes further that stress states around the crack tip do not change significantly when the crack grows by a small amount ( $\Delta a$ ). 2-D Crack Geometry For 2-D crack geometry with a low-order element mesh, the energy-release rate is defined as:

$$\begin{split} G1 &= -\frac{1}{2\Delta a} R_Y \, \Delta V \\ G1 &= -\frac{1}{2\Delta a} R_Y \, \Delta U \\ & ---(4) \end{split}$$

GI and GII = mode I and II energy-release rate, respectively  $\Delta u$  and  $\Delta v$  = relative displacement between the top and bottom nodes of the crack face in local coordinates x and y, respectively Rx and Ry = reaction forces at the crack-tip node



Fig-2 2-D Crack Geometry Schematic diagram

**Stress Intensity Factor (KI):** 

$$\mathbf{K}_{\mathbf{I}} = \sqrt{\frac{\mathsf{GE}}{1-\vartheta^2}} - - (4)$$

Where G-is the Strain energy release rate E-is the Young's modulus  $\vartheta$  –is the Poisson's ratio

Volume No: 4 (2017), Issue No: 1 (January) www.ijmetmr.com



A Peer Reviewed Open Access International Journal

#### **3. STRAIN-ENERGY RELEASE RATE**

The traditional-materials strength stress analysis of a cracked component may be hardly tackled. Although the stress discretisation may be improved by using crack tip elements, the meaningful analysis is generally that performed using the energy release rate (G)[6]. It is well-known that plastic deformation occurs in engineering metal, alloys and some polymers. Due to this fact, Irwin and Orowan modified Griffith's elastic surface energy expression.

 $2\gamma_{\rm s} = \frac{\Pi\beta \, a\sigma^2}{E}$ , by adding a plastic deformation energy or plastic strain work  $\gamma p$ in the fracture process. For tension loading, the total elastic-plastic strain-energy is known as the strain energy release rate *G*1=which is the energy per unit crack surface area available for infinitesimal crack extension thus,

$$G=2(\gamma S+\gamma P)-\cdots (5)$$
$$G=\frac{\pi a\sigma^{2}}{E}-\cdots (6)$$

Where

a-Crack length

 $\sigma$ - Nominal stress

E- Young's modulus

This is one of the most important relations in the field of linear fracture mechanics. Hence, equation (6) suggests that G represents the material's resistance(R) to crack extension and it is known as the crack driving force. On the other hand,  $K_{\rm I}$  is the intensity of the stress field at the crack tip.

#### **J-Integral**

The J-integral defines the amplitude of the Hutchinson-Rice Rosengren (HRR) singularity field, just as the stress intensity factorcharacterizes the amplitude of the stress field in linear elastic fracture mechanics (LEFM). Thus J completely describes the conditions within the plastic zone. A cracked structure in small-scaled yielding has two singularity-dominated zones: one in the elastic region, where stress varies as  $r^{-1/2}$  and one in the plastic zone, where stress varies as  $r^{-1/2}$  and one in the plastic zone, where stress varies as the origin).

In EPFM, the latter often persists long after the linear elastic singularity zone has been destroyed by crack tip plasticity.

$$J = K_{IC}^{2} (1 - v^{2}/E) - --(7)$$

Where

K<sub>IC</sub> - Fracture Toughness v–Poisson's ratio E-Young's modulus

#### 4. Pressure Vessel

A pressure vessel is a closed container designed to hold gases or liquids at a pressure substantially different from the ambient pressure.

The pressure vessels are used to store fluids under pressure. The fluid being stored may undergo a change of state inside the pressure vessels as in case of steam boilers or it may combine with other reagents as in chemical plants.

Pressure vessels find wide applications in thermal and nuclear power plants, process and chemical industries, inspace and ocean depths, and in water, steam, gas and air supply system in industries.

The pressure vessel has been used in a various field of aerospace application, like emergency oxygen supplyor propulsion tank systems.

Therefore, the prevention of pressure vessel failure to enhance safety and reliability has received considerable attention.

The failure of pressure vessel are due to different causes are as follows-

1. Failure due to cracking, explosion ruptures, creep and stress rupture.

2. Overtempture, overpressure corrosion, Stress corrosion cracking.

3. Discontinuities, faulty welding, stress raisers, Caustic embrittlement, Erosion.

### Specifications and methodology Specification of Model

The minimum thickness for maximum AllowableWorking pressure of one-half cylindrical



A Peer Reviewed Open Access International Journal

shells shallbe the greater thickness of lesser pressure as given bycircumferential stress (Hoop stress).Longitudinal stress (Circumferential joints).When the thickness does not exceed one-half ofthe inside radius.

Design of	pressure	vessel	as per	ASMECodes[10].
-----------	----------	--------	--------	----------------

Sl.No	Description	Dimension
-		S
1	Length(L)	3000mm
2	Outer	2400 mm
	Diameter(OD)	
3	Inner Diameter(ID)	2350 mm
4	Thickness(t)	25 mm
5	Pressure(P)	75 Psig

### **Design of Pressure vessel in Pro-E Introduction to pro/engineer**

**Pro-ENGINEER, PTC's parametric,** integrated **3D CAD/CAM/CAE solution** is used by discrete manufacturers for mechanical engineering, design and manufacturing. Created by Dr. Samuel P. Geisberg in the mid-1980s, Pro/ENGINEER was the industry's first successful parametric, 3D CAD modeling system. Theparametric modeling approach uses parameters, dimensions, features, and relationships to capture intended product behavior and create a recipe which enables design automation and the optimization of design and product development processes.

This powerful and rich design approach is used by companies whose product strategy is family-based or platform-driven, where a prescriptive design strategy is critical to the success of the design process by embedding engineering constraints and relationships to quickly optimize the design, or where the resulting geometry may be complex or based upon equations. Pro/ENGINEER provides a complete set of design, analysis and manufacturing capabilities on one, integral, scalable platform. These capabilities include Surfacing, Solid Modeling, Rendering, Data Interoperability, Routed Systems Design, Simulation, Tolerance Analysis, and NC and Tooling Design.

Companies use Pro/ENGINEER to create a complete 3D digital model of their products. The models consist of 2D and 3D solid model data which can also be used downstream in finite element analysis, rapid prototyping, tooling design, and CNC manufacturing. All data is associative and interchangeable between the CAD, CAE and CAM modules without conversion. A product and its entire bill of materials(BOM) can be modeled accurately with fully associative engineering drawings, and revision control information. The associatively in Pro/ENGINEER enables users to make changes in the design at any time during the product development process and automatically update downstream deliverables. This capability enables concurrent engineering — design, analysis and manufacturing engineers working in parallel — and streamlines productdevelopment processes.

Pro/ENGINEER is an integral part of a broader product development system developed by PTC. It seamlessly connects to PTC's other solutions including Wind chill, Product View, MathCAD and Arbor text.

### DIFFERENT MODULES IN PRO/ENGINEER

- PART DESIGN
- ASSEMBLY
- DRAWING
- SHEETMETAL
- MANUFACTURING

### **Advantages of Pro/Engineer**

- 1. It is much faster and more accurate. Once a design is completed. 2D and 3D views are readily obtainable.
- 2. The ability to incorporate changes in the design process is possible.
- 3. It provides a very accurate representation of model specifying all other dimensions hidden geometry etc.
- 4. It provides a greater flexibility for change. For example if we like to change the dimensions of our model, all the related dimensions in



A Peer Reviewed Open Access International Journal

design assembly, manufacturing etc. will automatically change.

- 5. It provides clear 3D models, which are easy to visualize and understand.
- 6. Pro/E provides easy assembly of the individual parts or models created it also decreases the time required for the assembly to a large extent.

#### **Isometric View of Pro-E model**



#### E. CALCULATIONS STRESS INTENSITY FACTOR BY USING EMPIRICAL FORMULA

 $K_{I} = F_{0}\sigma_{0}\sqrt{\pi a}$ --- (from eqn.1)  $F_0 = [1+0.52\lambda + 1.29\lambda^2 - 0.074 \lambda^3]^{1/2}$ ---(from eqn.3)  $\lambda = \frac{a}{\sqrt{Rt}} = \frac{10}{\sqrt{1200*25}} = 0.057$  $F_0 = [1+0.52(0.0577) +1.29(0.0577)^2 -0.074]$  $(0.0577)^3$ ]<sup>1/2</sup>  $F_0 = 1.0167$  $\sigma_0$  = Hoop Stress  $\sigma_0 = \frac{PD}{2t}$ ---(from eqn.2) P=0.517 Mpa (Pressure in the vessel (Ref.))  $\sigma_0 = \frac{0.517 * 2400}{2 * 25} = 24.816$  Mpa D=2400 mm (Diameter of the Pressure vessel) a = Crack length major radius 2a=20 mm R =1200 mm (Radius of the Pressure vessel) t =25 mm (Thickness of the Pressure vessel)  $K = 1.0167 * 24.818 \sqrt{\pi * 10}$ K = 141.43 Mpa $\sqrt{mm}$ 

#### STRESS INTENSITY FACTOR BY USINGVIRTUAL CRACK CLOSURE TECHNIQUE:

#### Material Properties

Material	Density(p) (g/cc)	Young's modulus (E) (Gpa)	Poisons ratio (9)	
Stainless steel	7.80	317	0.346	
A1 7075	2.81	71.7	0.33	
E-glass fibber	2.55	85	0.23	
SiC	3.1	410	0.14	
CFRP	1.6	150	0.1	
Table 1				

#### **Case-1:Stainless steel**

$$G = \frac{\pi a \sigma^2}{E} ---(\text{from eqn.6})$$

$$G = \frac{\pi * 10 * 24.818^2}{317 * 10^3} = 0.061 \text{ Mpa.mm}$$

$$K_{I} = \sqrt{\frac{GE}{1 - \vartheta^2}} ---(\text{from eqn.4})$$

$$K_{I} = \sqrt{\frac{0.061 * 317 * 10^3}{1 - 0.346^2}} = 148.29 \text{ Mpa}\sqrt{mm}$$

Case-2:Aluminum 7075  $G = \frac{\pi a \sigma^2}{E}$   $G = \frac{\pi * 10 * 24.818^2}{71.7 * 10^3} = 0.269 \text{ Mpa.mm}$   $K_{I} = \sqrt{\frac{GE}{1 - \vartheta^2}}$   $K_{I} = \sqrt{\frac{0.269 * 71.7 * 10^3}{1 - 0.33^2}} = 147.26 \text{ Mpa} \sqrt{mm}$ 

#### Case-3:E-glass fibber

$$G = \frac{\pi a \sigma^2}{E}$$

$$G = \frac{\pi * 10 * 24.818^2}{85 * 10^3} = 0.227 \text{ Mpa.mm}$$

$$K_{I} = \sqrt{\frac{GE}{1 - \vartheta^2}}$$

$$K_{I} = \sqrt{\frac{0.227 * 85 * 10^3}{1 - 0.23^2}} = 142.84 \text{ Mpa}\sqrt{mm}$$

Case-4: Silicon Carbide  $G = \frac{\pi a \sigma^2}{E}$ 



A Peer Reviewed Open Access International Journal



Case-5:CFRP

$$G = \frac{\pi a \sigma^2}{E}$$

$$G = \frac{\pi * 10 * 24.818^2}{150 * 10^3} = 0.128 \text{ Mpa.mm}$$

$$K_{I} = \sqrt{\frac{GE}{1 - \vartheta^2}}$$

$$K_{I} = \sqrt{\frac{0.269 * 150 * 10^3}{1 - 0.1^2}} = 139.713 \text{ Mpa}\sqrt{mm}$$

### **CALCULATIONS FOR J – INTEGRAL**

**J** – **Integral** =  $K_{IC}^2$  (1-  $v^2/E$ ) ---(from eqn.7) Where

$$\begin{split} \mathbf{K}_{IC} &= \mathbf{Fracture \ Toughness} \\ \mathbf{K}_{IC} &= \mathrm{Structural \ Steel} = 1581 \ \mathrm{MPa \ mm}^{0.5} \\ \mathbf{K}_{IC} &= \mathrm{Al \ 7075} = 790.56 \ \mathrm{MPa \ mm}^{0.5} \\ \mathbf{K}_{IC} &= \mathrm{E}\text{-glass \ fiber} = 150.208 \ \mathrm{MPa \ mm}^{0.5} \\ \mathbf{K}_{IC} &= \mathrm{Silicon \ Carbide} = 145.46 \ \mathrm{MPa \ mm}^{0.5} \\ \mathbf{K}_{IC} &= \mathrm{CFRP} = 192.89 \ \mathrm{MPa \ mm}^{0.5} \end{split}$$

1. Stainless Steel

J –Integral = 1581<sup>2</sup> (1- 0.346<sup>2</sup>/317000) = 6.94 mJ/mm<sup>2</sup> 2. Al 7075 J –Integral = 790.56<sup>2</sup> (1- 0.33<sup>2</sup>/71700) =7.7674

mJ/mm<sup>2</sup>

### 3. E-glass fibber

J –Integral = 150.208<sup>2</sup> (1- 0.23<sup>2</sup>/85000) =0.251 mJ/mm<sup>2</sup>

4. Silicon Carbide

J –Integral =  $145.46^2$  (1-  $0.14^2/410000$ ) =0.05 mJ/mm<sup>2</sup> 5. CFRP J –Integral = $192.89^2$  (1-  $0.1^2/15000$ ) =2.455 mJ/mm<sup>2</sup>

#### FINITE ELEMENT ANALYSIS (FEA)

Finite Element Analysis (FEA) was first developed in 1943 by R. Courant, who utilized the Ritz method of numerical analysis and minimization of variational calculus to obtain approximate solutions to vibration systems. Shortly thereafter, a published in 1956 by M. J. Turner, R. W. Clough, H. C. Martin, and L. J. Top established a broader definition of numerical analysis. The centered on the "stiffness and deflection of complex structures". By the early 70's, FEA was limited to expensive mainframe computers generally owned by the aeronautics, automotive, defense, and nuclear industries. Since the rapid decline in the cost of computers and the phenomenal increase in computing power, FEA has been developed to an incredible precision. Present day supercomputers are now able to produce accurate results for all kinds of parameters. FEA has become a solution to the task of predicting failure due to unknown stresses by showing problem areas in a material andallowing designers to see all of the theoretical stresses within. This method of product design and testing is far superior to the manufacturing costs which would accrue if each sample was actually built and tested.

In practice, a finite element analysis usually consists of three principal steps:

**Preprocessing:** The user constructs a model of the part to be analyzed in which the geometry is divided into a number of discrete sub regions, or elements," connected at discrete points called nodes." Certain of these nodes will have fixed displacements, and others will have prescribed loads. These models can be extremely time consuming to prepare, and commercial codes vie with one another to have the most user-friendly graphical "preprocessor" to assist in this rather tedious chore. Some of these preprocessors can overlay a mesh on a preexisting CAD file, so that finite element analysis can be done conveniently as part of the computerized drafting-and-design process.

**Analysis:** The dataset prepared by the preprocessor is used as input to the finite element code itself, which constructs and solves a system of linear or nonlinear algebraic equations

#### Kijuj = fi

Where u and f are the displacements and externally applied forces at the nodal points. One of FEA's principal advantages is that many problem types can



A Peer Reviewed Open Access International Journal

be addressed with the same code, merelyby specifying the appropriate element types from the library.

**Post processing:** In the earlier days of finite element analysis, the user would pore through reams of numbers generated by the code, listing displacements and stresses at discrete positions within the model. It is easy to miss important trends and hot spots this way, and modern codes use graphical displays to assist in visualizing the results. Typical postprocessor display overlays colored contours representing stress levels on the model, showing a full field picture similar to that of photo elastic or moiré experimental results.

### FRACTURE ANALYSIS OF COMPOSITE CYLINDRICAL PANELS OF PRESSURE VESSEL

#### MATERIAL – STAINLESS STEEL

Save Pro-E Model as .iges format

 $\rightarrow \rightarrow$  Ansys  $\rightarrow$  Workbench $\rightarrow$  Select analysis system  $\rightarrow$  static structural  $\rightarrow$  double click

 $\rightarrow$  Select geometry  $\rightarrow$  right click  $\rightarrow$  import geometry  $\rightarrow$  select browse  $\rightarrow$  open part  $\rightarrow$  ok

#### **Imported model**



Double click on geometry  $\rightarrow$  select MSBR  $\rightarrow$  edit material  $\rightarrow$ 

Select coordinate system  $\rightarrow$  right click  $\rightarrow$ insert co ordinate system

Select mesh on left side part tree  $\rightarrow$  right click  $\rightarrow$  generate mesh  $\rightarrow$ 





Select model  $\rightarrow$ select fracture tool Select geometry – Select first plate Select Crack Shape – Semi Elliptical Enter major radius  $\rightarrow 20$  mm Enter minor radius  $\rightarrow 2$  mm Enter Fracture affected zone Height – 1.9511mm Enter largest contour radius – 10 mm

#### **Crack on Pressure vessel**



Select crack  $\rightarrow$  right click  $\rightarrow$  insert  $\rightarrow$ generate all crack meshes.

Select static structural right click  $\rightarrow$  insert  $\rightarrow$  select pressure and Fixed support $\rightarrow$ 

#### **Fixed support**





A Peer Reviewed Open Access International Journal

Select fixes support  $\rightarrow$  select required area  $\rightarrow$  click on apply  $\rightarrow$ 

Select pressure  $\rightarrow$  select required area  $\rightarrow$  click on apply  $\rightarrow$  enter pressure value 2 Mpa  $\rightarrow$ 

#### Pressure



Select solution right click  $\rightarrow$  solve  $\rightarrow$ 

Solution right click  $\rightarrow$  insert  $\rightarrow$  deformation  $\rightarrow$  total  $\rightarrow$  Solution right click  $\rightarrow$  insert  $\rightarrow$  strain  $\rightarrow$ equivant (von-mises)  $\rightarrow$ 

Solution right click  $\rightarrow$  insert  $\rightarrow$  stress  $\rightarrow$  equivant(von-mises)  $\rightarrow$ 

Right click on deformation  $\rightarrow$  evaluate all result  $\rightarrow$ select solution  $\rightarrow$ right click  $\rightarrow$  insert  $\rightarrow$ fracture tool  $\rightarrow$  select fracture tool $\rightarrow$  right click  $\rightarrow$ insert  $\rightarrow$ stress intensity factor and j-integral .  $\rightarrow$ select stress intensity factor  $\rightarrow$ right click $\rightarrow$  evaluate all results.

#### STRESS INTENSITY FACTOR



### J-INTEGRAL



## Volume No: 4 (2017), Issue No: 1 (January) www.ijmetmr.com

#### MATERIAL –AL 7075 STRESS INTENSITY FACTOR



#### **J-INTEGRAL**



### MATERIALS – E-GLASS FIBER STRESS INTENSITY FACTOR



#### J-INTEGRAL



## MATERIALS – SILICON CARBIDE STRESS INTENSITY FACTOR





A Peer Reviewed Open Access International Journal

#### J-INTEGRAL



### MATERIALS – CFRP STRESS INTENSITY FACTOR



### J-INTEGRAL



#### FRACTURE ANALYSIS RESULTS Stress Intensity Factor and Fracture Toughness

Material	Stress Intensity Factor by Theoreticall y (K <sub>I</sub> ) Mpa√mm	Stress Intensity Factor by FEA(KI) Mpa√mm	Fracture Toughness(K <sub>IC</sub> ) Mpa $\sqrt{mm}$
Stainless Steel	148.09	104.50	1581
Al 7075	147.26	104.13	790.56
E-glass fibber	142.84	102.19	150.208
Silicon Carbide	140.34	100.86	145.46
CFRP	139.713	82.636	192.89

## J-Integral and Strain Energy Release Rate

Material	J-Integral by Theoretically (J) (mJ/mm <sup>2</sup> )	J-Integral by FEA(J) (mJ/mm²)	Strain Energy Release Rate(G) (Mpa.mm)
Stainless Steel	6.94	0.03	0.061
Al 7075	7.767	0.13	0.269
E-glass fibber	0.251	0.11	0.227
Silicon Carbide	0.05	0.02	0.0471
CFRP	2.455	0.044	0.128

# FRACTURE ANALYSIS GRAPHS

**Stress Intensity Factor** 



#### **Strain Energy Release Rate**





A Peer Reviewed Open Access International Journal

#### **J-Integral**



#### **IV.CONCLUSION**

In the present paper, the three-dimensional finite element calculations are performed to investigate the axial surface crack inside the cylindrical pressure vessel with hoop stress.

The Stress Intensity Factor along the crack are determined theoretically and compared with FEA.

Stress Intensity should be lower than fracture toughness (KI  $\leq$  KIC), The failure occurs when the Stress Intensity near the discontinuity reaches the fracture toughness.

The Stress Intensity Factor is less compared with Fracture toughness then the material is safe.

The Stress Intensity Factor is less for the composite materials (E-glass fiber, SiC and CFRP) compared with Alloy Materials (Stainless Steel and AL 7075) by theoretically and analytically from the resultsobtained the composites are better compared with alloys.

#### REFERENCES

[1] Fracture Mechanics Analysis of Cylindrical Panels with Non-Through Cracks, S. V. Glushkov and Yu. V. Skvortsov, ISSN 1068-7998, Russian Aeronautics (Iz.VUZ), 2014, Vol. 57, No. 3, pp. 240–244.

[2] Fracture analysisOf surface and through cracksIn cylindrical pressure vessels, J. C. Newman, Jr.Langley Research Center Hampton, NASA TECHNICAL NOTENASA TN D-8325 Va. 23665. [3] Stress intensityfactor of semi-elliptical surface crack in a cylinder with hoop wrapped composite layer, Jianjun Chen, Hongliang Pan,International Journal of Pressure Vessels and Piping 110 (2013) 77e81.

[4] An Investigation on Cracked Plate for Stress Intensity Factor for Selected Configurations under Different Loading Modes,Pravin R. Gawande, Ajay Bharule,International Journal of Scientific Engineering and Research (IJSER):ISSN (Online): 2347-3878.

[5] Analysis of Cracked Cylindrical Pressure Vessel by using Experimental approach, Mundhe N.D., Utpat Abhay A, International Journal of Latest Trends in Engineering and Technology (IJLTET).

[6] Computation of Stress Intensity Factor of Cracked Aluminium Plate Using Virtual Crack Closure Technique, A.Gopichand, M.Surendra Kumar, A.V.N.L.Sharma, International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622 Vol. 2, pp.460-465.

[7] A Review on Failure Modes of Composite Pressure Vessel, Mahesh B. Gosavi, Prof. A.S. Rao, Vinaay Patil, ISSN: 2321-9939, Volume 2.

[8] Pressure vessel failure analysis, Pablo ViníciusBassani, Heraldo José Amorim, Ignacio Iturrioz,20th International Congress of MechanicalEngineering.

[9] Probabilistic fracture analysis of cracked pipes with circumferential flaws,S.Rahman, International Jounal of Pressure Vessel & Piping,Vol.70,pp.223-236.

[10] Design of pressure vessel using ASME code, B.S.Thakkar, S.A.Thakkar Section viii, division 1",International Journal of Advanced Engineering Research and Studies E-ISSN2249–8974.