International Journal of Research in Advanced Engineering and Technology
ISSN: 2455-0876; Impact Factor: RJIF 5.44
www.engineeringresearchjournal.com
Volume 3; Issue 1; January 2017; Page No. 28-31

Crack analysis of a thin walled pressure vessel by using FEA

[1] Sandip Kumar, [2] Manoj Nikam, [3] Uvaraj Mane

[1, 3] lecturer, Department Of Mechanical Engineering, Bharati Vidyapeeth Institute of Technology, Mumbai University, Navi Mumbai, Maharashtra, India

[2]Research Scholar, Department Of Mechanical Engineering, VJTI, Mumbai University, Mumbai, Maharashtra, India

Abstract

Any crack on the surface of pressure vessel can lead to a fatal accident during operation, so it needs to be crack analysis. The present work has done on a horizontal type of thin walled pressure vessel. It was designed to work for an internal pressure of around 150 bars and external pressure of one atmospheric. Its maximum working temperature is 70^0C. Hence procedure has been developed to analyse the crack extension over the surface of pressure vessel due to operating parameters by using techniques like SIFs and J-integral supported by the results of FEA.

Keywords: pressure vessel, SIFs, J-integral, and FEA

1. Introduction

Industries have to deal with different toxic gases as well as chemicals for different applications so they have pressure vessel [1]. Generally, there is huge difference in internal and external pressures of the pressure vessel. If pressure vessel fails or get explode, it may cause a huge damage to human life and property [2]. The failure of the pressure vessel starts from a small crack may remain present during manufacturing processes or due to material deficiency on the surface and due to different operating parameters. So it becomes mandatory for the pressure vessel manufacturer and user to crack test the pressure vessel before operation.

2. Experimental work

Provide sufficient detail to allow the work to be reproduced. Methods already published should be indicated by a reference, only relevant modifications should be described.

Analytical calculation of a rectangular plate with crack at the centre

Even in a circular tube with a crack having its length much smaller than the diameter of the tube, the region around the crack tip may be regarded as a flat plate [3]. So keeping it as the base, 2D rectangular plate is considered for the analysis. It consists of a horizontal centre crack of length ‘2a’. Tensile stress ‘σ’ is applied at the two ends. ‘B’ represents the thickness of the plate and width of the plate is considered as ‘w’.

Analytical calculations for SIF (K)

From the calculation point of view, the dimension of the plate has taken as 200 × 50 × 5 mm. The force acting on the cross-section is given by,

Stress = 5MPa, Cross sectional area = 50×5 mm^2, Force for 5mm thickness = 1250 N, Force for 1mm thickness = 250 N, Stress (σ) = 5MPa, Consider crack length (2a) = 5 mm

Then,

$$K = \sigma \sqrt{\pi a}\, f(\alpha) \quad \ldots\ldots (1)$$

$\alpha = a/w$ for $0 \le \alpha \le 0.7$

$$f(\alpha) = 1 + 0.128\,\alpha - 0.288\,\alpha^2 + 1.523\,\alpha^3$$
$$=1.005870375$$

SIF “K”= 14.0947 $MPa.mm^{0.5}$

Analytical Calculations for J-Integral (J)

The J-Integral (J) of a crack is equal to the summation of the J-Integral of the elastic region (J_{el}) and J-Integral of the plastic region (J_{pl}). The J_{el} value of the material is equal to the stress concentration factor ‘G’ and valueJ_{pl} can be calculated by using Ramberg-Osgood relation.

$$J= J_{el} + J_{pl} \quad \ldots\ldots (2)$$
$$J_{el}=K^2/E \quad \ldots\ldots (3)$$

Where,

K= Stress Intensity Factor of the material for a given crack length.
K= 14.0947, for a length of crack 5mm.
E= Modulus of elasticity of the material.
E= 2 x 105,

Now by using the equation (3)

Jel = (14.09472) / 2 x 105
= 9.9×10-4MJ/mm2

Similarly, the relation of the J-integral for the plastic material is given in equation (4) below;

$$J_{PL}= \alpha\sigma_0 \in b g_1 h_1 \left(\frac{p}{p_0}\right)^{n+1} \quad \ldots\ldots (4)$$

Where,

α = Is a dimensional number
α = [(2a/b)2 + 2(2a/b)1/2 - (2a/b) +1]
a= length of the crack along the width
ε= σo /E
w = width of the plate
b = (w-a)/2
σo= Yield stress of the material
P= is the applied load per unit thickness of the plate.
ε= Yield strain of the material

P= is the applied load per unit thickness of the plate.
b= length of uncracked ligament
P_0=2bσys.
n = 4.3, Strain hardening constant for the given material
For the given material,
Now by using the equation (4), the J_{PL} can be calculated for a given crack lengths as below;
For crack length 'a'=5mm.
$\sigma_0 = 170MPa$
n=4.3, α=2.09

P= 250 N/mm
$b = \frac{50-5}{2} = 22.5mm.$
$h_1 = 4.5$
$\epsilon = \frac{\sigma_0}{E} = 0.00085$
$g_1 = \frac{a}{w} = \frac{5}{50} = 0.1$
$p_0 = 2b\sigma ys$
= 2×22.5×170=7650 N/mm

$$Jpl = 2.09 \times 170 \times 0.00085 \times 22.5 \times 0.1 \times 4.5(\frac{250}{7650})^{4.3+1}$$

$Jpl = 4.0839 \times 10^{-8}$ MJ/mm^2

Now the total J-integral from equation (2) by adding both J_{el} and J_{pl} is given in the table 1 below for the given crack lengths.

Table 1: Results of total J-Integral

Sr. No.	Crack length (mm)	SIF 'K$_1$'(MPa.mm$^{0.5}$)	J-integral elastic, (MJ/mm^2)	J-integral plastic, (MJ/ mm^2)	Total J-Integral, (MJ/ mm^2)
1	5	14.0947	9.9×10^{-4}	0.0004084×10^{-4}	9.9004084×10^{-4}
2	6	15.4569	11.9×10^{-4}	0.0005759×10^{-4}	11.9005759×10^{-4}
3	7	16.7136	13.9×10^{-4}	0.0007920×10^{-4}	13.900792×10^{-4}
4	8	17.8872	14.0×10^{-4}	0.0010768×10^{-4}	14.0010768×10^{-4}
5	9	18.9933	16.0×10^{-4}	0.0014539×10^{-4}	16.0014539×10^{-4}
6	10	20.0433	20.1×10^{-4}	0.0013835×10^{-4}	20.1013835×10^{-4}

Finite element Analysis of a 2D rectangular metal plate having crack at center

ANSYS software is one of the most popular commercial software is used for the Finite element analysis of the pressure vessel. The first step in the analysis is to create a geometrical CAD model of the rectangular plate that is created in the Ansys workbench2016.

Material Properties

At this point we have to define the material physical properties of the material used for the given problem. The material used for the manufacturing of pressure vessel is SA 240 GR 316 and its properties are given in table 2 below.

Table 2: Properties of material in ANSYS Workbench

Sr. No.	Properties	Values
1	Yield strength	170 N/mm2
2	Tensile strength	485 N/mm2
3	Modules of elasticity	2 x 105 N/mm2
4	Elongation	40%
5	Hardness	217 Brinell, 97 Rockwell
6	Poison's ratio	0.3
7	Density of material	7833Kg/m3
8	Element type	Solid
9	Material	SA 240 GR 316

Generation of crack

The fracture tool was selected to generate an automatic crack at the new crack coordinates on the surface of the rectangular plate. The dimensions have been selected arbitrary as used during analytical calculations. Following are the dimensions used for the crack at crack coordinate.

Coordinate system= crack location. Crack shape = semi elliptical
Major axis =5mm. Minor axis =1mm
Largest contour radius =0.5m Circumferential divisions =8
Mesh contours =6 Crack front divisions =15

After defining the crack length in the ANSYS Workbench16.1 the next step is to generate the mesh around the crack geometry that is semi-elliptical in this case.

Loading and Boundary conditions

The boundary conditions are applied and those are at upper section line DX=0, DZ=0 and at bottom section line all DOF =1. A static structural analysis is selected for the current analysis. So the pressure applied to the plate is as follows:
A = pressure applied -5MPa, B = Pressure applied -5MPa.

Comparison of analytical & FEA results for a2D rectangular plate (crack at the centre)

By the comparison of the results obtained by analytical calculations and by using FEA Ansys Workbench16.1, it is going to be proved that the method followed so far is correct and can be used for the crack analysis of any component. Refer fig. 1 and table 3.

Table 3: Percentage difference between analytical and FEA SIF results

Crack length, (mm)	Analytical J-Integral, (MJ/mm^2)	ANSYS J-Integral, (MJ/mm^2)	Percentage difference
5	9.90×10^{-4}	9.85×10^{-4}	0.5
6	11.90×10^{-4}	11.32×10^{-4}	4.8
7	13.90×10^{-4}	13.28×10^{-4}	4.4
8	14.00×10^{-4}	13.53×10^{-4}	3.4
9	16.00×10^{-4}	15.42×10^{-4}	3.6
10	20.10×10^{-4}	19.30×10^{-4}	3.9

Comparison of SIF results

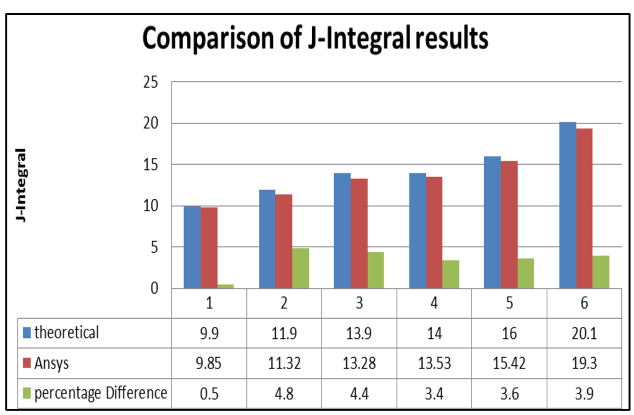

Fig 1: Comparison between analytical and FEA values of SIF

After the comparison of both results, the percentage difference between the results of two methods is very less and the maximum difference is only 1.72%. So the method followed in this project to analyse a crack present on the surface of a material plate is correct. The results obtained for the Stress Intensity Factor has a maximum value 20.0433MPa.mm$^{0.5}$ for a length of 10 mm of crack.

Comparison of J-Integral results

The percentage difference between the two can be neglected. So it has been proved that the procedure followed to analyse a crack present on a rectangular plate is correct. The same techniques can be used to analyse any crack present on any surface. Refer fig. 2 and table 4.

Table 4: The percentage difference between analytical and FEA values of J-integral

S. No.	Analytical J-Integral, (MJ/mm^2)	ANSYS J-Integral, (MJ/mm^2)	Percentage difference
5	9.90×10^{-4}	9.85×10^{-4}	0.5
6	11.90×10^{-4}	11.32×10^{-4}	4.8
7	13.90×10^{-4}	13.28×10^{-4}	4.4
8	14.00×10^{-4}	13.53×10^{-4}	3.4
9	16.00×10^{-4}	15.42×10^{-4}	3.6
10	20.10×10^{-4}	19.30×10^{-4}	3.9

Calculation of Stress Intensity Factor and J-Integral of a thin walled pressure vessel by using FEA

Model or Geometry

Among the different softwares, Solid works is used to create CAD model of the pressure vessel, because it is user friendly and easy to use. The assembly of the thin walled pressure with C-clamps is imported into the Ansys Workbench 16.1 for the further analysis.

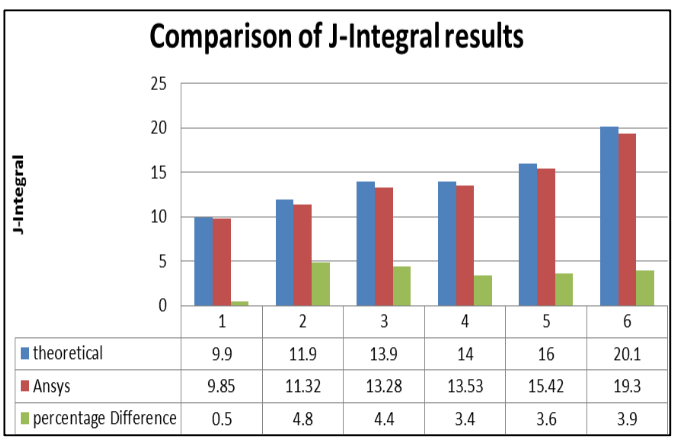

Fig 2: Comparison between analytical and FEA values of J-Integral

Mesh the object created

The best possible method control is tetrahedron as per the analysis and type of geometry is concerned. After the meshing the CAD model of the pressure vessel looks the same. Tetrahedrons (4 nodes) element has been adopted for the analysis of the thin walled pressure vessel. The parameters related to this are as follows;

Type of Element = Tetrahedron
Min. element size = 4mm
Max. Element size = 20 mm.
Relevance centre = fine
No. of Nodes= 83275
No. of Elements =39411
Mid side Node= dropped.

Generation of crack

A crack is characterized by its shape, crack front/tip, crack discontinuity plane, crack normal, and crack direction [7]. A crack front in three dimensional analyses represents the line of separation of the discontinuous crack surface. The same is represented by a crack tip in three dimensional analyses. From the past experiences of pressure vessel failures, the pressure vessel got exploded suddenly around the nozzles or from any of the openings around the surface of the pressure vessel that indicate that the maximum stresses exerted around the openings. Following are the dimensions used to generate a crack at crack coordinate, and they are same as used for the analysis of the plate earlier and they are as follows below;

Coordinate system= crack location.
Crack shape = semi elliptical
Major axis =5mm.
Minor axis =1mm
Largest contour radius =0.5mm
Circumferential divisions =8
Mesh contours =6
Crack front divisions =15

Loading and Boundary conditions on plate

A static structural analysis is chosen for the current analysis. The value of the pressure applied is as follows:

A = pressure applied -5MPa, B = Pressure applied -5MPa. C= Fixed Support.

3. Results and Discussion

The parameters which are required for the crack analysis of a thin walled pressure vessel are as follows;

Equivalent Stress

On the basis of the equivalent Von Mises stress a designer can predict that its design will fail, if the value of the equivalent Von Mises stress obtained is more than the material toughness. As per the current Finite Element Analysis the value obtained for equivalent stress is 423.34 MPa. The location of these stresses would be around the surface area of the nozzle. (Fig.3)

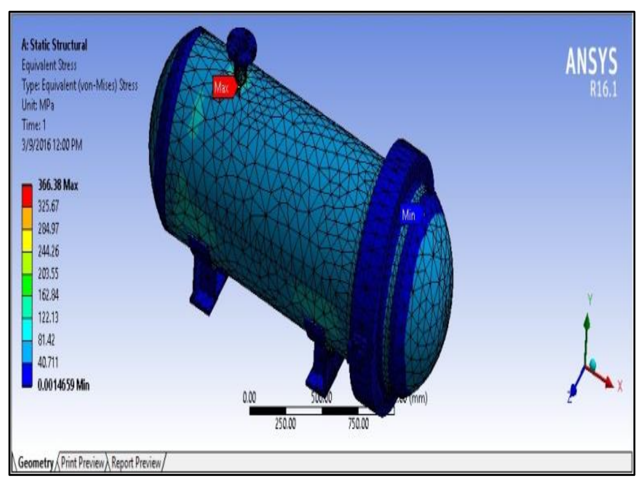

Fig 3: Total equivalent Von-Mises stress

Table 5: Equivalent stresses for different crack lengths

Sr. no.	Crack lengths(mm)	Equivalent stresses, (MPa)
1	5	366.38
2	6	393.34
3	7	413.30
4	8	413.44
5	9	415.74
6	10	423.34

As per the procedure of the analysis, the results obtained for the first technique i.e. equivalent stresses are summarized in the table 5.

Among the equivalent stresses for different crack lengths, the maximum value obtained is 423.34MPa for the given crack length of 10mm.The strength value of the same material with young's modulus is equal to 485MPa. It is found that ANSYS WORKBENCH equivalent Von Mises stress is low as compared to the strength of the material. This means that the material can withstand stresses more than applied, so the design is safe.

Stress Intensity Factor (K1)

The maximum value of stress intensity factor 'K1' in the direction-x obtained by using finite element tool ANSYS WORKBENCH 16.1 is 64.301MPamm0$^{0.5}$ for a given crack length of 5mm. The results of the SIF 'K1' obtained for the same boundary conditions and for different crack lengths are summarized in the table 6 below. The maximum stress intensity factor 'K1' from the table 11is 75.965MPamm$^{0.5}$ for a crack of length of around 10mm. The critical stress intensity factor of the material is 220 965 MPamm$^{0.5}$. So it has been proved that the design is safe. Even if there is any crack present on the surface then it will not further propagate.

Table 6: SIF, K1 for different crack lengths

Sr. No.	Crack lengths(mm)	SIF, K1 (MPa.mm$^{0.5}$)	SIF, K_{II}, (MPa.mm$^{0.5}$)	SIF,K3 (MPa.mm$^{0.5}$)	J-Integral, (MJ/mm^2)
1	5	64.301	**23.832**	36.142	**0.027288**
2	6	65.546	17.317	37.286	0.028559
3	7	69.727	14.028	39.642	0.032255
4	8	73.565	11.953	39.580	0.034750
5	9	74.874	10.526	39.868	0.035814
6	10	75.965	9.5584	41.006	0.037043

Stress Intensity Factor (K2)

The maximum value of stress intensity factor 'K2' in the direction-y obtained by using finite element tool ANSYS WORKBENCH 16.1 is 23.832965 MPamm$^{0.5}$ for a given crack length of 5mm. Among the different values obtained during the analysis, the maximum value obtained is 23.832 965 MPamm$^{0.5}$ for a given length of 5mm and When it is compared this value of 'K2' obtained from finite element tool with the 'K_{IC}' i.e. critical stress intensity factor or material toughness of the same material it comes out to be very less.

Stress Intensity Factor (K3)

The maximum value of stress intensity factor 'K3' in the direction-z obtained by using finite element tool ANSYS WORKBENCH 16.1 is 36. 965 MPamm$^{0.5}$ for a given crack length of 5mm. The stress intensity factor for z-direction is given by 'K3' for different crack lengths when subjected to same boundary conditions. Among the different values obtained during the analysis, the maximum value obtained is 41.006MPamm$^{0.5}$ for a given length of 10mm.

J-Integral (J)

The J-Integral for elastic material would be the same as that of stress concentration factor 'G' but the J-Integral for plastic material is a complicated concept. The value of J-Integral for the thin walled pressure vessel with a crack on the surface obtained by using finite element tool ANSYS WORKBENCH 16.1 is 0.027288 MJ/mm2 for a given length of crack 5mm.

The results obtained for J-Integral for different crack lengths by using the same boundary conditions is given in table.

4. Conclusion

The main conclusion of the study is clear that all the values are within safe limit, which means that if there is any crack present on the surface it will not propagate further. So we can conclude that the design of the pressure vessel is safe for operating conditions.

5. References

1. M.Perl, A. Nachum,"3-D stress intensity factors for internal cracks in an overstrained cylindrical pressure vessel-part II: the combined effect of pressure andautofrettage" Journal of Pressure Vessel Technology, 2001; 123-135.
2. Wang JA, Liu KC,"A new approach to evaluate fracture toughness of structural materials", Transactions of the ASME,534/.2004, 126
3. Stoychev S, Kujawski D," Crack tip stresses and their effect on stress intensity factor for crack propagation", Engineering Fracture Mechanics 75, 2007.
4. Youli Ma,"Evaluating stress distributions and stress intensity factors of a crack under mixed–mode conditions in aluminum alloy", International Conference on Measuring Technology and Mechatronics Automation, 2010
5. NidhiDiwedi, Veerendra Kumar, AshwaniShrivastava, Rajinareliya,"Burst pressure assessment of pressure vessel using finite element analysis: A review" Journal of Pressure Vessel Technology, 135/044502-1, 2013.
6. Rafael G. Savioli, Claudio Ruggieri, "Experimental study on the cleavage fracture behaviour of an ASTM A285 grade C pressure vessel steel" Journal of Pressure Vessel Technology, Vol.137/021206-1, 2015.
7. Christopher J. Evans, Timothy F.Miller," Failure Prediction of Pressure Vessels using Finite Element Analysis" Journal of Pressure Vessel Technology, vol.137/051206-1, 2015.