Finite Element Simulation on Crack Analysis of a Thick-Tube

A.M.Ahmad Zaidi [1*], M.M.Md Asif [2], I.Abdul Rahman[3], S.Rasool Mohideen[1], A.F.Ahmad Zaidi [4], N.H.Ahmad Zaidi[5] and M.S.Yusof [1]

[1]AdVeS, Faculty of Mechanical and Manufacturing Engineering,
Universiti Tun Onn Hussein Malaysia
[2]School of MACE,
The University of Manchester, United Kingdom
[3]Faculty of Civil and Environmental Engineering,
Universiti Tun Onn Hussein Malaysia
[4]School of Mechatronic Engineering,
Universiti Malaysia Perlis
[5]School of Material Engineering,
Universiti Malaysia Perlis

Corresponding author: mujahid@uthm.edu.my

Abstract

Most engineering failure began with cracks. Crack may caused by material defect, discontinuities in geometry or damage in service. Thus, Fracture Mechanics is introduced as a method for predicting failure of a surface containing a crack. This project is focusing on pre-existing crack with assumption that no microscopic defects are presents. Linear Elastic Fracture Mechanics (LEFM) is used to evaluate the Stress Intensity Factor (SIF) of the specimen. Further analysis is done by incorporating Elastic Plastic Fracture Mechanics (EPFM) to understand the crack growth over period of time. In this study, the important parameters in fracture mechanics such as Stress Intensity Factor (SIF), Crack Mouth Opening Displacement (CMOD), J Integral and stable crack growth are been investigated. A complex loading simulation of NKS-3 specimen is done using finite element modeling. The NKS-3 is a thick-tube used in Pressurized Water Reactor (PWR) which has a circumferential flaw on its inner surface. The cylinder is loaded with axial tensile load and internal pressure combined with thermal shock. A 2-Dimensional Axysimmetric-4 nodes element with focused mesh at the crack tip is employed in the simulation. Several analyses have been done using hardening data at different temperature. A comparison of solution is made for simulation with and without thermal load history. Further analysis showed that the stable crack growth is estimated to be around 3.0mm.

Keywords: Finite element, J integral, stable crack growth

1. INTRODUCTION

This project is mainly concern about the application of FEM in the study of fracture mechanics. In this project, a circumferential crack in a thick-walled cylinder is modeled to simulate the real geometry and loading conditions in Pressurized Water Reactor (PWR).

Most engineering failure began with cracks. Crack may caused by material defect, discontinuities in geometry or damage in service. Thus, fracture mechanics is introduced as a method for predicting failure of a surface containing a crack. This project is focusing on pre-existing crack with assumption that no microscopic defects are presents. Linear Elastic Fracture Mechanics (LEFM) is used to evaluate the Stress Intensity Factor (SIF) of the specimen. Further analysis is done by incorporating Elastic Plastic Fracture Mechanics (EPFM) to understand the crack growth over period of time [1-6].

A complex loading is applied to the model to simulate real situation loading conditions. This involves axial loading, internal pressure at high temperature and thermal shock due to cooling mechanism.

The NKS-3 specimen has been examined by more than eight assessment groups using different Finite Element Analysis (FEA) packages. Based on small-specimen J resistance curve for the NKS-3 material, the groups have predicted a stable crack growth in the range of 3.0 mm to 5.0 mm [5-6]. These analyses have a variation in geometric modelling, crack configuration, temperature distributions, loading history and J evaluation method.

2. NKS-3 GEOMETRY AND LOADING HISTORY

The NKS-3 experiment is carried out at MPA, Stuttgart (FRG), with a thick-walled hollow cylinder containing a circumferential flaw with average depth of 62.8mm on its inner surface. The cylinder has a thickness of 200mm and outer diameter of 400mm. An axial tensile load of 100 MN and internal pressure of 30MPa at 3300C are applied to the cylinder. These combined loads are used to simulate the stress conditions

in a nuclear Pressurized Water Reactor (PWR). The specimen is then experiencing thermal shock when cold water (200C) is sprayed towards the inner cylinder surface through evenly distributed nozzles. The full geometry of NKS-3 is in Figure 1(a) and (b). A measurement of stable crack growth is done and showed that an average crack extension is about 3.6mm [5-6].

The simulation has been carried out using finite element package ABAQUS/CAE. The J Integral and CMOD are sought for duration of 30 minutes after the loadings are applied. A change in temperature along the thickness over time creates thermal shock that contribute to the variation of J value.

A CAX4R (4 node bilinear axisymmetric quadrilateral, reduced integration) element is used in the simulation [7]. The 2 Dimensional axisymmetric model is generated using ABAQUS and shown in Figure 1(c).

A bilinear set of data which is temperature dependence is used as in table 1 [5-6].

Mechanical Properties (RPV Steel)	
Yield Stress (MPa)	$\sigma_y = 604.5 + 01106T$
Initial Modulus (MPa)	$E_o = 212600 - 86.09T$
Post-yield Modulus (MPa)	$E_T = 1209 + 2.114T$

*T = Temperature in ºC

Table 1: Temperature dependence properties for NKS-3
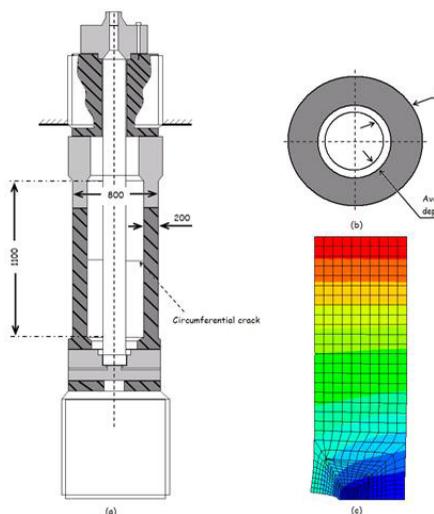
Figure 1: NKS-3 Geometry (a) side view (b) top

view (c) finite element model with displacement contour

3. J INTEGRAL EVALUATION IN ABAQUS/CAE

The J integral is usually used to characterize the energy release associated with crack growth. If the material response is linear, J integral can be directly related to the stress intensity factor. It is define as [7]

$$\bar{J} = \int_A \lambda(s)\, n \cdot H \cdot q \, dA \qquad (1)$$

where λ (s) is Virtual Crack Advance dA is surface element along a vanishing small tubular surface enclosing the crack tip or crack line, n is outward normal to dA, q is the local direction of virtual crack advance and H is given by,

$$H = \left(WI - \sigma \cdot \frac{\partial u}{\partial x} \right) \qquad (2)$$

where W is elastic strain energy for elastic material.

Ideally, J Integral is independent of the domain used given that the crack faces are parallel to each other. However, J Integral values estimated from different rings or contours may vary due to the approximation nature of finite element solution.

A large variation in the estimation suggests that a finer mesh is needed or the contour integral does not completely include the plastic zone.

The first few contours close to the crack tip may not be accurate. Thus, more contours are needed and the value that appears approximately constant from one contour to the next is taken.

4. ANALYSIS

The analysis that are carried out for NKS-3 specimen are divided into several parts,

1.   The evaluation of stress intensity factor at temperature 330oC without thermal shock. The evaluation is carried out using different methods to determine the average stress intensity factor.

2.   The second analysis involved a simulation of detailed loading history applied to NKS-3 using bilinear hardening data at an average temperature of 20oC, 50oC, 150oC and 355oC. A comparison of highest J Integral value is made between the temperatures.

3.   Further analysis is carried out using multilinear hardening data at 20oC. The solution is compared with second analysis with the same average temperature (20oC).

4.   The fourth analysis is carried out irrespective of thermal load history. Mechanical and thermal loading are applied simultaneously at a time instant. The results are then compared with the fourth analysis (with thermal loading history).

5.   The fifth analysis involved the application of mechanical loading followed by thermal loading (sequential loading). In this case, the intermediate temperature inputs have not been considered.

6.   The stable crack growth is determined by using data from fifth analysis above and J resistance curve for NKS-3 material (Sievers, 1987) in [5-6].

5. RESULTS

The stress intensity factor of NKS-3 specimen at 330oC is evaluated to be around 123 MPa. This analysis is done without the application of thermal load and using Linear Elastic Fracture Mechanics (LEFM).

A simulation of detailed loading history of NKS-3 has been done using bilinear hardening data at an average temperature of 20oC, 50oC, 150oC

and 330oC. Figure 2 illustrated the J Integral against time at each temperature. The highest J Integral value is 427.8 KN/m at time 600s using average temperature of 20oC. At 330oC, the maximum J value is 377 KN/m. In general, the J Integral value is decreasing when hardening data corresponding to higher temperatures is used.
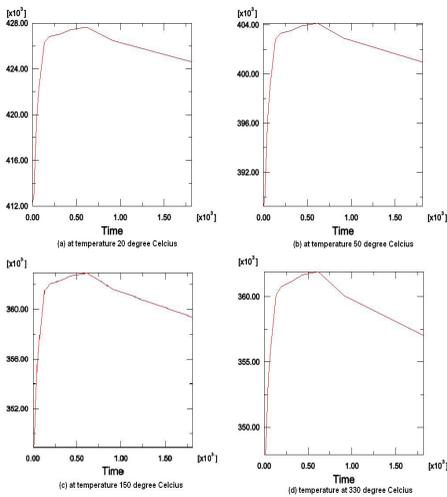
Figure 2: J Integral against time by using bi-linear hardening data at temperature (a) 200c (b)500C (c)1500c and (d) 3300C

The following analysis is carried out but using multi-linear hardening data at 20oC(Figure 3). It is observed that the maximum J value is lower compared to the same simulation (at 20oC) in previous analysis.
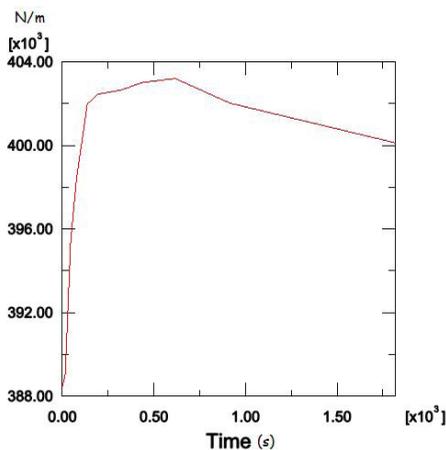
Figure 3: J Integral against time using multi-linear hardening data at 20oC

It is observed that there is a significant difference in J Integral and CMOD value between simulation with and without thermal loading history. The J Integral and CMOD values are higher for analysis with respect to thermal load history. This indicated that the way axial and pressure load are applied during the simulation will affect the solution. The highest J Integral value is observed at 600s for both analyses.

The stable crack growth is determined from the maximum J value from the plot of J Integral against time (figure 4).
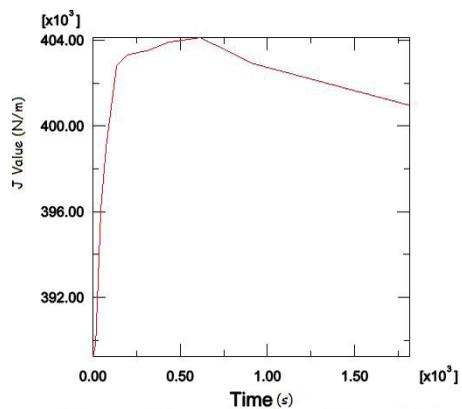
Figure 4: J Integral against time for NKS-3 with thermal load history

The corresponding stable crack growth is evaluated at 3mm as in figure 5.
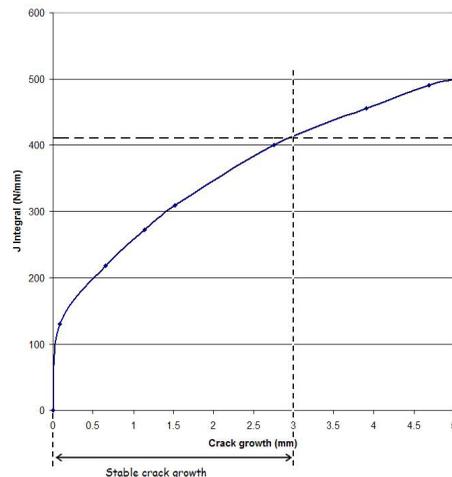
Figure 5 : J resistance curve for NKS-3 experiment

This value may differ from previous researches carried out by various assessment

groups [5-6]. This may happen due to different geometry and FE package used in the simulation. However, the stable crack growth obtain in this simulation is within the expected range which is 3mm to 5mm.

6. CONCLUSION

The FE package, ABAQUS/CAE is capable of simulating complex loading in fracture mechanics problems. It is a helpful tool to evaluate fracture mechanics parameter such as CMOD, J value and stable crack growth.

There is a significant different in solution for NKS-3 simulation with and without thermal load history. The J Integral and CMOD are higher for simulation with respect to thermal load history.

The stable crack growth of NKS-3 is found to be 3.0 mm at 600s with maximum J value of 404 N/m. This is within the range as done by previous assessment groups[5-6].

ACKNOWLEDGEMENT

REFERENCES

[1] T. Itoh, "*J*-integral estimate for biaxially stressed Mode I cracks based on COD strain", *International Journal of Fatigue*, 21, pp.1087-1097, 1999.

[2] R. Echle, G.Z. Vojiadjis, "Simulation of damage evolution in a uni-directional titanium matrix composite subjected to high cycle fatigue", *International Journal of Fatigue*, 21, pp.909-923, 1999.

[3] C.Y. Hou, "Simultaneous simulation of closure behavior and shape development of fatigue surface cracks", *International Journal of Fatigue*, 30, pp.1036-1046.

[4] A. Bacila, X. Decoopman, R. Vatavu, G. Mesmacque, M. Voda, V.A. Serban, "Computer simulation of fatigue crack propagation under random loading conditions", *International Journal of Fatigue*, 29, pp. 1772-1780, 2007.

[5] I.C Howard, M.A. Sheikh, *Simulation of Nuclear safety Tests (NKS-3 & NESC)*, Technical Note, Department of Mechanical & Process Engineering, University of Sheffield, 1994.

[6] M.M.Md Asif, A.M. Ahmad Zaidi, "Finite element simulation of a cracked thick-walled hollow cylinder", *Proceeding of MRG 2006 IC*, University of Salford, June 2006, pp.178-190.

[7] ABAQUS Analysis User's Manual Volume II Version 6.5, ABAQUS Inc, 2004 (pdf format).