Failure Assessment Diagram Constraint Used for Integrity Analysis of Cylindrical Shell with Crack

Musthafa Akbar,[a,*] and Rachman Setiawan,[b]

[a)]*Mechanical Engineering, Universitas Riau, Indonesia*
[b)]*Mechanical & Aerospace Engineering, Institut Teknologi Bandung, Indonesia*

*Corresponding author: musthafa.akbar@lecturer.unri.ac.id

Paper History

Received: 23-November-2015
Received in revised form: 17-January-2016
Accepted: 20-January-2016

ABSTRACT

During its operation time, cylindrical pressure vessel could experience cracks. If this happens, the question is raised whether the pressure vessel could still be used or not, moreoever whether further treatment is required. In process and petroleum industry, an integrity analysis using Fitness For Service methodology is common, for instance referring to API 579/ASME FFS-1 2007 Code. Level 3 assessment within the Code requires a finite element simulation in order to generate both the evaluation point and the Failure Assessment Diagram (FAD) that serves as an acceptance criteria. Here, a parametric study based on the methodology given by the Code has been carried out to generate such result for the cases of internal longitudinal crack defect in a cylindrical shells for a number of common cases, in terms of thickness-to-radius ratio, crack size ratio, and crack aspect ratio. The evaluation of Stress Intensity Factor is determined through *J*-integral parameter found using a finite element analysis with a specially-meshed strategy incorporating the crack. The result of the model is first verified with that of the Code for a number of cases, before being used for parametric study. The model yields a relatively close comparison with that of the Code. A number of regressed equation was derived for several cases, and proposed to be used in integrity assessment of cylindrical shell. A procedure of using the parametric study result from this investigation is also outlined here.

KEY WORDS: *Integriy analysis, failure assessment diagram, J-integral, cylindrical shell.*

NOMENCLATURE

API	American Petroleum Institute
K_J	Elastic-Plastic Stress Intensity Factor
K_I	Elastic Stress Intensity Factor
E	Modulus Elasticity
J	Energy Release Rate
v	Poisson Ratio
σ_{ref}	Reference Stress
σ_y	Yield Strength
σ_c	Circumferential Stress
K_r	Brittle Fracture Ratio
L_r	Plastic Collapse Ratio
P_0	Collapse Pressure
P	Internal Pressure
a	Crack Depth
c	Crack Length
t	Thickness
R_i	Internal Radius

1.0 INTRODUCTION

Cylindrical shells are a common geometry of the structure that usually used in static mechanical equipment, e.g. pressure vessels, tank, boiler, and pipe. This geometry is used widely in industry due to excellence in rigidity, easy to manufacture and has a simple design. In design, cylindrical geometry commonly categorized as a thin shell. It means cylindrical geometry has a very large radius when compared to the thickness of the wall. Cylinder with radius to thickness ratio greater than 20 usually considered as a thin shell. Once shell assumptions are used in design and analysis, stress distribution along the thickness of the wall can be considered to be uniform.

During its operation time, mechanical equipment will encounter deterioration due to mechanical damage, i.e. crack, corrosion, pitting, laminations, shell distortion, and weld misalignment. Such damage will decrease the integrity of its equipment and further analysis is required to maintain reliability of equipment. Integrity analysis is an urgent issue, especially in industries with high safety and has a major impact to its surroundings.

Crack or crack-like flaw emerge in a cylindrical shell due to imperfection of material during production, fabrication, or in service. Excessive loading in the form of fatigue or impact gouges can be a trigger of crack initiation. Failure will occur when stress intensity factor along a crack tip is greater than the ability of material to sustain a stable crack.

Numerous investigations were made by researchers to develop methods for integrity analysis of cylindrical structure that contains crack. In 1982, Raju and Newman [1] started their work to find out stress intensity value of crack that emerge in cylindrical vessels. In this research, they use nodal force-method applied to surface crack at inside and outside of cylindrical vessels subjected to various loading conditions, namely; uniform, linear, quadratic and cubic. Good agreement is shown when their result compared to another methodology used by other researchers. Unfortunately, this paper, limited only to determination of stress intensity value, discussion about the integrity of cylindrical shell did not provide.

The most famous method used in integrity analysis of the structure contains crack is a failure assessment diagram constraint. Tipple and Thorwald [2] using this method to evaluate the end-of-life critical crack size for a fatigue crack growth analysis. The benefit of *J*-Integral method is used here to calculate stress intensity factor along the crack front under linear elastic and elastic plastic condition. This paper also describes two methodologies for generating Failure Assessment Diagram (FAD) based on Code and using numerical method. Both of the method are shown in a graph and used to evaluate the integrity of nozzle-cylindrical shell junction with a single corner crack.

Not only for crack, FAD approach also used for notched structural components. These are usually analyzed under the assumptions that notches behave a crack, providing over conservative results [3]. In this case, some modifications are needed to use the diagram for notched structure. Furthermore, Ainsworth [4] using FAD approach for high temperature defect assessment. The assessment method using procedure describes in this paper is straightforward to apply in pressurized plant.

The procedure of use and generate FAD slope already stated in some Code, e.g. API 579, BS 7910, SINTAP, and R-6. In these Codes, FAD slope is generated based on given equation. The equation can be used for all types of material with various size and type of crack. This condition tends to resulting over conservative results and further analysis need to be conducted. Specific FAD which depends on the specific crack size, material and geometry of component need to be generated, usually using finite element method.

The use of finite element method to generate FAD slope and assessment point of certain crack, need a hard effort. It's a time consuming and high cost process for modelling and create meshing of a large number of crack dimensions in a specified geometry of the shells. The main purpose of this paper are to provide an equation of FAD slope for certain size of crack dimensions, i.e. thickness ratio, aspect ratio, and crack depth ratio.

2.0 METHODOLOGY

2.1 Failure Assessment Diagram

Dowling and Townley [5] was introduced the concept of FAD in 1974. They introduce the relationship between brittle fracture and ductile fracture. The original concept this diagram was derived from the strip-yield plastic zone correction [6]. The strip-yield model has some limitations, for example, it does not account for strain hardening and residual stress. A general equation of this diagram was developed based on interpolation of two cases, namely linear elastic fracture and plastic collapse. A good agreement was shown when the result of their research compared to laboratory experiment.

A new concept of FAD was developed based on energy method. In this method, FAD was derived from Elastic Plastic Fracture Mechanics (EPFM) analysis using J-Integral solution. This diagram was constructed with account an effect of strain hardening and plasticity. There are some types of FAD, as reported by Ainsworth [7] in his paper. The most famous and universal model is the Option 3 Curve. In this model, a general equation was derived based on curve fitting of various types of material and crack dimensions. Therefore, this curve will give an over conservative result. It means, in some cases the curve is safe to use, but in other case a further and detail analysis need to be conducted.

The result of curve fitting that has been derived by Ainsworth [7] is shown in Equation (5). This equation is used widely in industry and accommodate in API 579 Code to build Level 1 and 2 FAD curve. In this curve, x-axis is Load Ratio *(Lr)* and y-axis is Brittle Fracture Ratio *(Kr)*. Load Ratio is the reference stress of certain crack compared to yielding point of the material, whereas Brittle Fracture Ratio is the stress intensity factor of elastic load compared to plastic load. *Kr* and *Lr* is determined using Equation (3) and (4). Relation between *Lr* and *Kr* is given in Equation (5) and (6). These equations can be used to generate FAD slope. In order to carried out analysis in both region of elastic and plastic, collapse pressure of cylindrical shell need to be calculated using Equation (7).

$$K_J = \sqrt{\frac{JE}{1-v^2}} \quad (1)$$

$$K_J = \sqrt{JE} \quad (2)$$

$$K_r = \frac{K_I}{K_J} \quad (3)$$

$$L_r = \frac{\sigma_{ref}}{\sigma_y} \quad (4)$$

$$f_1(L_R) = [1 - 0{,}14{L_r}^2][0{,}3 + 0{,}7\exp(-0{,}65(L_r)^6)] \quad (5)$$

$$K_{r|Lr=1} = \left[1 + \frac{0.002E}{\sigma_y} + \frac{1}{2}\left(1 + \frac{0.002E}{\sigma_y}\right)^{-1}\right]^{-0.5} \quad (6)$$

$$P_0 = \frac{\sigma_c\, t}{R_i} \tag{7}$$

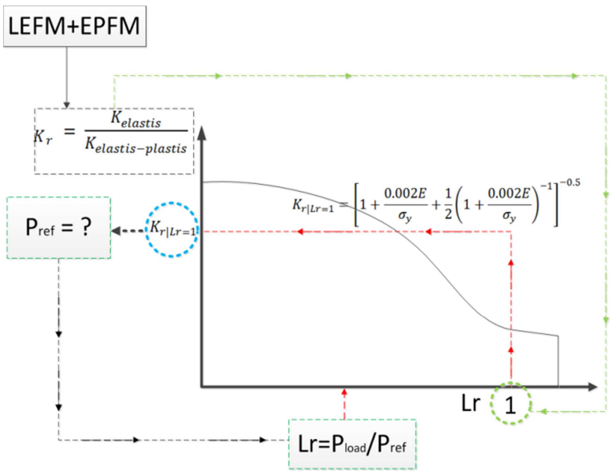


Figure 1: Scheme diagram to generate specific FAD using finite element simulation

Based on scheme diagram in Figure 1, Load Ratio is calculated using an equation which compare the value of internal pressure *(Pload)* which is simulated using finite element method to internal load when *Lr=Kr=1 (Pref)*. There are two type of finite element analysis were used here, namely Linear Elastic Fracture Mechanics (LEFM) and Elastic Plastic Fracture Mechanics (EPFM). In this diagram, the value of Brittle Fracture Ratio *(Kr)* are results of LEFM analysis compared to result of EPFM for variations of increased load, from elastic until plastic region.

In this research, 10 variations of cylinder dimensions and crack size were modelled and simulated. All of model are cylindrical shell geometry containing semi-elliptical crack with internal longitudinal position. This type and position of crack has a major impact to integrity of the shell. Length of cylindrical shell is chosen quite long to prevent effect of its length to the value of stress intensity factor. Shells with cylinder length to crack length ratio *(L/C)* ≥ *10* is considered to be used in this simulations [9]. Dimension of the models as shown in Table 1.

Table 1: Variations size of crack and cylindrical shell geometry in finite element simulations

No	*Ri*	*t*	*a*	*2c*	*t/Ri*	*a/c*	*a/t*
1	600	10	2	32	0.0167	0.125	0.2
2			6	24		0.5	0.6
3	200		2	32	0.05	0.125	0.2
4			6	24		0.5	0.6
5	100		2	32	0.1	0.125	0.2
6			6	24		0.5	0.6
7	20		2	32	0.5	0.125	0.2
8			6	24		0.5	0.6
9	10		2	32	1	0.125	0.2
10			6	24		0.5	0.6

Dimensions are in mm.

2.2Finite Element Modelling

Analytical method is used when simple crack problem need to be solve, whereas when complicated geometry is exist, numerical method is a solution. In general, there are two categories of numerical methods; one is based on displacement and other on energy. Displacement method only applicable for linear analysis, whereas later method can be used for elastic plastic analysis. *J*-Integral is one of energy method that widely use and adopted in commercial finite element software. Finite element modelling of crack geometry tend to difficult and need special attention on meshing strategy around crack tip. Mesh design for the crack tip region is a spider web mesh with elements concentrated at the crack tip, as shown in the Figure 2. The first ring of elements is made up of quadrilateral degenerated to triangles with several nodes coincident at the crack tip. Isoperimetric bricks elements with 27 node is used here. A typical 3D finite element model of crack geometry is shown in Figure 3.

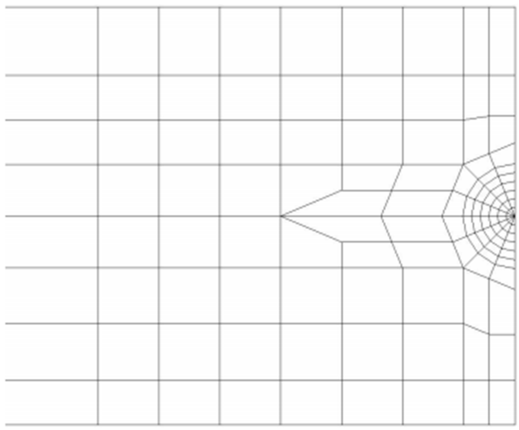


Figure 2: Spider web mesh design with elements concentrated at the crack tip [6]

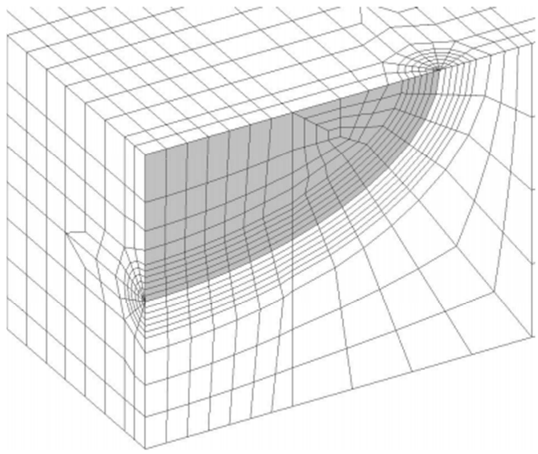


Figure 3: Typical 3D finite model surface crack [6]

In some Code, it is recommended to use a more accurate stress intensity factor calculation using *J*-integral. This method provided a solution to determine value of stress intensity factor either in elastic or elastic-plastic condition based on energy release rate. Another solution is based on displacement method and only available in elastic manner. In two dimensional case, *J*-Integral is define as follow [8].

$$J = \int_{\Gamma} \left(W dy - T \frac{\partial u}{\partial x} ds \right) \quad (8)$$

where

$$W = W(x, y) = W(\varepsilon) = \int_0^{\varepsilon} \sigma_{ij} d\varepsilon_{ij} \quad (9)$$

Γ in equation (8) is a close contour with counter clock wise direction, *T* is traction, $T_i = \sigma_{ij} n_j$, *u* is displacement in x-axis direction, and *ds* is an element of Γ. Based on above equations, close contour will have *J* value equal to zero. The position of *J*-contour along crack front is shown in Figure 4 and Figure 5.

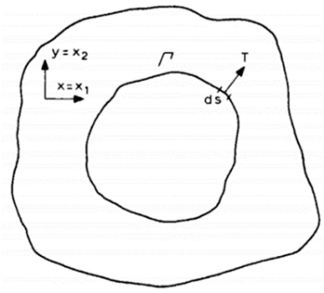
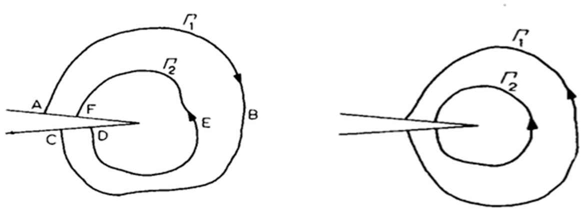


Figure 4: Definition of *J*-integral method [8]



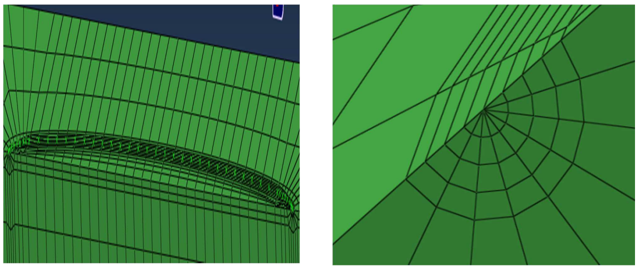
(a) (b)

Figure 5: *J*-Integral contour around crack front: (a) close contour with zero value, (b) two contour with same integral value [8]

3.0 RESULTS AND DISCUSSION

3.1Validation of the Results

The results of finite element simulations were done in this paper and compared directly with the Code. API 579 Code only provide required tables and an equation to calculate the value of Stress Intensity Factor (K_I) for certain geometry of crack and shell. Therefore, the value of *J*-Integral resulting from finite element simulations need to be converted into Stress Intensity Factor (K_I) using Eq. (1). Validation was take for a case of semi elliptical crack with position at inner wall of the shell. A cylindrical shell with thickness ratio (t/R_i)=0.0167, and a crack with (a/t)=0.2 and (a/c)=0.125 was modeled here. Finite element software was used to solve the *J*-Integral solution of 45 nodal position along crack front. The position of each nodal and meshing that used in this simulation is shown in Figure 6. The results of energy release rate of semi elliptical crack along crack front can be seen in Figure 7. Figure 8 show the comparison between the values of K_I resulting from finite element simulations to K_I provided by the API 579 Code. A good agreement was shown with the value of Sum Square Error (SSE) equal to 0.62 (Table 1).



(a) (b)

Figure 6: Typical mesh of crack front: (a) nodal position along crack front, (b) spider web mesh near crack tip.

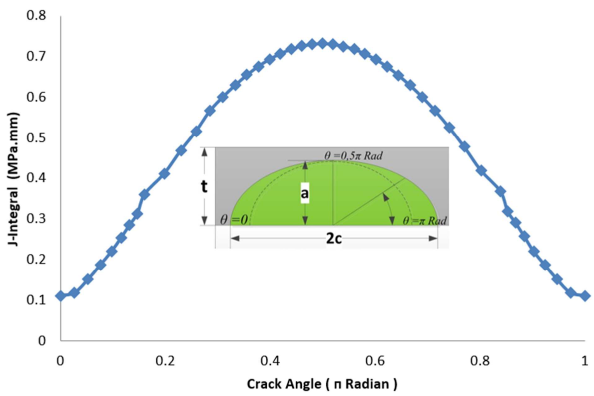


Figure 7: Results of finite element simulations in form of energy release rate (*J*-Integral) along crack front.

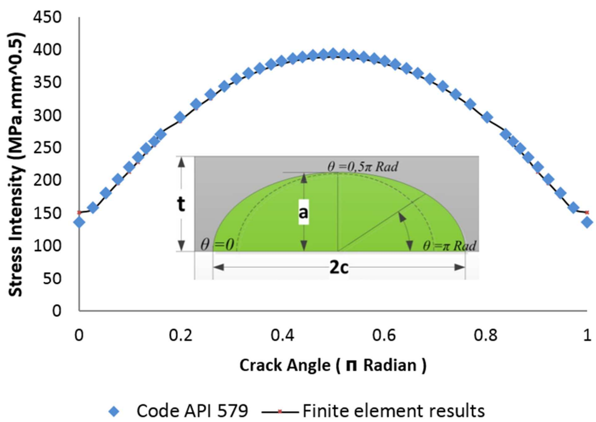


Figure 8: The results of finite element simulations compared to solutions from API 579 Code.

Table 2: The value of K_I as results of FE simulations and API 579 Code

Crack Angle (π rad)	FEM Results (ksi.in^0.5)	API 579 (ksi.in^0.5)	SSE
0	4.3555	3.9447	0.1688
0.0271	4.4884	4.5708	0.0068
0.0529	5.0888	5.2132	0.0155
0.0765	5.6603	5.8100	0.0224
0.0978	6.1486	6.3448	0.0385
0.1157	6.5907	6.7825	0.0368
0.1321	6.9938	7.1682	0.0304
0.1470	7.3295	7.5074	0.0317
0.1608	7.8673	7.8079	0.0035
0.1981	8.3997	8.5571	0.0248
0.2300	8.9671	9.1236	0.0245
0.2587	9.4050	9.5725	0.0280
0.2850	9.8490	9.9378	0.0079
0.3098	10.1426	10.2400	0.0095
0.3333	10.3845	10.4920	0.0116
0.3558	10.5897	10.7022	0.0127
0.3776	10.7621	10.8765	0.0131
0.3988	10.8968	11.0189	0.0149
0.4195	11.0063	11.1323	0.0159
0.4399	11.0896	11.2188	0.0167
0.4601	11.1480	11.2799	0.0174
0.4800	11.1829	11.3174	0.0181
0.5	11.1943	11.3327	0.0192
			SSE=0.6241

3.2 Parametric Study of FAD

Parametric study with variations of crack size and dimension of cylindrical shell also simulated here. There are 10 variation of models to be investigated, as shown in Table 3. All of models are semi elliptical crack and internal longitudinal position. In this position, crack face will be subjected to hoop stress from shell and it consider to be the most serious threat to integrity of the cylindrical shell.

Table 3: Models of cylinder made a specific FAD curve

No	*Ri*	*t*	*a*	*2c*	*t/Ri*	*a/c*	*a/t*
1	600	10	2	32	0.0167	0.125	0.2
2			6	24		0.5	0.6
3	200		2	32	0.05	0.125	0.2
4			6	24		0.5	0.6
5	100		2	32	0.1	0.125	0.2
6			6	24		0.5	0.6
7	20		2	32	0.5	0.125	0.2
8			6	24		0.5	0.6
9	10		2	32	1	0.125	0.2
10			6	24		0.5	0.6

Dimensions are in mm.

In FAD procedures, iteration process were conducted to find out collapse pressure of cylinder under elastic and elastic-plastic conditions. Using both of assumption, the value of energy release rate increase with variations of internal pressure. For all of cases, it can be seen in Figure 9 and Figure 10, maximum value of either *J*-Integral or Stress Intensity Factor always at deeper position of crack (crack angle=0.5 π Rad). It can concluded that the failure of structure will started at this position. Therefore, in depth investigations need to be carried out at this position.

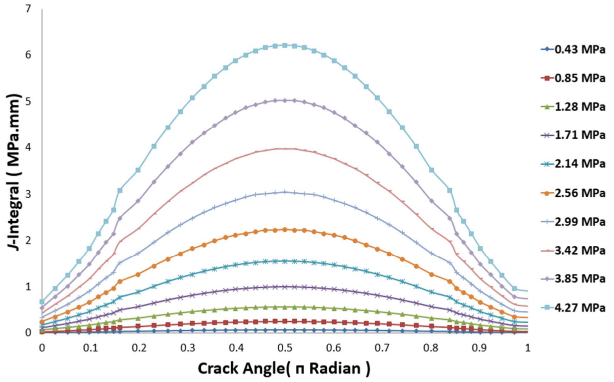


Figure 9: Results of finite element simulations in form of energy release rate (*J*-Integral) along crack front.

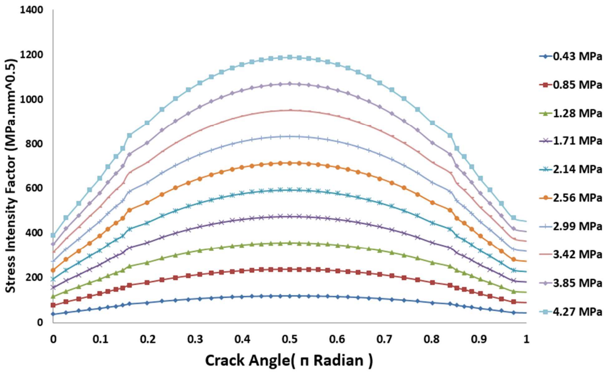


Figure 10: Results of finite element simulations in form of energy release rate (*J*-Integral) along crack front.

Failure Assessment Diagram can be created for every position along crack front. But then, maximum criteria is applied to anticipating the worst cases. In this study, two kind of simulation were taken separately. Under elastic condition, the models are subjected to variations of internal pressure which is below its collapse pressure. Collapse pressure of the model is predicted using Equation (7) and then validated using the results of finite element simulations. In the procedures of FAD, Brittle Fracture Ratio (Kr) is calculated based on the value of Stress Intensity Factor under elastic condition (K_I) compared to the value of Stress Intensity Factor under elastic-plastic condition (K_J), as shown in Eq. (3). To use this equation, the value of J-Integral either in elastic or elastic-plastic assumption need to be converted into K_I and K_J using Eq. (1) and (2). The value of Stress Intensity Factor under elastic condition (K_I) is calculated using 5 step of increased pressure. The value of internal pressure limited only up to calculated collapse pressure, therefore an extrapolation formula need to infer the value of Stress Intensity Factor above its collapse pressure, as shown in Figure 11- Figure 15.

For elastic-plastic assumption, large deformation algorithm involved within the software. Nonlinearities of material data in the area of plasticity generated using the Ramberg-Osgood equation. In this kind of simulation, internal pressure increased gradually up to 20 step. A close iteration step need to be refined in the area of nonlinearities, as shown in the Figure 11- Figure 15.

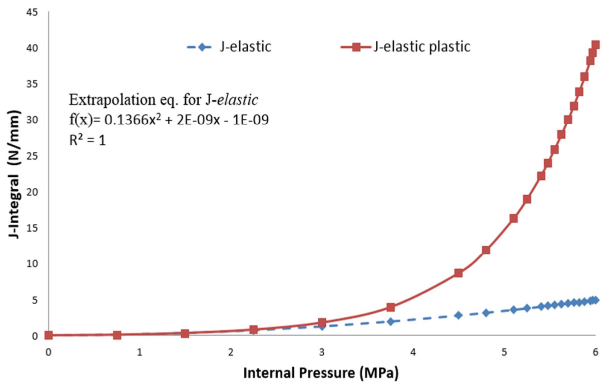


Figure 11: *J*-Integral results of cylindrical shell with t/Ri=0.0167 under two kind of simulation, elastic and elastic-plastic assumption

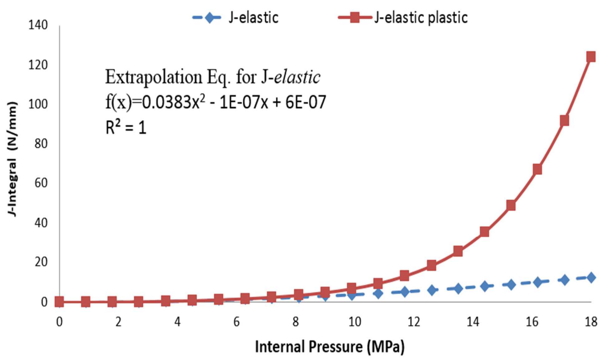


Figure 12: *J*-Integral results of cylindrical shell with t/Ri=0.05 under two kind of simulation, elastic and elastic-plastic assumption

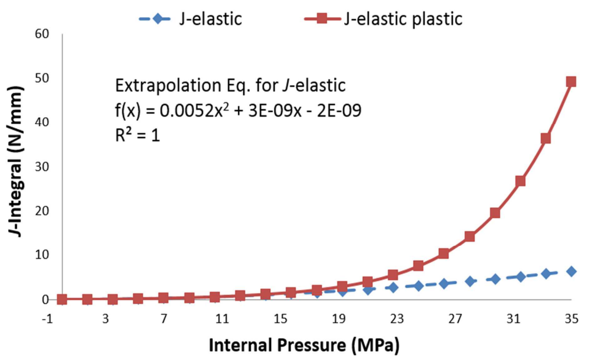


Figure 13: *J*-Integral results of cylindrical shell with t/Ri=0.1 under two kind of simulation, elastic and elastic-plastic assumption

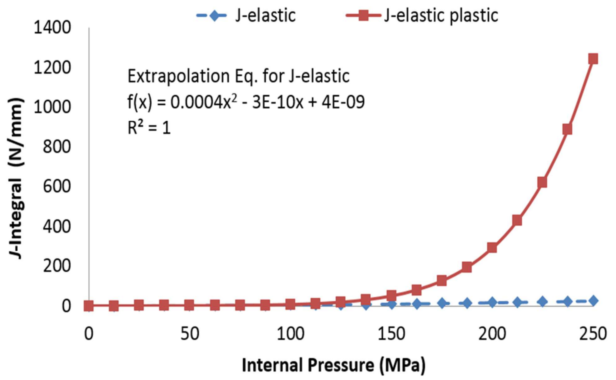


Figure 14: *J*-Integral results of cylindrical shell with t/Ri=0.5 under two kind of simulation, elastic and elastic-plastic assumption

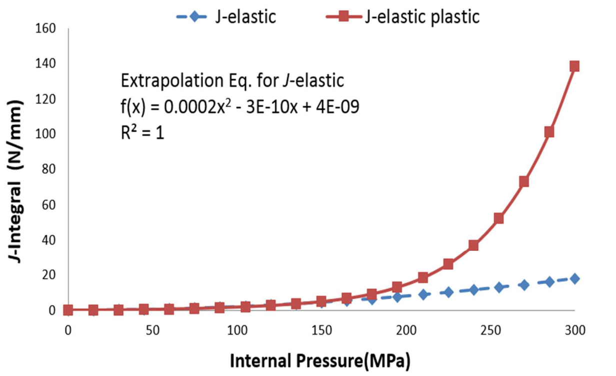


Figure 15: *J*-Integral results of cylindrical shell with t/Ri=1 under two kind of simulation, elastic and elastic-plastic assumption.

The ordinate axis of FAD is Load Ratio (L_r). This ratio is calculated based on reference stress and yielding point of material, as shown in Eq. (4). Another derived equation also can be used based on simplified methodology as provided in Figure 1.

3.3 Comparison of FAD

In order to obtain more rigorous analysis, failure assessment diagram were generated to build specific shell and crack-size Failure Assessment Diagram. This diagram can be used to assess feasibility operation of cylindrical shell containing crack with various size of crack and shell dimension. Failure Assessment Diagram then are generated for 10 configurations of cylinder and crack dimensions as shown in the Table 3.

The Failure Assessment Diagrams were generated using abovementioned methodology then compared with one created based on given equation in some literature and Code. The given equation (Eq.(4) is applied generally. It means, the equation acceptable to be used for all type and size of crack with various dimension and geometry of shell. In this study, comparisons were made to measure the validity of the equation when applied to certain size of crack and cylindrical shell. Finally, the result of simulations then plotted together with FAD which generated using general equation as shown in Figure 16.

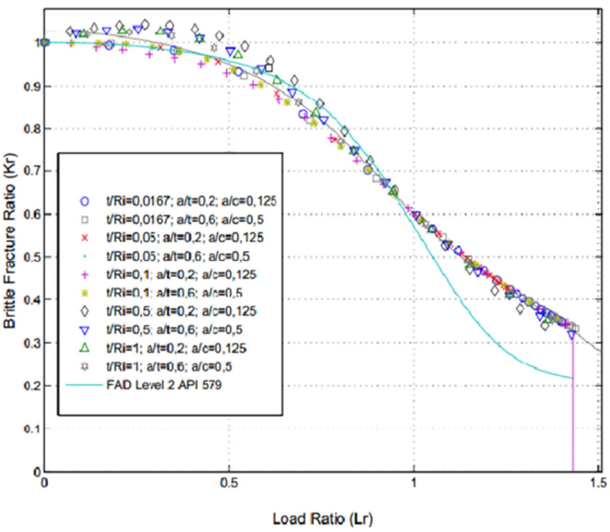


Figure 16: Comparison of FAD Level 2 in code with FAD generated based on FEM simulation results

Based on above diagram (Figure 16), there is a similarities or a close agreement between those slopes. FAD which generated using general equation given in the literature tend to relatively more conservative when compared to FAD which generated manually using finite element simulation. The conservatism degree is relatively higher in area with Load Ratio (Lr) greater than unity. It means, when assessment point of crack pointed out in this area, a more detail and refine analysis need to be conducted to prevent a failure.

In this study, regressed equations were derived based on data from the simulations. The equations in Table 4 can be used specifically for cylindrical shell containing crack with certain dimension.

Table 4: Regressed equations for FAD Level 3

No	t/Ri	Equations
1	1/60	$f(L_r) = [2{,}097 - 0{,}582{L_r}^2][0{,}363 + 0{,}116\exp(-1{,}333{L_r}^6)]$
2	1/20	$f(L_r) = [1{,}583 - 0{,}439{L_r}^2][0{,}484 + 0{,}152\exp(-1{,}344{L_r}^6)]$
3	1/10	$f(L_r) = [0{,}841 - 0{,}236{L_r}^2][0{,}921 + 0{,}271\exp(-1{,}374{L_r}^6)]$
4	1/2	$f(L_r) = [0{,}593 - 0{,}144{L_r}^2][0{,}104 + 1{,}538\exp(-0{,}044{L_r}^6)]$
5	1	$f(L_r) = [0{,}914 - 0{,}215{L_r}^2][0{,}694 + 0{,}451\exp(-0{,}997{L_r}^6)]$

4.0 CONCLUSION

This research describes the usage of Failure Assessment Diagram constraint for integrity assessment of cylindrical shell. The standard procedure of this method are stated in several code and widely used in industry. The equation of FAD slope that given by the Code are applied generally. It means, the equation can be used for various types, geometry and position of crack at any structural geometry. The consequence is the result of FAD curve usually tend to over conservative. To generate more accurate diagram, specific equation for each crack need to be developed.

In this research, finite element study with assumptions of linear elastic and elastic plastic fracture mechanics were carried out systematically to develop slope equations of FAD. The finite element model is first verified with that of the Code for a number of cases, before being used for parametric study. The model yields a relatively close comparison with that of the Code. Then, a total of 200simulations were conducted for 10 models of cylindrical shell containing crack. Based on this research, the following conclusions can be made:

1. Failure Assessment Diagram (FAD), which is generated using equation that given in Code, generally tend to over conservative when compared to specific FAD where the solutions resulted from finite element analysis.
2. Failure Assessment Diagram (FAD), which are generated using finite element simulations, tend to have a limited acceptance area in elastic-plastic region (0.4≤Lr≤1). Meanwhile, in the area toward plastic collapse region (Lr>1), this curve tend to more optimistic or wider acceptance area.
3. For practice purpose, the proposed equations in this paper can be used for integrity assessment of cylindrical shells containing crack.

REFERENCE

1. I. Raju and J. Newman Jr., “Stress-intensity factors for internal and external surface cracks in cylindrical vessels,” *Journal of Pressure Vessel Technology*, vol. 104, no. November. p. 293, 1982.
2. C. Tipple and G. Thorwald, “Using the Failure Assessment Diagram Method with Fatigue Crack Growth to Determine Leak-before-Rupture,” *2012 SIMULIA Community Conference.*, pp. 1–15, 2012.
3. S. Cicero, V. Madrazo, I. A. Carrascal, and R. Cicero, “Assessment of notched structural components using Failure Assessment Diagrams and the Theory of Critical Distances,” *Eng. Fract. Mech.*, vol. 78, no. 16, pp. 2809–2825, 2011.
4. R. A. Ainsworth, D. G. Hooton, and D. Green, “Failure assessment diagrams for high temperature defect assessment,” *Eng. Fract. Mech.*, vol. 62, no. 1, pp. 95–109, 1999.
5. A.R. Dowling, C.H.A. Townley, “The effect of defects on structural failure: A two-criteria approach”, International Journal of Pressure Vessels and Piping, Volume 3, Issue 2, April 1975, Pages 77-107.
6. Anderson, T.L., Fracture Mechanics: Fundamentals and Applications. Third Edition ed.1985, Danvers, United States: Taylor & Francis Group.
7. I. Milne and R. Ainsworth, “Background to and validation of CEGB Report R/H/R6—Revision 3,” *Int. Journal of Pressure Vessel and Piping*, vol. 32, pp. 105–196, 1988.
8. X. B. Lin and R. A. Smith, “Fatigue Growth Prediction of Internal Surface Cracks in Pressure Vessels,” *J. Press. Vessel Technol.*, vol. 120, no. 1, pp. 17–23, Feb. 1998.

9. J. Newman J. C. and I. S. Raju, “Stress-Intensity Factors for Internal Surface Cracks in Cylindrical Pressure Vessels,” *J. Press. Vessel Technol.*, vol. 102, no. 4, pp. 342–346, Nov. 1980.
10. API 579-1/ASME FFS-1. Recommended Practice for Fitness-For-Service, 2nd ed., American Society of Mechanical Engineers, Washington, D.C. June 5, 2007.