18th International Conference on Structural Mechanics in Reactor Technology (SMiRT 18)
Beijing, China, August 7-12, 2005
SMiRT18-G05-8

ELASTIC-PLASTIC BEHAVIOUR OF A NUCLEAR PIPE ELBOW WITH AXIAL THROUGH WALL CRACK AT CROWN UNDER IN-PLANE BENDING

K.M. Prabhakaran*
Reactor Safety Division,
Bhabha Atomic Research Centre,
Mumbai-400 085, INDIA.
Phone: :+91 22 25593502
Fax: +91 22 25505151
E-mail: kmprabha@apsara.barc.ernet.in

V. Venkat Raj
Department of Mechanical Engineering
Bharati Vidyapeeth College of Engineering,
Sector 7, CBD, Navi Mumbai 400 614, INDIA.
Phone: +91-22-27708554
Fax: +91-22-27573196
E-mail: vvrajindra@yahoo.com

ABSTRACT

16" pipe elbows are used in the primary heat transport (PHT) system of Indian Pressurised Heavy water Reactors (PHWRs). Cracking at crown of elbows under bending is an important consideration in design. The structural integrity assessment of elbow requires knowledge of fracture parameters like Stress intensity factor, J-integral, crack mouth opening displacement etc. In earlier experimental studies, a full scale 16-inch carbon steel (SA 333 Gr. 6) elbow used in the PHT system, was subjected to fatigue loading. As a consequence, the elbow had a through wall axial crack at crown having a length of 104 mm. In the present study, the behaviour of this axially cracked 90-degree, long radius nuclear pipe elbow under in-plane closing moment was studied using 3D non-linear finite element analysis with WARP3D software. Limit load, load line displacement, crack mouth opening displacement and J-integral were evaluated for this elbow. The FE model consists of elbow with extended straight pipes and the geometry and thickness are exactly same as that used in the experiment (measured). The comparison of predicted FE results with experimental values were found to be good. This paper presents the details of the FE study and the results obtained, along with the comparisons.

Keywords: Elbow, Fracture, through-wall crack, Finite element analysis, J-integral.

1. INTRODUCTION

Nuclear power plants are required to have safety related structures, systems and equipments designed to maintain the intended function. High energy carbon steel pipes and elbows are extensively used in piping systems of power plants. Elbows exhibit significant material and geometric non-linear behavior, when subjected to severe loadings. When a thin-walled (diameter to thickness ratio > 10) 90° long radius elbow (R_b/R_m = 3.0) is subjected to an in-plane moment, ovalisation of the circular section into an elliptical one introduces high bending stress (circumferential component) at the crown and a crack may initiate at the crown. This has been confirmed by analytical (Kano et al., 1997) and the experimental work (Muller et al., 1985). A pipe bend failure experiment conducted by Diem et al., (1989), for in-plane bending moment, developed longitudinally oriented cracks in the elbow crown and the final crack length was around 67% of the elbow centerline. Sixteen inch pipe elbows are used in the primary heat transport (PHT) system of Indian Pressurised Heavy water Reactor (PHWR). Cracking at crown location of piping elbows under bending moment is considered as important from the design point of view. The structural integrity assessment of elbows requires a knowledge of fracture parameters like Stress intensity factor, J-integral, crack mouth opening displacement etc. Hence it is important to study the behaviour of axially cracked elbows under in-plane bending moment. In many of the finite element based studies the computed results have not been confirmed by experimental data. Hence, in this present study attempt is made to compare the results with the available experimental data.

2. NEED AND SCOPE OF THE PRESENT WORK

Griffiths (1979) performed some experiments on healthy and cracked elbows, to find the effect of cracks on limit loads. It was summarized by him that for short radius elbows with axial defects, the limit loads do not reduce substantially; but significant reduction can occur for long radius bends with large cracks. Behaviour of elbows in the plastic regime is quite different under in-plane closing and opening moment due to the large deformations involved and the difference in geometric stiffness of the elbow cross section. Solutions for limit loads for long radius healthy elbows under opening and closing moment are given by Shalabay et al. (1997, 1998) and Chattopadhyay et al. (2000). Similarly for long radius cracked elbow the limit load solutions are provided by Zahoor (1991). Ductile fracture experiments were conducted by Naoki et al. (1997) on through wall cracked elbows under high temperature.

In an earlier experimental study (Seetharaman et al., 1999), a full scale carbon steel (SA 333 Gr. 6) elbow (16” SCH. 120) used in the Primary Heat Transport (PHT) piping system, was subjected to fatigue loading. As a consequence, the elbow had a through wall axial crack at crown having a length of 104 mm. As described above, even though much research have been performed on elbow under in-plane bending moment, the problem of simulating the test results with finite element analysis accurately has not been fully solved. This paper presents the details of a finite element study performed on an axially cracked 16-inch elbow and comparison of the prediction with experimental results.

3. DETAILS OF THE STUDIES PERFORMED

In the present study, the behaviour of this axially cracked 90-degree, long radius nuclear pipe elbow under in-plane closing bending moment was studied. The dimensions of the elbow were exactly same as the one used in the experiment. A three-dimensional non-linear finite element analysis was carried out using WARP3D (Gullerud et al., 2001) software. Limit load, load line displacement, crack mouth opening displacement and J-integral were evaluated for this elbow. The finite element analysis was performed by assuming the material to follow the Ramberg-Osgood relation. The predicted finite element results were compared with the experimental values. The behaviour of load-load-line displacement and crack mouth opening displacement are also presented. Apart from these, analyses were also performed for the evaluation of J-integral and the behaviour of the elbow was also studied.

4. GEOMETRY OF THE ELBOW

In the present study an actual 16” elbow with through wall crack (TWC) at the crown subjected to in-plane closing moment was considered. The finite element study simulated the exact geometry and loading of the experiment performed earlier (Seetharaman et al., 1999). The actual thickness values of the elbow measured during the experiment were found to be varying from 35.47 mm to 41.52 mm. The measured variation of the elbow thickness during the test is shown in Table-1. The loading setup for the experiment is shown in *Fig. 1*. The elbow was connected to the loading frame through extended pipes on both sides, having a length of 500 mm each. The finite element model consists of elbow with extended straight pipes where the geometry and thickness are exactly the same as that used in the experiment. The crack angle considered was 9.77° (i.e. crack length of 104 mm). The geometrical details of the elbow are given below,

Elbow size	=	16 inch NB SCH 120.
Outer Diameter	=	406.0 mm
Thickness (min), t	=	35.47 mm
Thickness (max), t	=	41.0 mm
Axial Crack length	=	104.0 mm
Extended pipe length	=	500.0 mm

Material properties of SA 333 Gr. 6 are as follows

Young's Modulus, E	=	203 GPa
Yield Stress, σ_y	=	302 MPa
Ultimate Tensile Stress, σ_u	=	450 MPa
Elongation (% over 25 mm GL)	=	36.7

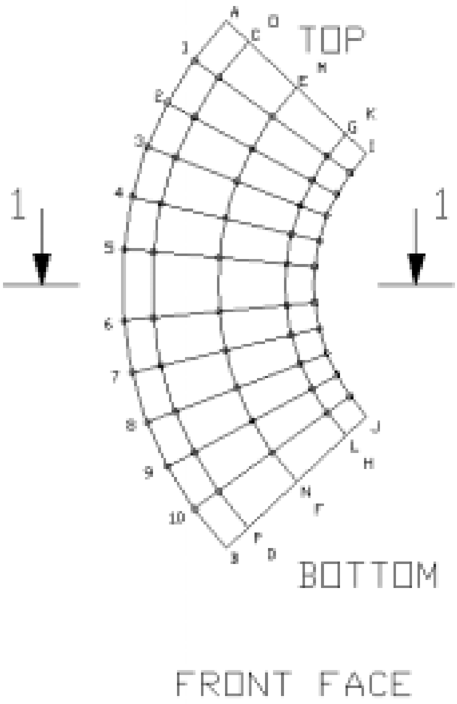
Chemical composition of SA 333 Gr. 6 carbon steel material used in the present studies (Singh et al., 1998) is as follows,

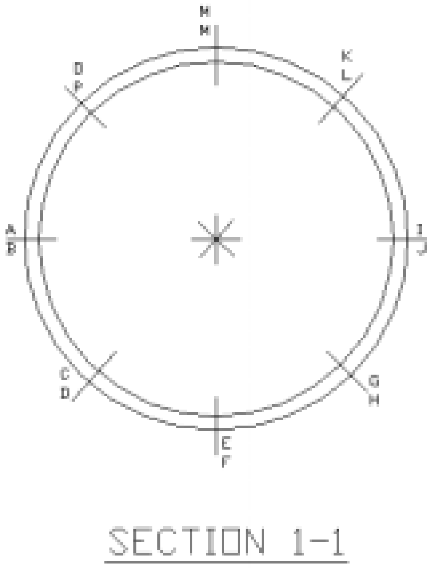
Specification	C	Mn	Si	Ni	Cr	Al	Cu	P	S	N
ASTM	≤ 0.3	≤ 1.35	0.15-0.30	≤ 0.4	≤ 0.25	≤ 0.04	≤ 0.3	≤ 0.025	≤ 0.025	≤ 0.013
Present Material	0.14	0.9	0.25	0.062	0.081	0.1	0.054	0.016		0.01

Table -1 Measured thickness (in mm) of Elbow at different locations

Location	1	2	3	4	5	6	7	8	9	10
AB	39.16	37.62	37.09	36.49	36.6	36.20	35.59	35.85	37.71	38.16
CD	36.88	36.89	36.75	36.16	35.47	35.51	36.10	36.32	35.71	35.56
EF	35.95	37.1	39.11	39.75	40.21	39.58	40.23	40.03	40.03	38.27
GH	37.93	38.54	38.25	37.90	37.41	37.60	37.57	37.69	38.44	38.37
IJ	40.69	41.33	40.38	39.01	37.95	38.26	38.75	40.03	40.69	40.65
KL	38.95	39.95	39.98	38.97	38.46	38.60	38.79	39.48	40.02	39.62
MN	37.67	39.34	40.56	41.12	40.62	40.91	41.52	41.05	39.98	39.06
OP	38.76	38.36	38.03	37.78	37.46	37.29	37.29	37.23	37.42	37.92

Note: Measurements as per the locations given below.





5. METHOD AND FINITE ELEMENT MODEL

Three-dimensional non-linear finite element method (FEM) analysis using WARP3D (Gullerud et al., 2001) code was used to study the elbow. The model included the elbow and the 500 mm extended pipes on both sides. The full size finite element model was generated using the in-house software ELBOWGEN (Prabhakaran et al., 2002) using three dimensional 20 node solid elements available in the computer code (Gullerud et al., 2001). The model includes full elbow along the length and 360 degrees in the circumferential direction. The elbow was divided into 12 divisions along axial direction and 22 divisions along the circumference. Twenty-two division along the circumference is considered in order to take care of the varying measured thickness of the elbow. The crack tip region was modelled by 'spider-web' mesh configuration, having 32 wedge shaped elements. The wedge-shaped elements were constructed by collapsing one face of the twenty-node solid element to a line along the crack front. The length of the edge of the collapsed element, i.e. at crack tip is less than 7% of the half crack length. Fourteen point integration order has been used in the study. One end of the extended pipe was applied with a moment, which was increased in each step of the analysis. The other end of the elbow was completely restrained, as in the experiment.

To check the adequacy of the finite element model, a mesh convergence study has been performed with one and two elements along the wall thickness. The maximum variation of load-line displacement and crack mouth opening displacement for the case of one element as compared to two elements along the elbow wall thickness was found to be very less. Similarly *J-integral* variation for the two cases mentioned above is less than 7%. The applied load at this condition was 84.3% of the limit moment given by Chattopadhyay et al. (2000). Moreover, in the present study, the integration order used for the present model (20 noded element) is 14 point integration which gives more accurate prediction than the 2x2x2 integration order. The *J-integral* was evaluated for 7 rings around the crack tip. The level of path independence of *J-integral* for this model was confirmed.. This path independence in *J-integral* for different domains is one measure of the adequacy of mesh refinement and convergence to equilibrium (Baker, 1983). For the domains very close to the crack tip full path independence may not be observed (Hutchinson, 1968). Hence the first path, which is very close to the crack tip, was not considered. The accuracy of the *J-integral* solutions obtained with different domains is estimated to be of the order of 2%. Hence it was decided to consider only one element through the elbow wall thickness. The final model consisted of 3226 nodal points and 640 elements. The finite element model is shown in *Fig. 2*.

Fig. 1 Test Setup of Elbow with axial TWC at Crown (Seetharaman et al., 1999)

The geometric and material non-linearities were included in the study. The material was assumed to follow the stress-strain response represented by the well-known Ramberg-Osgood model, given as, $\varepsilon/\varepsilon_0 = (\sigma/\sigma_0) + \alpha(\sigma/\sigma_0)^n$. Here ε and σ are true strain and true stress, and σ_0 is a normalising stress, E is the modulus of elasticity, $\varepsilon_0 = \sigma_0/E$ is a normalising strain and α and n are model parameters usually chosen from a best fit of actual test data. The properties considered were $\alpha = 19.826$, n = 3.887, σ_0 = reference stress (taken as flow stress) i.e. $\sigma_0 = (\sigma_y + \sigma_u)/2$, E = 203 GPa, σ_y = 302 MPa, σ_u = 450 MPa. The above mentioned values were taken from the results of the experimental studies performed by Singh et al. (1998).

6. LOADING

The elbow was subjected to in-plane closing load, applied in monotonically increasing order with definite steps, at the end of the extended straight pipe. The closing load was applied by distributing the load equally at all the nodes at the end of the straight pipe.

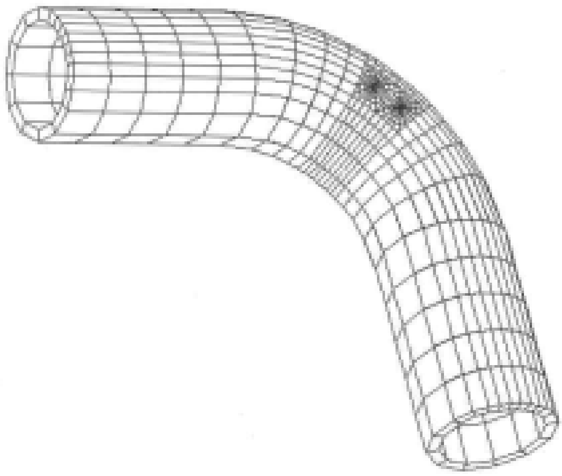


Fig. 2 Finite Element Model of the Elbow with axial TWC at Crown

7. RESULTS AND DISCUSSIONS

A non-linear finite element analysis was performed using the WARP3D (Gullerud et al., 2001) code. The material was assumed to follow the Ramberg-Osgood relation. The constants used for the above relation are given in paragraph 5 above. Monotonic incremental in-plane closing load was applied as mentioned above. The load line displacement at the end of the extended straight pipe and the applied moment are used to generate the load-load line displacement curve. From this curve, the limit load is evaluated. The limit load mentioned herein is obtained by using the double slope of linear response method (twice elastic slope method). This is evaluated by drawing a line at a slope two times that of the elastic portion of the load-load line displacement curve, as shown in *Fig. 3.* The load with load-line displacement is plotted in a graph and the limit load was evaluated. *Fig. 4* shows the variation of the load-line displacement with applied load for this elbow. From this graph, the limit load evaluated from the finite element results as per the twice-elastic slope method is 1452.03 KN. The limit load predicted experimentally gives 1463 KN. The result shows excellent agreement for the limit load with an error of 0.75%. *Fig. 4* also shows the comparison of the load-line displacement with applied load for both experiment and the finite element prediction. It can be seen form the graph the load-line displacement of the finite element analysis agrees very well with the experimental results.

The crack mouth opening displacement (CMOD) was obtained for the applied loads. When the elbow is subjected to in-plane closing bending moment, it was noticed that the outer surface of the crown was opened and the inner crack surface was closed. The crack opening displacement can be characterised by two components. The first component is the net opening displacement of the centre line of the crack surface. The second component is generated due to the rotation of the crack faces with respect to centre line. The net displacement of any point on the crack surface is the superimposition of these two components. In the present case it has been observed that the rotation is such that the net inner surface displacement is very low and they are in contact with each other at high loads. The crack opening behaviour from the deformed shape at crack surface indicates that the response of elbow was a combination of mode I and mode III. The CMOD at the outer surface is positive and at the inner surface it is very less or negative depending upon the load. The crack mouth opening displacement mentioned herein is the maximum displacement at the outer surface of the crown. *Fig. 5* shows the variation of the crack mouth opening displacement with the applied load. This figure also shows the comparison of the results of experiment and the finite element predictions. It can be seen that here also the results of the finite element analysis agree well with the experimental values.

As the inside crack surface faces at crown get closed and touch each other, the crack opening displacement at the inner surface is negative. The contact of the two crack faces is a non-linear boundary condition and is not considered in the present finite element model. The present study allows inter penetration of the two closing faces at the inside surface of the elbow. From the analysis performed, *J-integral* for the elbow was also evaluated. Even though there are no experimental results available for comparing *J-integral*, the finite element evaluated results are shown in *Fig. 6* for completeness.

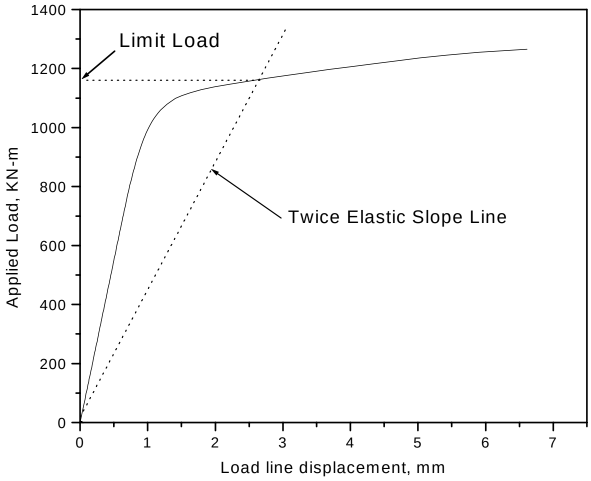


Fig. 3 Twice Elastic Slope Method for Evaluation of Limit Load

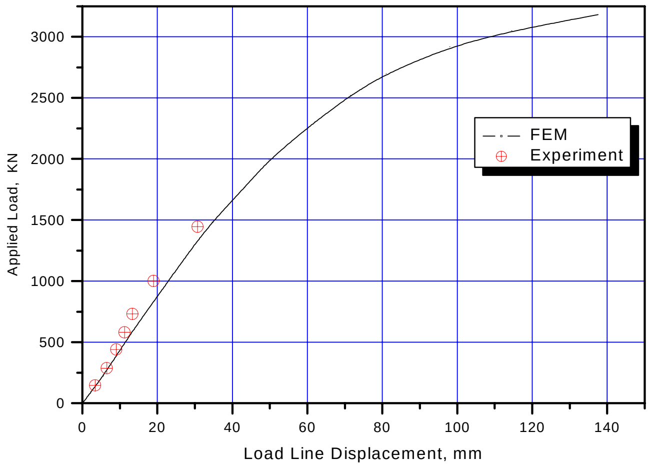


Fig. 4 Variation of Load line displacement with the applied load

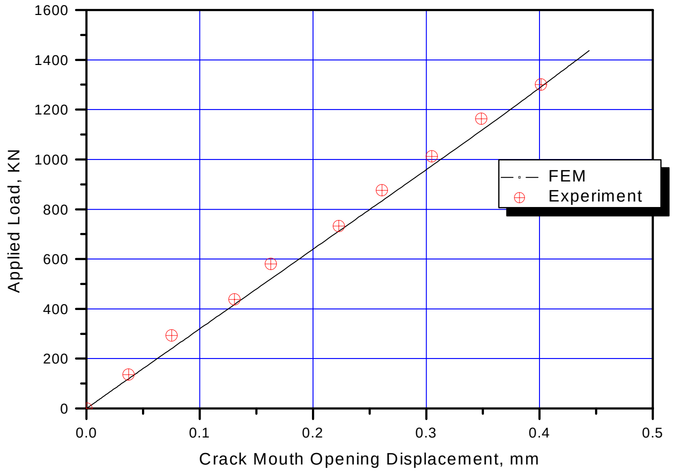


Fig. 5 Variation of CMOD with the applied load

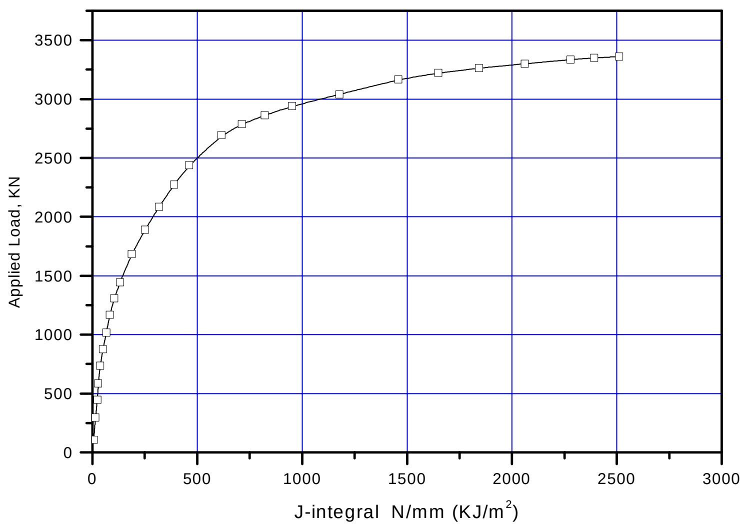


Fig. 6 Variation of J-integral with the applied load

8. CONCLUSIONS

This paper presents the results of the finite element results performed on a 16-inch carbon steel elbow. The model was generated using the actual measured dimensions of the elbow which was tested earlier. Both geometric and material non-linearity were considered in the study. The non-linear finite element analysis was performed by assuming the material to follow Ramberg-Osgood relation. The predicted finite element results were compared with the experimental values. The limit load prediction showed an excellent agreement with the experiment, the error being only 0.75%. The load line displacement and crack mouth opening displacements were evaluated and the predictions obtained also compare very well with their respective experimental values. Apart from these, analyses were also performed for J-integral evaluation and also the behaviour of the elbow under in-plane bending. The finite element predictions agree very well with the experimental results. The present study is a useful addition in the area of fracture behaviour of a cracked elbow with applied in-plane bending moment.

The following conclusions can be drawn from the analyses performed for the elbows with axial through-wall crack at crown, subjected to in-plane closing bending moment.

a) The results shown above prove that for the elbow under monotonic in-plane bending, the finite element procedure is capable of providing quite accurate prediction of the pipe bend behaviour. This three-dimensional FE analysis is an alternative for performing the very expensive tests.

b) When the elbow is subjected to in-plane closing bending moment, it was noticed that the outer surface of the crown was opened and the inner crack surface was closed. It has been observed that the inner surface displacement is very low and the crack faces are in contact with each other at high loads. The deformed shape at the crack surface indicates that the crack opening behaviour of the elbow is a combination of mode I and mode III.

REFERENCES

Bakker, A., (1983), An analysis of the numerical path dependence of the J-integral, *International Journal of Pressure Vessels and piping* ;Vol. 14; 153-179.

Chattopadhyay, J., Nathani, D.K., Dutta, B.K., Kushwaha, H.S., (2000), Closed form Collapse moment equations of elbows under combined internal pressure and in-plane bending moment, *ASME Journal of Pressure Vessel Technology*, Vol-122, 431-436

Diem, H., Blind, D., Katzenmeier, G., Hunger, H.A., (1989), Crack initiation and crack propagation of an elbow under in-plane bending in high temperature water of elevated oxygen content, *Transactions of 10th international conference of the Structural Mechanics in Reactor Technology,* Paper G03/1,65-76.

Griffiths, J.E., (1979), The effect of cracks on the limit load of pipe bends under in-plane bending: Experimental study, *Int. Journal of Mechanical Sciences,* 21, 119-130.

Gullerud, A.S., Koppenhoefer, K.C., Roy, A., Dodds, R.H., (2001), WARP3D –Release 13.12 3D dynamic non-linear fracture analysis of solids using parallel computers and workstations; Dept. of Civil Engineering University of Illinois at Urbana-Champaign, USA.

Hutchinson, J.W., (1968), Singular behaviour at the end of tensile crack in a hardening material, *Journal of Mechanics and Physics of Solids*, Vol-16; No. 1: pp 13-31.

Kano, T., Iwata, K., Asakura, J., Takeda, H., (1997), Stress Distributions of an elbow with straight pipes, *Transactions of 4th international conference of the Structural Mechanics in Reactor Technology*; Vol. F F-1/5.

Muller, K.U., Diem, H., (1985), Deformation and failure behaviour of elbows, *Transactions of the 8th international conference of the Structural Mechanics in Reactor Technology*, Vol-F1-7/9.

Naoki Miura., Yasunari Nakayana, (1997), Ductile fracture experiments for through wall cracked elbows at high temperature subjected to in-place bending, *ASME PVP*, 350, 97-105.

Prabhakaran, K.M., Venkat Raj,. V., (2002), ELBOWGEN A Graphic Software for Generating Finite Element model of an Elbow with Axial through wall Crack at Crown, Unpublished work.

Seetharaman, S., Ramachandramurthy, D.S., Gandhi, P., (1999), Fracture Studies on carbon Steel Elbow, Structural Engineering Research Centre, Chennai, *Bhabha Atomic Research Centre Sponsored Consultancy Project* CNP 033241, November, 1999.

Shalaby, M.A., Younan, M.Y.A., (1997), Limit Loads for Pipe Elbows with Internal Pressure under in-plane Closing Bending Moments, ASME PVP, Vol. 347, 203-213.

Shalaby, M.A., Youman, M.Y.A., (1998), Nonlinear analysis and plastic deformation of pipe elbows subjected to in-plane bending, *Int. Journal of pressure vessels and piping,* 75, 603-611.

Singh, P.K., Chattopadhyay, J., Kushwaha, H.S., Tarafdar, S., Ranganath, V.R., (1998), Tensile and Fracture properties evaluation of PHT system piping material of PHWR, *International Journal of Pressure Vessels and Piping*, Vol-75, 271-280

Zahoor Akram, (1991), Ductile Fracture Hand Book; EPRI/NP-6301.

