Prediction of Crack Initiation Load in Throughwall Circumferentially Cracked Pipes Using 3D Cohesive Zone Model

Rehan.M.Siddiqui*, Dattatray.N.Jadhav, Nilesh.R.Raykar
Department of Mechanical Engineering Sardar Patel College of Engineering, Mumbai, India.

* Corresponding author: Tel: 91-9699723153; Email: s.rehan91@gmail.com
Manuscript submitted February 6, 2015; accepted July 8, 2015.
doi: 10.17706/ijmse.2015.3.3.183-192

Abstract: The present study deals with the use of cohesive zone model to predict crack initiation load in throughwall circumferentially cracked pipes. Crack propagation is first simulated using 2D Cohesive Zone Model (CZM) on a three point bend (TPB) specimen to determine the set of cohesive parameters. The same sets of parameters are used to predict the initiation of crack in throughwall circumferentially cracked pipes by 3D finite element analysis. The pipes were modelled using shell-solid coupling to save computational time and memory. The study predicted crack initiation load for three pipe configurations using 3D CZM with an error of 6 to12%.

Key words: Cohesive zone model, crack initiation, fracture, pipes.

1. Introduction

The possibility of predicting fracture in pressure vessels and piping has always been an issue of interest to researchers. It is one of the driving forces which had led to the development of fracture mechanics. This study is of particular interest to meet the high safety and reliability norms in design of equipment for nuclear power plants, Integrity assessment of these structures involves the use of Leak Before Break (LBB) concept which aims at the application of the fracture mechanics principle to demonstrate that pressure components are very unlikely to experience sudden catastrophic break without prior indication of leakage, in case of piping system investigation. This assessment requires detailed fracture mechanics analysis of different piping components, e.g. straight pipes, elbows and branch tees. In structural integrity assessment, an accurate and reliable way to evaluate fracture behaviour of components containing crack-like defects is a full-scale experimental test [1], [2]. However, from a cost-effectiveness perspective, it is economically unfavourable. One efficient way to replace such extensive test programmes is to use finite element (FE) damage analyses. Piping components are mostly made of ductile material. Damage in such materials exhibits in three stages viz. crack initiation, stable crack growth and instability. For leak-before-break demonstration it is important to predict the two stages i.e. Crack initiation and development of instability. Finite element analysis can be used to predict crack initiation and crack growth up to instability and beyond.

For the prediction of crack initiation load in pipes schemes based on J-integral and COD (popularly known as GE/EPRI method) are available. These schemes were first evolved for 2D geometries [3] and then for pipe geometry with throughwall crack [4] and surface crack [5]. Afterwards, various researchers [6]-[11]

worked on the application of the method for different crack geometries and loading conditions. Recently crack initiation was predicted in through-wall circumferentially cracked pipes and through-wall circumferentially cracked elbows under bending load based on the J-integral concept using elastic plastic fracture mechanics (EPFM) analysis [12], [13]. Han *et al.* [14] simulated ductile fracture of full-scale circumferential cracked pipes using 3D fem based on stress-modified fracture strain model. Some other approaches to simulate fracture are based on continuum damage mechanics approach (e.g., Needleman and Tvergaard, 1987) one of such approach is the cohesive zone model (CZM).

The cohesive zone model, based on an idea of Dugdale [15] and Barenblatt [16], has found increasing interest and applications in the past 20 years, because of its robustness, its limited set of parameters and broad range of possible applications. The CZM is a phenomenological model, it incorporates the mechanical processes that occur on crack propagation. Unlike other crack propagation methods of FEM e.g. Virtual crack closure technique, Node release technique, it does not require calculation of stress singularity at the crack tip this simplifies the crack growth calculations. Within the cohesive zone model, the damage and failure of a structure is modelled along the predetermined crack path, by a damage-free bulk material and special interface elements called as cohesive elements in which the material separation takes place. The CZM does not represent any physical material, but describes the cohesive forces which occur when material along crack path is being pulled apart. The damage behaviour of the material within cohesive elements is described by the so-called traction-separation law (TSL), which contains two parameters, namely the maximum traction sustainable by the element, $\sigma 0$, and a maximum opening separation $\delta 0$, at which the element totally fails. Beside the two parameters $\sigma 0$ and $\delta 0$, a third quantity is defined by area below the TSL curve, which represents the cumulative energy dissipated by the cohesive element upto total failure. It is also known as the cohesive energy $\Gamma 0$. The cohesive energy represents the amount of energy supplied to the material during the period of crack initiation to the end of crack extension. The cohesive energy is assumed to be equal to the energy release rate in fracture mechanics [17]-[19]. Cui et al [20] showed that cohesive zone model can be used predict the initiation of a crack and its subsequent growth. Cornec et al [21] validated the use of cohesive zone model for crack simulation on three quite different materials in the form of CT specimens covering a wide size range, of a tensile panel with a surface crack, and of a MT specimen with a through crack. Jadhav *et al* [22] predicted both mode-I and mode-II stable crack growth using 2D CZM within CT specimen. Recently some investigations have been carried out on crack simulation using cohesive elements in 3D. Chen *et al* [23] investigated the effect on transfer of cohesive parameters from 2D to 3D model on a CT specimen. Giuliese *et al* [24] simulated fatigue crack growth on a DCB specimen using 3D CZM and evaluated energy release rate using contour integral. Tardif *et al* [25] simulated stable crack propagation in steel at 1173 K using a large displacement 3D cohesive element with eight nodes. Evangelista *et al* [26] formulated and implemented a 3-D cohesive zone for mode I separation in a SENB specimen made up of cementitious materials, which took into account for the thermodynamics of the irreversible crack opening process. Danielsson *et al* [27] performed parametric study on beams with a hole, by nonlinear 3D FE-analyses, using CZM based on plasticity theory.

Most of these researches were limited to the specimen level, in this paper an attempt has been made to apply the cohesive zone models to the TPB specimen as well as pipe component. The present study has two parts. In the first part, crack growth is simulated in 2D on a TPB specimen made from the SA333Gr6 pipe material to determine the set of cohesive parameters for crack growth simulation. In the second part the cohesive parameters determined from the 2D analysis are applied for 3D crack simulation in pipes with different crack geometries.

2. Rack Growth in TPB Specimen

2.1. Specimen Geometry

The three point bend specimen was of the pipe material SA333Gr6 carbon steel. The details of the TPB specimen are given in Table 1

Table 1. Details of TPB Specimen [28]

Sr. no	Width of the specimen w(mm)	Crack depth/width (a/w)	Thickness t (mm)
1	25.06		0.513
8.00			

2.2. Material Parameters

The true stress-strain curve derived from the uniaxial tensile test on specimen made up the pipe material is shown in Fig. 1 [28].

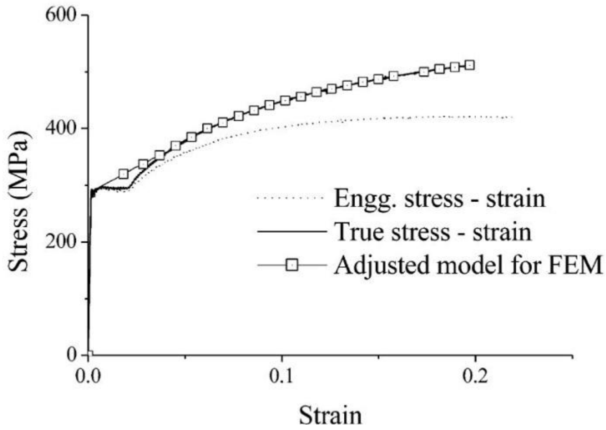


Fig. 1. Stress-strain diagram for tensile samples machined from 219 mm OD pipes [28]

Table 2. Summarises the Material Properties

Material properties of SA333 Gr6 steel [28]	
Yield stress σ_0	288MPa
Ultimate tensile stress σ_u	420MPa
Young`s Modulus E	203GPa
Poisson`s ratio μ	0.3

2.3. Simulation of Crack Growth in TPB Specimen

To simulate the crack growth the TPB specimen was modelled in ABAQUS 6.10. Details of the study are presented below.

2.3.1. Modelling

The TPB specimen was modelled in 2-D.The specimen was modelled as per the dimensions given in Table 1. Two sides of the TPB specimen on either sides of the crack were modelled separately and then these FE models were attached together using a cohesive layer of negligible thickness. Since only a small amount of crack growth is to be studied, the length of cohesive layer is taken 10mm from the crack tip, the rest of the portion is just merged to obtain a single part. The material properties for the TPB specimen are taken from Table 3. Incremental plasticity is used as per the data obtained from Fig. 1 and isotropic hardening law is assumed. The material properties for cohesive elements are taken based on the traction separation laws two laws were separately employed i.e. exponential and trapezoidal.

Plane stress (CPS8R) elements were used for the body of TPB specimen and cohesive elements (COH2D4) for the crack growth zone. A fine mesh near the crack front was employed. Element size of 0.2mm is used along the crack front. Fig. 2a shows the finite element mesh for the entire TPB specimen. Fig. 2b shows mesh near the crack front. The entire mesh consists of 50 linear quadrilateral elements of type COH2D4 and 16632 quadratic quadrilateral elements of type CPS8R. To simulate the experimental procedure, the roller supports of TPB specimen was modelled using appropriate displacement constraints and on the top centre node a uniform displacement in the y direction was specified.

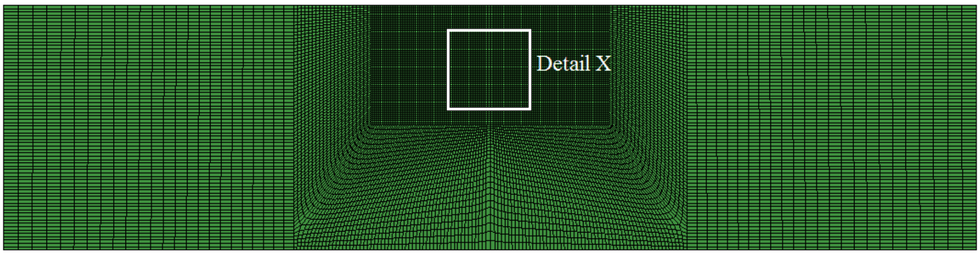
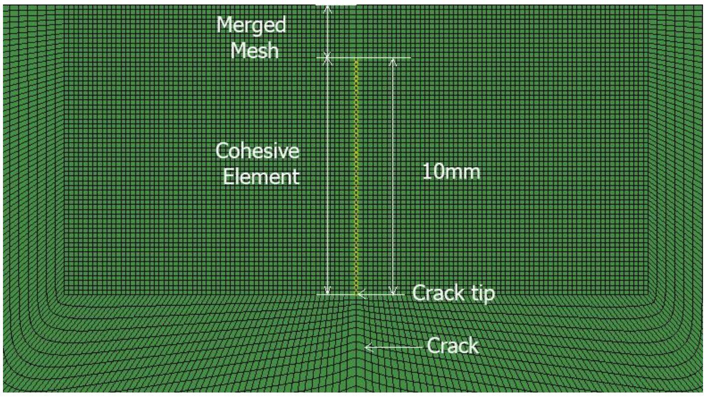


Fig. 2 (a). FE discretization



Detail X

Fig. 2 (b). Detail near crack tip

2.3.2. Traction-separation law and cohesive parameters

For the ductile materials literature suggests (e.g., Scheider and Brocks, 2003a) use of exponential or constant variation of normal traction with the relative normal displacement. The exponential traction-separation law as shown in Fig. 3 was taken here in the following form:

In the exponential TSL dependence of the traction on the separation is given by [29]

$$T(\delta) = T_0 z \frac{\delta}{\delta_0} e(1 - z(\delta/\delta_0)) \text{ ...... where } z = 16e/9,$$

The corresponding cohesive fracture energy Γ_0 is given, $\Gamma_0 = \int_0^{\delta_0} \sigma(\delta) d\delta$

Hence the corresponding fracture energy is given by $\Gamma_0 = \frac{9}{16}\sigma_0\delta_0$...... (1)

For crack growth simulation, the two parameters, i.e., maximum normal traction/cohesive strength σ_0 and as the maximum/critical normal separation δ_0, are required to be given as input. Based on experimental stress-strain data [28] stress corresponding to final fracture was assumed to be ultimate tensile stress, this was taken as the cohesive strength (σ_0). J at crack initiation i.e. 220 N/mm as calculated

[28] was taken as cohesive energy Γ_0. Substituting value of Γ_0= 220N/mm and σ_0= 420Mpa in Eq. (1), δ_0 value is obtained, which is used as input in FE analysis.

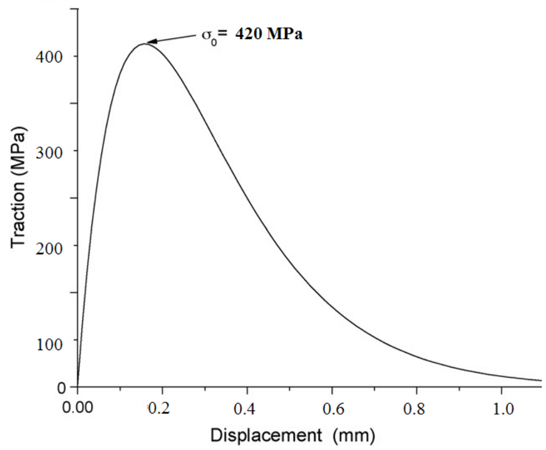


Fig. 3. Exponential cohesive law

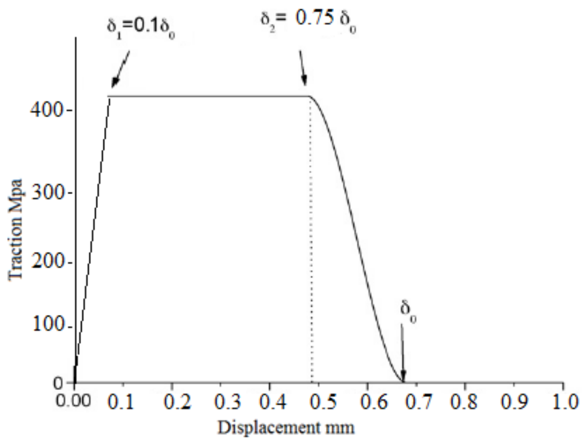


Fig. 4. Trapezoidal cohesive law

Keeping the cohesive parameters, i.e., cohesive energy (220N/mm) and maximum stress (420 MPa) the same, analysis was again done for the same specimen considering a constant variation of traction with separation [21] as shown in Fig. 4. This variation corresponds to δ_0=0.65mm.

2.3.3. Results

The comparison of the experimental and numerical variation of load vs load line deflection is shown in Fig. 5. There is a good match between the experimental data and predicted numerical values obtained from the FE analysis. Table 3 shows the maximum load values for various cases

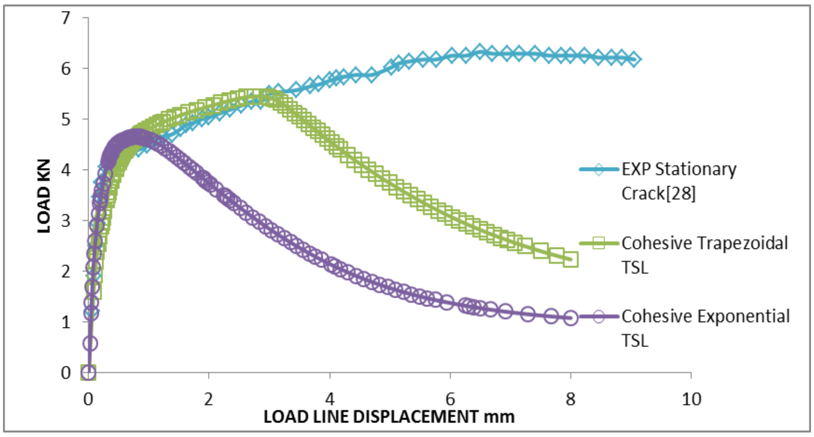


Fig. 5. Load vs load line displacement for TPB specimen

Table 3. Maximum Load Values for TPB Specimen

Cohesive law	Experimental load value	FEM load values	Error
Exponential	6.25 KN	4.6 KN	-26%
Trapezoidal	6.25 KN	5.6 KN	-10.4%

It is seen that in case of the numerical solutions the load decreases after reaching the maximum value this is because crack growth is taken in account in case of the numerical solution while the experimental values are for stationary crack. The decrease in load is due to decrease in the stiffness of the specimen on crack extension. Negative error indicates over estimation of the load. One reason behind the over estimation may be the high stress values obtained at the load point nodes in the FEM procedure. Since the error in the maximum load predicted by trapezoidal TSL is less, the trapezoidal TSL is considered for analysis on pipes.

3. Simulation on Throughwall Circumferentially Cracked Pipes

For the 3D analysis the cohesive parameters obtained from the 2D analysis on TPB specimen are used.

3.1. Specimen Geometry

Test components consist of straight pipes made of SA333Gr6 carbon steel material circumferential crack at the centre of its outer span as shown in Fig. 6, these pipe specimens are subjected to four point bending load. The geometric details of the test specimens are given in Table 4.

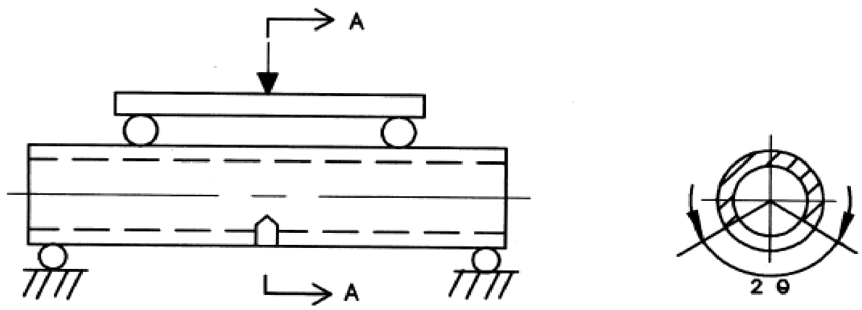


Fig. 6. Geometry of pipe specimen.[1]

Table 4. Details of Test Components [1]

Test no.	Outer dia. (mm)	Wall thickness(mm)	Span (mm) Outer	Span (mm) Inner	Crack angle (degrees) Initial	Crack angle (degrees) After fatigue pre-crack
SPBMTWC8-1	219	15.15	4000	1480	60	65.6
SPBMTWC8-2	219	15.10	4000	1480	90	93.9
SPBMTWC8-3	219	15.29	4000	1480	120	126.4

3.2. Modelling

The length of pipe is 4m with a diameter of 219 mm; hence to reduce the FE model size the pipe is modelled using shell-solid coupling. The part of the pipe near the crack tip is modelled using solid elements and rest using shell elements. The length of the solid portion is considered as $5\sqrt{Rt}$ (R=Radius of the pipe, t=thickness of the pipe). The limit of $5\sqrt{Rt}$ is drawn from the definition of influence of distance of discontinuity defined in ASME pressure vessel and boiler code [31].The crack was modelled considering the pre-cracked geometry.

The domain is discretised using solid and shell elements, The cohesive layer is discretised cohesive element (COH3D8); with an element length of 2 mm. the details of the mesh for specimen SPBMTWC-1 are as follows: Total number of elements: 121773, 5136 quadratic quadrilateral elements of type S8R 114912 linear hexahedral elements of type C3D8R, 1725 linear hexahedral elements of type COH3D8. Fig.7 shows the mesh along the crack front along with the shell-solid coupling.

3.3. Boundary Conditions

Since the pipe is subjected to four point bending, the lower edges of the pipe are supported by rollers using appropriate displacement constraints at its outer span, and displacement control loading is given at the upper nodes situated 1480mm apart. Fig.8 shows the boundary condition on the pipe model.

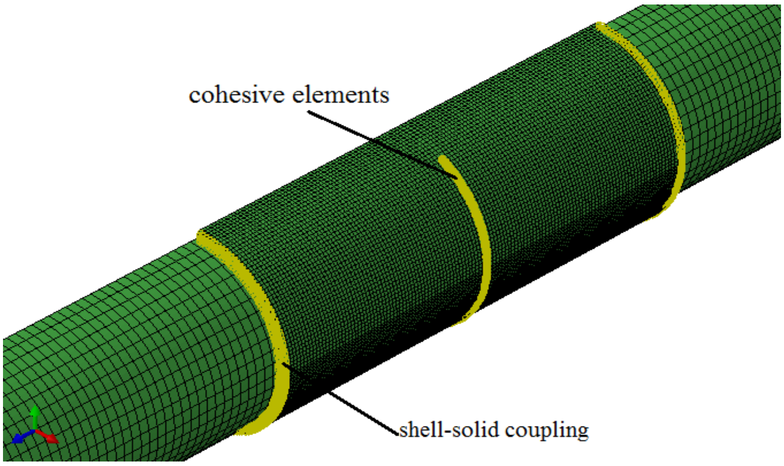


Fig. 7. Finite element mesh on pipe specimen SPBMTWC-1

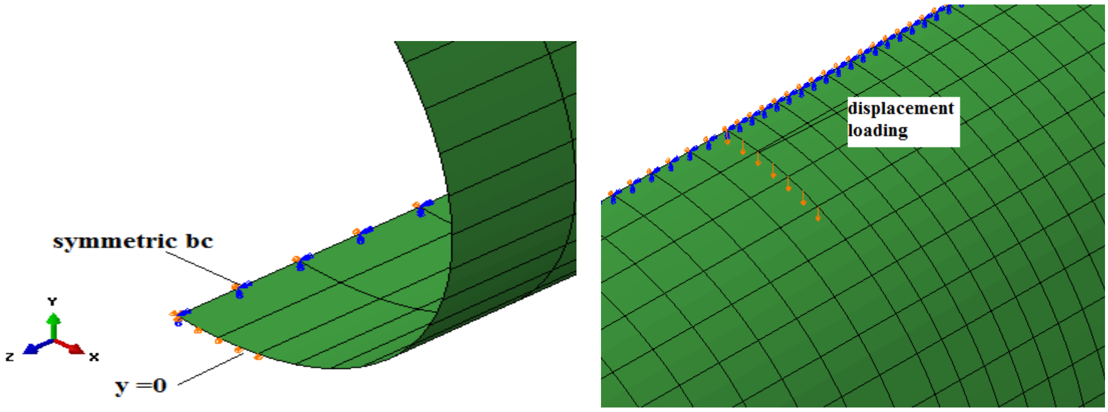


Fig. 8. Boundary condition on the pipe model

3.4. Results and Discussions

Fig. 9 shows the load vs load-point displacement curves for various pipes.

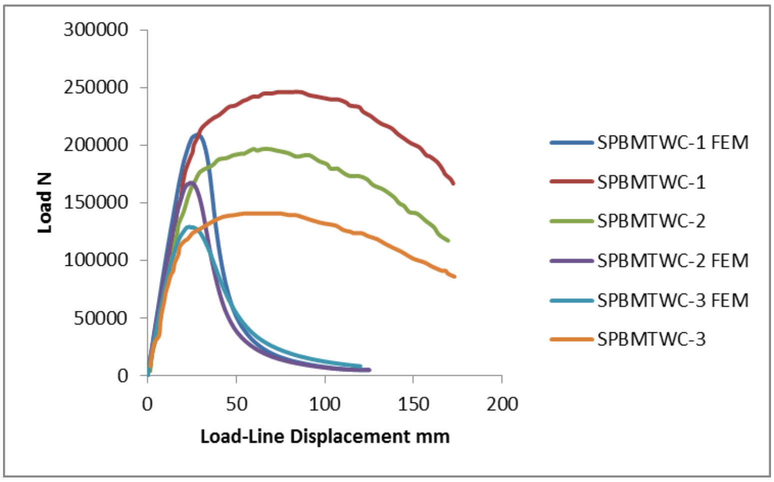


Fig. 9. Experimental load vs. load point displacement curves for pipes.

The time increment corresponding to the first cohesive element deletion was recorded and the load corresponding to this increment was taken as the crack initiation load. Based on this the details of the crack initiation load for three cracked pipe configurations are given in Table 5.

It is seen from the results that the load vs load line displacement has good agreement with the experimental values upto the start of crack extension After crack initiation there is rapid reduction in the load in case of the CZM. One reason for this deviation after correct prediction of crack initiation load is the value of cohesive energy used.

Table 5. Crack Initiation Load for Various Pipe Geometries.

Pipe configuration	Crack initiation load (KN)		Error
	Experiment	FEM	(%)
SPBMTWC8-1	194	206.88	-6.63
SPBMTWC8-2	148	166.4	-12.23
SPBMTWC8-3	116	128.87	-11.09

The cohesive energy was taken equal to the J at crack initiation this assumption still remains a paradox for some researchers. The fracture energy release rate from the cohesive zone model computation is generally not equal to the cohesive energy Γ_0. The difference depends on the cohesive law and disappears only in an elastic specimen. The maximum deviation exceeds 40% [30]. In 2D CZM, very fine mesh 0.2mm along the crack front is applied, in 3D CZM, however, due to computational constraints, we cannot use sufficiently fine discretization. This constraint also has an influence on the accuracy of prediction of load vs load line data. In case of the experimental procedure strain rate of 0.05mm/sec was applied, while in the numerical solution due to limitation of computational resources a strain rate of 1.875 mm per time step was applied. In case of the experimental tests on pipes there was out of plane crack growth while in case of FEM simulation only in plane crack growth was considered. The transferability of various fracture parameters from specimen to component level has always been an issue in fracture mechanics. The cohesive parameters that were used for simulation on pipes were determined from simulation on the TPB specimen, and were directly used for simulation on pipes. Due to these reasons the results deviated from the experimental data.

4. Conclusion

From the present study on TPB specimen and throughwall circumferentially cracked pipes it can be concluded that the cohesive zone models can be used to predict the crack initiation load with reasonable accuracy. However a fine mesh along with smaller load increment can be used to obtain a good matching between FEM and experimental load vs load line displacement. A study based on transferability of cohesive parameters from specimen to component level is desired to predict the load-vs load line data with good accuracy.

References

[1] Chattopadhyay, J., Dutta, B. K., & Kushwaha, H. S. (2000). Experimental and analytical study of three point bend specimen and throughwall circumferentially cracked straight pipe. *International Journal of Pressure Vessels and Piping*, 77, 455-471.

[2] Gandhi, P., Saravanan, M., Vishnuvardhan, S., Pukazhendhi, D. M., Raghava, Sahu, G., M. K., Chattopadhyay, J., Dutta, B. K., & Vaze K. K.(2013). Experimental fracture studies on carbon steel elbows with and without internal pressure. *International Journal of Pressure Vessels and Piping*, 1-7.

[3] Kumar, V, German, & M. D, Shih, C. F. (1981). An engineering approach for elastic–plastic fracture analysis. *Topical Report, Electric Power Research Institute*, *Palo Alto, CA*. EPRI-NP-1931, Project 1287-1.

[4] Kumar, V, German, M. D, Wilkening, A. W, deLorenzi H, & Mowbray, D. (1984). Advances in elastic–plastic fracture analysis. *Final Report, Electric Power Research Institute, Palo Alto, CA*, EPRI-NP-3607.

[5] Kumar, V, & German, M. D. (1988). Elastic–plastic fracture analysis of throughwall and surface flaws in

cylinders. EPRI-NP-5596, Electric Power Research Institute, Palo Alto, CA.

[6] Klecker, R, Brust, F. W, & Wilkowski, G. M. (1986). NRC leak-before-break analysis method for circumferentially throughwall cracked pipes under axial plus bending loads. *United States Nuclear Regulatory Commission* NUREG/CR 4572.

[7] Brust, F. W. (1987). Approximate methods for fracture analysis of throughwall cracked pipes. *United States Nuclear Regulatory Commission*, NUREG/CR-4583.

[8] Rahman, S, & Brust, F. W. (1992). An estimation method for evaluating energy release rates of circumferential throughwall cracked pipe welds, *Engineering. Fracture Mechanics*, 43, 417–430.

[9] Kim Y. J., Huh N. S., Park Y. J., & Kim Y. J. (2002). Elastic–plastic J and COD estimates for axial through-wall cracked pipes. *International Journal of Pressure Vessel and Piping*, 79, 451–464.

[10] Mohan, R., Krishna, A., Brust, F. W., & Wilkowski, G. M. (1998). J-estimation schemes for internal circumferential and axial surface cracks in pipe elbows. *Journal of Pressure Vessel Technology, ASME Trans*, 120, 418–423.

[11] Chattopadhyay, J., Tomar, A. K. S., Dutta, B. K., & Kushwaha, H. S. (2005). Elastic–plastic J and COD estimation schemes for throughwall circumferentially cracked elbow under in-plane closing moment, *Engineering Fracture Mechanics*, 72, 2186–2217.

[12] Sanjeev, S., & Ramachandra, M. D. S. (2004). Elastic–plastic fracture mechanics based prediction of crack initiation load in through-wall cracked pipes, *Engineering Structures*, 26, 165–172.

[13] Sanjeev, S., & Ramakrishnan, N. (2007). Characterizing the crack initiation load in circumferentially through-wall cracked elbows under bending load. *International Journal of Pressure Vessels and Piping*, 84, 493–501.

[14] Han, J. J., Ryu, H. W., Kim, Y. J., Kim, N. H., & Kim, J. H. (2014). Ductile fracture simulation of full-scale circumferential cracked pipes: (I) Carbon steel, *Procedia Materials Science, 3*, 855 –860.

[15] Dugdale, D. S., (1960). Yielding of steel sheets containing slits. *Journal of Mechanics and Physics of Solids*, 8100-8104.

[16] Barenblatt, G. I. (1962). The mathematical theory of equilibrium cracks in brittle fracture. *Advances in Applied Mechanics*, 7, 55–129.

[17] Scheider, I., & Brocks, W., 2003b, (2003), Simulation of cup-cone fracture using cohesive model, *Engineering. Fracture Mechanics*, 70, 1943–1961.

[18] Needleman, A. (1990). An analysis of decohesion along an imperfect interface*, International Journal of Fracture,* 42, 21–40.

[19] Tvergaard, V, & Hutchinson, J. W. (1992). The relation between crack growth resistance and fracture process parameters in elastic–plastic solids. *Journal of Mechanics and Physics of Solids*, 40, 1377–1397.

[20] Cui, W., & Wisnom, M., A combined stress-based and fracture-mechanics-based model for predicting delamination in composites. *Composites*, 24, 467–474.

[21] Alfred C., Ingo S., & Karl-Heinz S. (2003). On the practical application of the cohesive model. *Engineering Fracture Mechanics,* 70, 1963–1987.

[22] Jadhav, D. N., & Maiti, S. K., (2010). Characterization of stable crack growth through AISI 4340 steel using cohesive zone modeling and CTOD/CTOA criterion. *Nuclear Engineering and Design*, 240, 713–721.

[23] Chen, C. R., Kolednik, O., J. Heerens, & Fischer, F. D. (2005). Three-dimensional modeling of ductile crack growth: Cohesive zone parameters and crack tip triaxiality. *Engineering Fracture Mechanics,* 72, 2072–2094.

[24] Giuliese, G., Pirondi, & A., F. Moroni. (2014). Cohesive zone model for three-dimensional fatigue debonding/ delamination. *Procedia Materials Science,* 3, 1473 –1478

[25] Tardif, N., Combescure, A., Coret, & Matheron, M. P. (2010). Stable crack propagation in steel at 1173 K: Experimental investigation and simulation using 3D cohesive elements in large-displacements. *Engineering Fracture Mechanics*, 77, 776–792.

[26] Francisco, E. Jr., Jeffery, R. R., & Sérgio, P. P. (2013). Three-dimensional cohesive zone model for fracture of cementitious materials based on the thermodynamics of irreversible processes. *Engineering Fracture Mechanics*, 97, 261–280.

[27] Henrik D., & Per J. G. (2014). Fracture analysis of glued laminated timber beams with a hole using a 3D cohesive zone. *Engineering Fracture Mechanics*, 124–125,182–195.

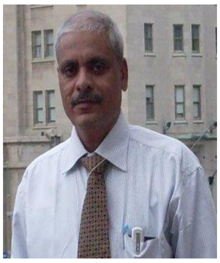
[28] Pavankumar, T. V., Chattopadhyay, J., Dutta, B. K., & Kushwaha, H. S. (2002). Transferability of specimen J-R curves to straight pipes with throughwall circumferential flaws. *International Journal of Pressure Vessels and Piping*, 79, 127-134.

[29] Scheider, I., & Brocks, W. (2003). The effect of the traction separation law on the results of cohesive zone crack propagation analyses. *Key Engineering. Materials*, 251-252, 313–318.

[30] Huang Y., & Li. X. (2014). Effects of the cohesive law on ductile crack propagation simulation by using cohesive zone models. *Engineering Fracture Mechanics*, 126, 1–11.

[31] ASME Boiler and Pressure vessel code (2010).

Rehan.M.Siddiqui was born on 23 February, 1991 in Mumbai India. He completed his primary education from St.Blaise High School Mumbai in 2006. He completed his Bachelor of Engineering in Mechanical from University of Mumbai in 2012. He then pursued Master of Technology in Mechanical Engineering discipline subjects from Sardar Patel College of Engineering Mumbai in 2014. He is currently working as an Assistant Professor in Department of Mechanical Engineering at Rajiv Gandhi Institute of Technology Mumbai. His research interest includes Stress Analysis, Finite Element Analysis, and Fracture Mechanics.



Dattatray.N.Jadhav was born on 27 January, 1967 in Maharashtra India. He completed his Bachelor of Engineering in Mechanical from Walchand College of Engineering Sangli Maharashtra. He then pursued Master of Technology in Mechanical Engineering discipline from Indian Institute of Technology Madras. He has been a PhD scholar at Indian Institute of Technology Bombay, His research work includes the characterization of stable crack growth using cohesive zone modelling. He is currently working as an Associate Professor in the department of Mechanical Engineering at Sardar Patel College of Engineering Mumbai. His research interests include Stress Analysis, Machine Design, Finite Element. Analysis, Fracture Mechanics. He has guided many research projects related to Stress Analysis Finite Element analysis at graduate and post graduate levels.

Nilesh.R.Raykar was born in Maharashtra India. He completed his Bachelor of Engineering in Mechanical from Veermata Jijamata Technological Institute Mumbai in 1988. Het then pursued Master of Technology in Mechanical Engineering discipline from Indian Institute of Technology Bombay in 1990. He obtained his doctor of philosophy (PhD) from Institute of Technology Bombay-Monash research academy in 2013. His research work includes the modelling and characterization of Hydrogen assisted stress corrosion cracking. Before obtaining his PhD he has worked in the Product Design Department of Larsen and Tubro group of companies Mumbai for a period of around 15 years. Currently he is working as a Professor department of Mechanical Engineering at Sardar Patel College of Engineering Mumbai. His research interests include Stress Analysis, Machine Design, Finite Element Analysis, Fracture Mechanics. He has guided many research projects related to Stress Analysis Finite Element analysis at graduate and post graduate levels.