PREDICTION OF FATIGUE CRACK PROPAGATION IN CIRCUMFERENTIALLY CRACKED PIPE SPECIMEN USING CASCA AND FRANC2D

A THESIS SUBMITTED IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE DEGREE OF

BACHELOR OF TECHNOLOGY

IN

MECHANICAL ENGINEERING

By:

ASHUTOSH SHARAN

TOM ATUL DUNG DUNG

DEPARTMENT OF MECHANICAL ENGINEERING

NATIONAL INSTITUTE OF TECHNOLOGY

ROURKELA 769008

2012

PREDICTION OF FATIGUE CRACK PROPAGATION IN CIRCUMFERENTIALLY CRACKED PIPE SPECIMEN USING CASCA AND FRANC2D

A THESIS SUBMITTED IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE DEGREE OF

BACHELOR OF TECHNOLOGY

IN

MECHANICAL ENGINEERING

By:

ASHUTOSH SHARAN

TOM ATUL DUNG DUNG

Under the guidance of

PROF. P. K. RAY

DEPARTMENT OF MECHANICAL ENGINEERING

NATIONAL INSTITUTE OF TECHNOLOGY

ROURKELA, 769008

2012

NATIONAL INSTITUTE OF TECHNOLOGY, ROURKELA

CERTIFICATE

This is to certify that the thesis entitled "**Prediction of Fatigue Crack Propagation in Circumferentially cracked pipe using CASCA and FRANC2D**" submitted by **Ashutosh Sharan (Roll No. 108ME075)** and **Tom Atul Dung Dung (Roll No. 108ME035)** in the partial fulfillment of the requirements for the award of Bachelor of Technology degree in Mechanical Engineering at National Institute of Technology, Rourkela (Deemed University) is an authentic work carried out by them under my supervision and guidance.

To the best of my knowledge, the matter embodied in the thesis has not been submitted to any other University/Institute for the award of any Degree or Diploma.

Date:

Prof. P. K. Ray

Department of Mechanical Engineering

National Institute of Technology, Rourkela

ACKNOWLEDGEMENT

We wish to express our deep sense of gratitude and indebtedness to Prof. P.K. Ray, Department of Mechanical Engineering, N.I.T Rourkela, for introducing the present topic and for his inspiring guidance, constructive criticism and valuable suggestion throughout this project work.

We also extend our sincere thanks to all our friends who have patiently helped us in accomplishing this undertaking. We also thank Mr. Hemendra Patel and Mr. Pawan Kumar for their constant support throughout the project work.

We would also like to thank Cornell Fracture Group, Cornell University, New York for making the software CASCA and FRANC2D available free of cost.

Date: Ashutosh Sharan (108ME075)

Place:

Tom Atul Dung Dung(108ME035)

CONTENTS

LIST OF FIGURES

LIST OF TABLES

ABSTRACT

Pipes are subjected to different types of loading like internal pressure, tension, compression, bending or any combination of these. These different types of loading situation may initiate and propagate a crack. This becomes more significant if the pipes carry hazardous fluid. In this project TP316L stainless steel pipe is considered for study of propagation of an existing crack. Two methods are proposed to convert three dimensional study of pipe into a two dimensional study of a beam. A finite element based two dimensional crack propagation simulator software FRANC2D and a pre-processor software for this simulator CASCA developed by Cornell Fracture Group of Cornell University was used for simulation of crack propagation in two dimensional beam obtained by two methods and results found in both the cases were compared. These simulations were performed for four point bending test.

CHAPTER 1

INTRODUCTION

Pipes are often used in plants and industries like coolants transmission in nuclear power plants, gas transmissions etc. Prediction of fatigue crack growth for an existing surface flaw is important for developing flaw acceptance criteria. Many researchers have studied crack growth behavior and fatigue fracture of pipes containing surface flaws under different loading conditions [1, 2]. There are many practical methods to study fatigue crack growth in structural components like plate, bar, pipe etc. but these methods are destructive in nature and require a lot of money, skill and time. To solve these problems many analytical approach based on fracture mechanics and finite element method have been proposed. Three dimensional analysis of partial circumferential crack in pipe requires immense computational effort to meet the adequate mesh pattern and enormous computer storage. In the present project work two methods have been proposed to predict fatigue crack growth behaviour of a straight circular pipe having a circumferential crack on its outer surface on a two dimensional crack growth simulator. In the first method three dimensional pipe problem was converted into a two dimensional beam problem by equating the deflection under four point bending of pipe to that of beam. In second method three dimensional pipe problem was converted into a two dimensional beam problem by equating stress intensity factors in both beam and pipe for four point bending test. Then crack propagation simulation was performed on the dimensions of beam obtained from both the methods using a mesh generator program CASCA and a FEM based crack propagation simulator program FRANC2D. Results obtained from simulation of both the cases were then compared.

CHAPTER 2

LITERATURE REVIEW

2.1 Fatigue

Fatigue failures occur when metal is subjected to a repetitive or fluctuating stress (load) and will fail at a stress much lower than its tensile strength. It occurs without any plastic deformation. Fatigue surface appears as a smooth region, showing beach mark or region of fatigue crack. Fatigue process involves:-

1. Crack initiation – development of fatigue damage and it can be removed by thermal anneal.

2. Slip band crack growth - deepening on plane of high shear stress also known as stage 1 crack growth.

3. Crack growth on planes of high tensile stress - growth of well-defined crack in a direction normal to maximum tensile stress.

4. Ultimate ductile failure - occurs when crack reaches sufficient length so that the remaining cross section cannot support the load.

2.2 Fatigue Crack Propagation

When a component containing a crack is loaded statically, no crack growth occurs as long as the crack length or the loading remains below a critical value. If the loading is oscillating crack growth in small steps can be observed already for loading amplitudes far below the critical static load. Such a crack growth is called *fatigue crack growth.* Usually fatigue crack growth is characterized by the *crack growth rate* (da/dN), where N is the number of load cycles [3].

Fatigue crack propagation behavior for metals can be divided into three regions (fig.1). The behavior in region 1 exhibits a fatigue-threshold cyclic stress intensity factor range, ΔK_{th} below which cracks do not propagate under cyclic stress fluctuations.

Region II represents the fatigue crack propagation behavior above ΔK_{th} which can be represented by, [4]

$$\frac{da}{dN} = C(\Delta \mathrm{K})^{m}$$

Where, a = crack length; N = no. of cycles; ΔK = stress intensity factor range,

C and m are material constants.

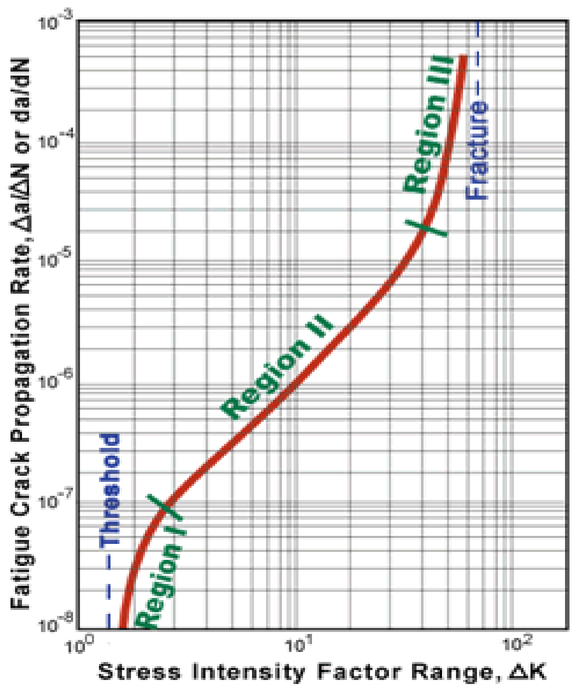

Fig.1. Fatigue crack growth rate vs stress intensity factor range [5]

In region III the fatigue crack growth per cycle is higher than that predicted for region II. Following characteristics are shown by the materials regionwise.

Region I: This stage is non-propagating. In this stage of crack growth there is large influence of microstructure, mean stress and environment on the fatigue crack growth.

Region II: This is widely studied stage among all the stages of fatigue crack propagation. This is also stable stage fatigue crack propagation process. Continuous behavior, striations or transition from non-continuous behaviour with

(a) Small to large influence of microstructure, deepening on material.

(b) Large influence of certain combinations of environment, mean stress and frequency.

Region III: in this stage unstable fatigue crack growth occurs and then failure occurs. Object shows static mode of behaviour. In this stage there is a large influence of microstructure, mean stress and thickness but little influence of environment cleavage, inter-granular and dimples affects this stage of crack growth.

2.3 Stress analysis for members with cracks- fracture mechanics approach.

Fracture mechanics is an engineering method for analyzing fracture and fatigue behavior of sharply notched structural members (cracked or flawed) in terms of stress and crack length. Stress concentration factor and stress intensity factor are two quantities which are used to analyze a stress in vicinity and ahead of a well-defined notch or a sharp crack respectively.

Stress Concentration Factor is used to analyze stress at a point in vicinity of well-defined notches. most structural components have discontinuities of some type , for example holes, fillets, notches etc. if these discontinuities have well-defined geometry, it is usually possible to determine the stress concentration factor ,K_t for these geometries [6]. Stress concentration factor gives a relation between local maximum stress and applied nominal stress.

$$\sigma_{max} = K_t . \sigma_{nom}$$

However, if the stress concentration is severe, for example approaching a sharp crack in which the radius of the crack tip approaches zero, an analytical method different from the stress concentration approach is needed to analyze the behavior of structural components containing sharp imperfections.

Stress Intensity Factor (K) is a parameter which is used to analyze stress field ahead of a sharp crack. It is related to both nominal stress level (σ) in the member and the size of crack (a) and has units of ksi√in, (MPa√m). To establish methods of stress analysis for cracks elastic solids, it is convenient to define three types of relative movements of two crack surfaces. These displacement modes (fig. 2) represent the local deformation ahead of a crack. The opening mode, Mode I is characterized by local displacements that are symmetric with respect to x-y and x-z planes. The two fracture surfaces are displaced perpendicular to each other in opposite directions. Local displacements in the sliding or shear mode, Mode II, are symmetric with respect to x-y plane, and skew-symmetric with respect to the x-z plane. The two fracture surfaces slide over each other in a direction perpendicular to the line of the crack tip. Mode III, the tearing mode is associated with the local displacements that are skew-symmetric with respect to both x-y and x-z planes. The two fracture surfaces slide over each other in a direction parallel to the line of crack front. Each of these modes of deformation corresponds to a basic type of stress field in the vicinity of crack tips [7].

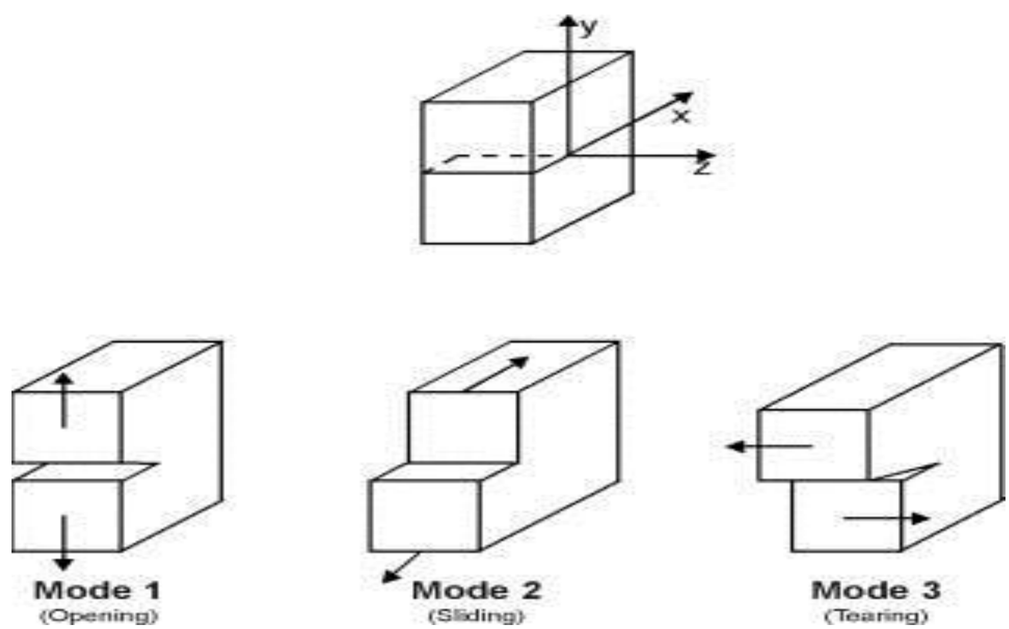

Fig.2 Different modes of crack surface displacement [8]

2.4 Assessment of Fatigue Crack Growth behavior of partly circumferentially cracked pipe

Generally finite element analysis is used to determine stress intensity factor and fatigue crack growth rate. Three dimensional modeling of partly circumferentially cracked pipe is much difficult and time taking process. Thus for sake of simplicity various models have been proposed to convert this three dimensional problem into a two dimensional problem like line spring model (by Rice and Levi, 1972), Conformal transform method (C. D. Wallbrink et al. 2003).

Above methods can be used to analyze complex circumferential crack problems and also to solve partly circumferential cracks in pipes. Problems related to non-linear stress distributions, double curvature, internal cracks can be accurately solved by these methods.

CHAPTER 3

Materials and methods

3.1 Piping material

Specimen used for simulation was A312 TP316L stainless steel seamless pipe.It is used in Indian PHWRs. Alloy 316L is molybdenum-bearing austenitic stainless steel .This grade of steel are more resistant to corrosion and pitting/crevice corrosion than conventional chromium-nickel austenitic stainless steel such as 304. This alloy also offers higher creep resistance, stress-to-rupture, and tensile strength at elevated temperature. With the excellent corrosion resistance and strength properties, the alloy 316L Cr-Ni-Mo alloy also provide good fabrication ability and formability which are typical of austenitic stainless steel.

316L grade gives the much increased corrosion resistance in aggressive environment. Molybdenum makes the steel more resistant to pitting and crevice corrosion in chloride-contaminated media, sea water and acetic acid vapor. This grade of steel possesses excellent mechanical properties and corrosion properties at sub-zero temperatures. When there is a danger of corrosion in the heat affected zones of weldments the low carbon variety 316L should be used. Grade 316L finds wide applications as pipe and heat exchanger tubes in chemical and petrochemical plant, in boilers, food industry and power plants. The chemical composition and mechanical properties of the specimen material are listed in following tables respectively.

Table 1: Chemical Composition of TP316L Stainless Steel

Elements	Percentage by weight
Iron(main constituent)	65.8
Carbon	0.03
Manganese	1.5
Silicon	0.5
Chromium	17.0
Nickel	12.5
Molybdenum	2.5
Phosphorus	0.03
Sulphur	0.025
Nitrogen	0.10

Table 2: Mechanical Properties of TP316L Stainless Steel

Young's modulus (E)	220GPa
Poisson's ratio(μ)	0.3
Yield strength	366MPa
Ultimate tensile strength	611MPa

3.2 Details of pipe specimen

The pipe of 60 mm outer diameter and 9 mm wall thickness used in the piping system of Indian PHWRs has been used for analysis. The pipe specimens had surface notches of different sizes machined at the outer surface in the circumferential direction by wire EDM machining process. Straight surface notches of different depths and notch angle 2θ (45^{o}) were made on the outer circumference by wire EDM maintain notch tip radius 0.8 mm. The detailed dimensions of the specimen and notch are given in Fig. 3 and Table 3 respectively.

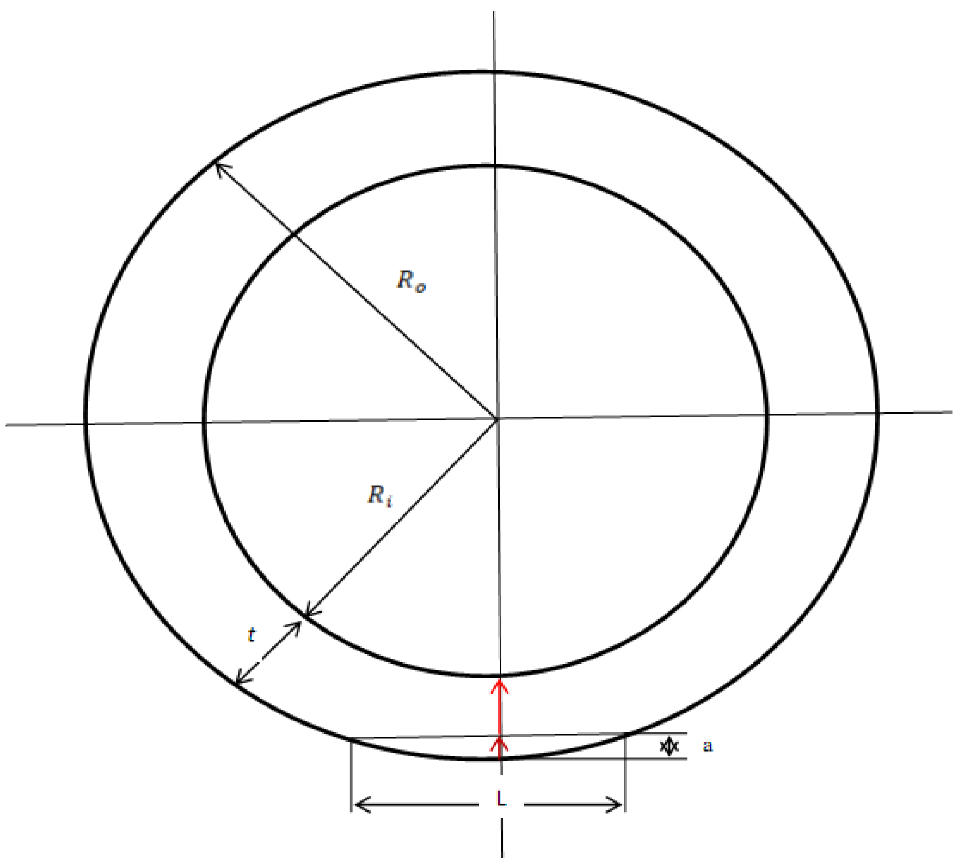

Fig. 3 Schematic diagram of specimen

Table 3: Notch dimension of pipe

Elements	Values (mm)
Outer diameter (d_0)	60
Inner diameter (d_i)	42
Thickness (t)	9
Crack depth (a)	2.28
Crack curve length (L)	23

3.3 Experimental setup

For simulation process on FRANC2D, four point bending test has been considered. This type of loading ensures that the mid-section of the specimen, where the notch is located is subjected to pure bending. The schematic diagram of the test set up is shown in the fig.4.

Pipe test arrangement constituted loading the pipe under four point bending up to large scale plastic deformation with periodic significant unloading so as to create a beach mark on the crack surface. The load is given in the form of sinusoidal wave.

The load range considered for the fatigue crack growth test was of the order of 45 KN which is below the yield strength of the piping material corresponding to given notch dimensions. This is to ensure that the crack growth is under gross elastic conditions.

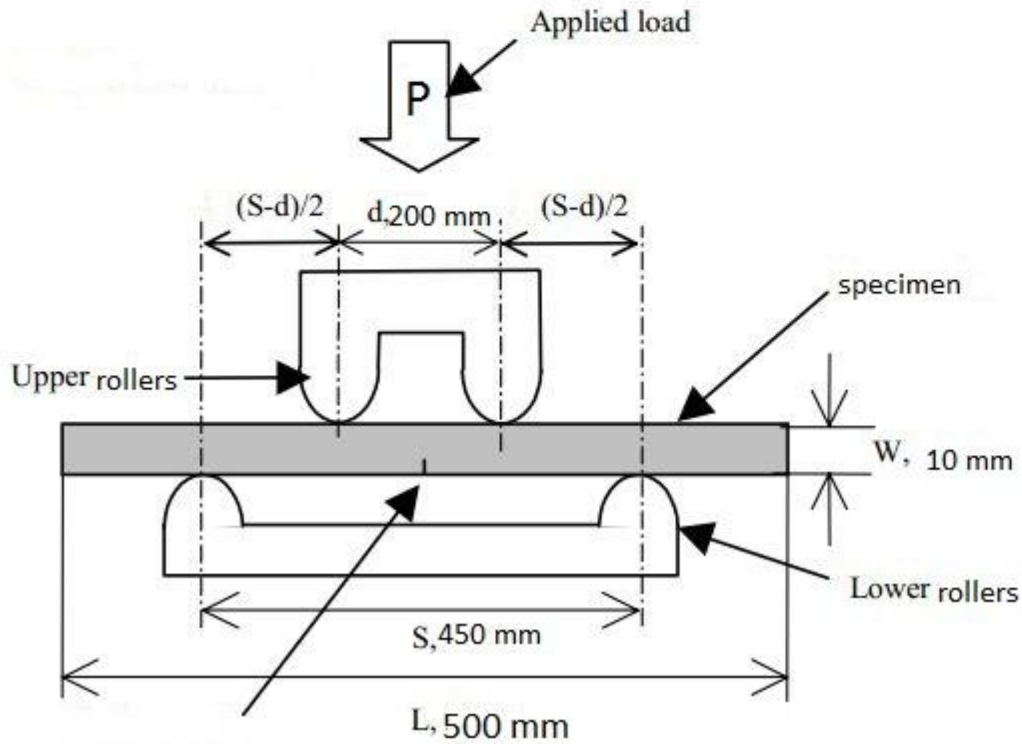

Fig. 4 Schematic diagram of four point bending test [9]

3.4 Conversion of pipe problem to a beam problem

Method 1

There is much computational complexity in three dimensional mesh generation and FEM analysis of crack propagation in partly circumferentially cracked pipe. Hence for simplify this three dimensional problem we have converted three dimensional pipe problem into a two dimensional beam problem based on some assumptions so that it can be simply simulated in a two dimensional FEM based crack propagation simulator.

Assumption:

1. For same material if the loading conditions and deflection are same, the crack propagation behaviour will be same for straight circular pipe and rectangular beam.
2. Length of beam and pipe is same.

Based on this assumption by equating the stiffness of rectangular beam and straight circular pipe, we can deduce a relation between their parameters. Hence, we can find the dimensions of the beam if dimensions of straight circular pipe are known.

$$(K)_{beam} = (K)_{pipe}$$

$$\text{Or, } \left(\frac{P}{\delta}\right)_{beam} = \left(\frac{P}{\delta}\right)_{pipe}$$

Where, K = Stiffness, P = applied load and δ = deflection

Since loading conditions are same and deflection (δ) is inversely proportional to flexural rigidity, i.e.

$$\delta \propto \frac{1}{EI}$$

Hence, $(EI)_{beam} = (EI)_{pipe}$

Where, E = Young's Modulus and I = Moment of Inertia.

Since materials for beam and pipe are also same, value of Young's modulus (E) will be same.

$$(I)_{beam} = (I)_{pipe}$$

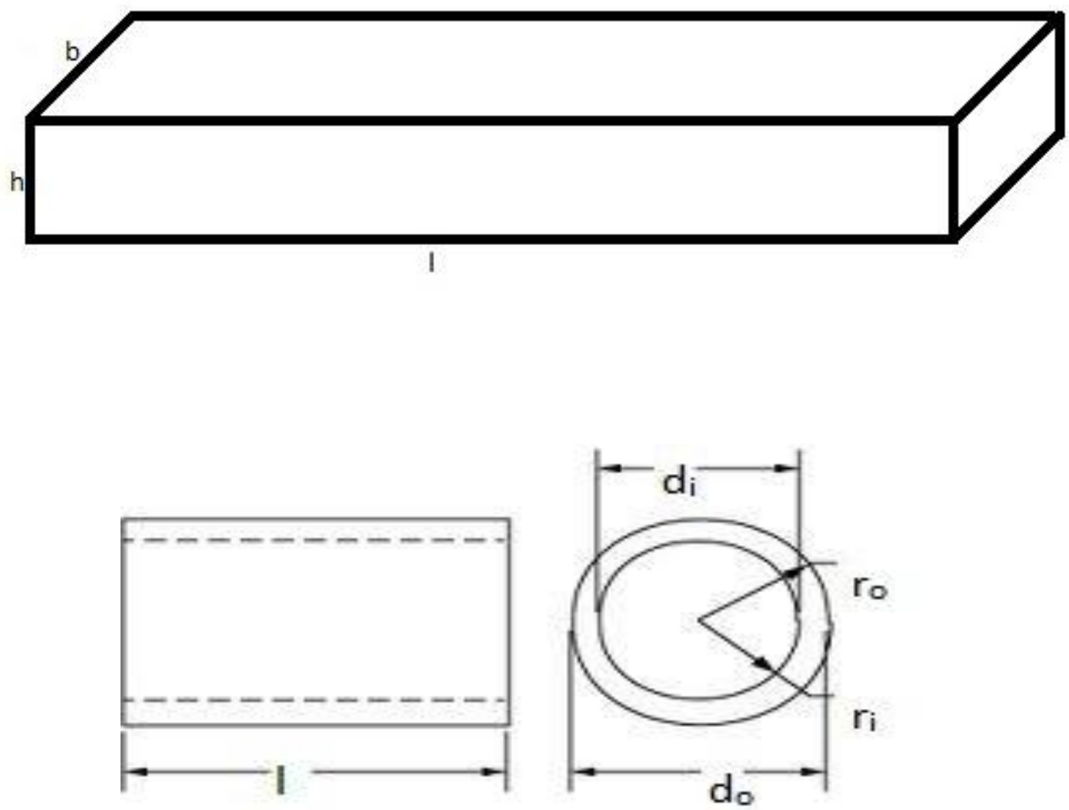

Fig. 5 Geometry of pipe and beam

Or, $$\frac{bh^3}{12} = \frac{\pi}{64}[d_o^4 - d_i^4] \qquad (1)$$

As the dimensions of our test specimen are as follows:

Inner diameter, d_i = 42 mm

Outer diameter, d_o= 60 mm

Length of the pipe = 505 mm.

Putting the values of d_i and d_o in equation 1 we get,

$$bh^3 = 5.8\ X\ 10^6$$

Now, as wall thickness of pipe specimen is 9 mm we can fix b=9 mm and find out corresponding value of h.

Thus, taking b= 9, h is found out to be $h = 86.38\ mm$

Method 2

Paris et al. [4] suggested an empirical relation between fatigue crack growth rate (da/dN) and stress intensity factor range (ΔK) for region II in Fig. 1. This relationship can be expressed as

$$\frac{da}{dN} = C\ (\Delta K)^m \qquad (1)$$

Since C and m are material constants in equation (1), fatigue crack growth rate depends on stress intensity factor range only as far as same material is concerned. Hence if a pipe and a beam of same material have same stress intensity factor range they will have same fatigue crack growth rate. To study the fatigue crack growth behaviour of a pipe we can study that in a beam of same stress intensity factor range (ΔK). Here we have taken a beam of width (w) equal to the wall thickness of the pipe specimen i.e. 9 mm as we have to study fatigue crack growth up to 9 mm. Then we equated the stress intensity factors for four point bend test on a beam and the pipe specimen. Formula used for stress intensity factor of a beam under four point bending is [10]

$$K = \frac{3P\lambda}{tw^2}\sqrt{\pi a}F(\alpha) \qquad (2)$$

where '*P*' is applied load, 'λ' is distance between adjacent upper and lower rod, i.e. (*s*–*d*)/2, 't' is thickness of the beam '*w*' is width of the beam and '*a*' is depth of the crack,[Fig. 4]

$\alpha = \left(\frac{a}{w}\right)$ and

$$F(\alpha) = 1.122 - 1.121\alpha + 3.740\alpha^2 + 3.873\alpha^3 - 19.05\alpha^4 + 22.55\alpha^5$$

By equating the value of 'K' from equation (2) for beam with the value of '*K*' calculated for pipe for a particular value of '*a*' we got a relation between '*P*' and '*t*'. Then we assumed thickness (t) of the beam equal to 1.5 times the width (*w*) and corresponding value of *'P'* was found. Calculations are shown in appendix-1. For these calculated values of beam simulation was performed on FRANC2D and necessary data and plots were recorded and analyzed.

3.5 Simulation on FRANC2D

FRANC2D is a two dimensional, finite element based program for simulating curvilinear crack propagation in planar structures. CASCA is a simple pre-processor for FRANC2D. Other 2D finite element programs can also be used as a pre-processor for FRANC2D provided the data can be converted to FRANC2D input. FRANC2D and CASCA software were developed by Cornell Fracture Group, Cornell University, Ithaca, New York.

The simulation procedures for both the methods stated above are as follows:

1. Geometrical layout of the beam of required dimension was created using CASCA pre-processor and a mesh pattern was generated for the same.

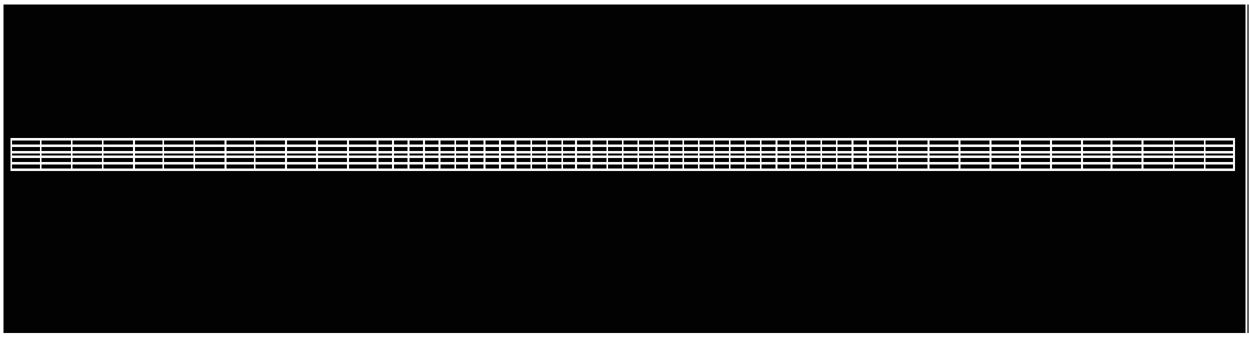

Fig. 6 Mesh generated in CASCA

2. The mesh generated in CASCA was then saved as a file that can be reopened in FRANC2D for simulation process.
3. After saving the file and exiting from CASCA, FRANC2D was opened. The mesh file saved in CASCA was opened in FRANC2D.
4. Then problem type was defined and appropriate material properties were set for the pipe material considered by following command sequence.

   PRE-PROCESS→ PROBLEM TYPE→PLANE STRESS.

   for material option command MATERIAL was selected. Corresponding values of the Young's modulus, Poisson's ratio, thickness was given by selecting E, NU and THICKNESS option respectively. Our material was TP316L stainless steel. Its material properties were entered using table 2[page 9].
5. After returning from the materials page, it is required to reformulate the element stiffness matrices. This was done by selecting ELEM STIFF option. Then file was saved.
6. The next step was to specify the boundary conditions. This is done by selecting PRE-

PROCESS and then FIXITY. Two nodes were fixed appropriately in X or Y direction. The size of the box containing the node can be adjusted using the tolerance window given below.

7. After the boundary conditions were provided, loads were given by selecting LOADS→POINT LOAD. Then the corresponding values of load were entered and point of loading was specified for both the simulations.
8. Then the stress analysis was done by selecting ANALYSIS→LINEAR→DIRECT STIFF. This provided a little report that summarized the size of the model and the time required for the analysis.
9. After the analysis, DEFORMD MESH option was used to see if the boundary conditions are properly given. Then POST-PROCESS option was selected, followed by CONTOUR option which provided us with various color stress contours to indicate principle tensile stress(SIG 1) , effective stress(EFF STRESS), shear stress(TAU MAX) etc.

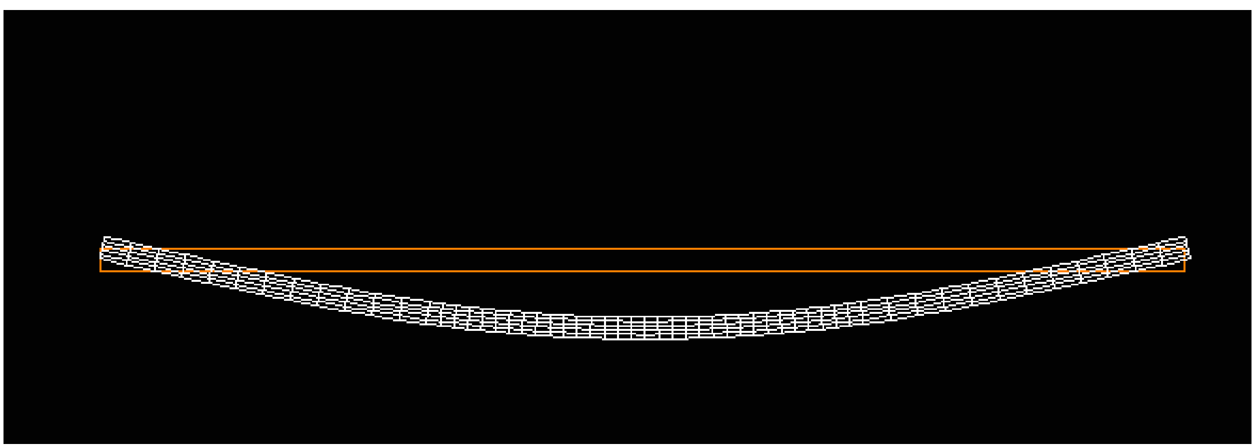

Fig. 7 Deformed mesh after applying boundary conditions and load.

10. Now we put the crack in the beam. Crack was initiated by selecting MODIFY→NEW CRACK→NON-COHESIVE→EDGE CRACK. The location of the crack was specified

in the middle of the beam. The crack length was then entered and the minimum no. of elements along crack extension was taken as 2. Then ACCEPT option is selected. Re-meshing of nodes takes place.

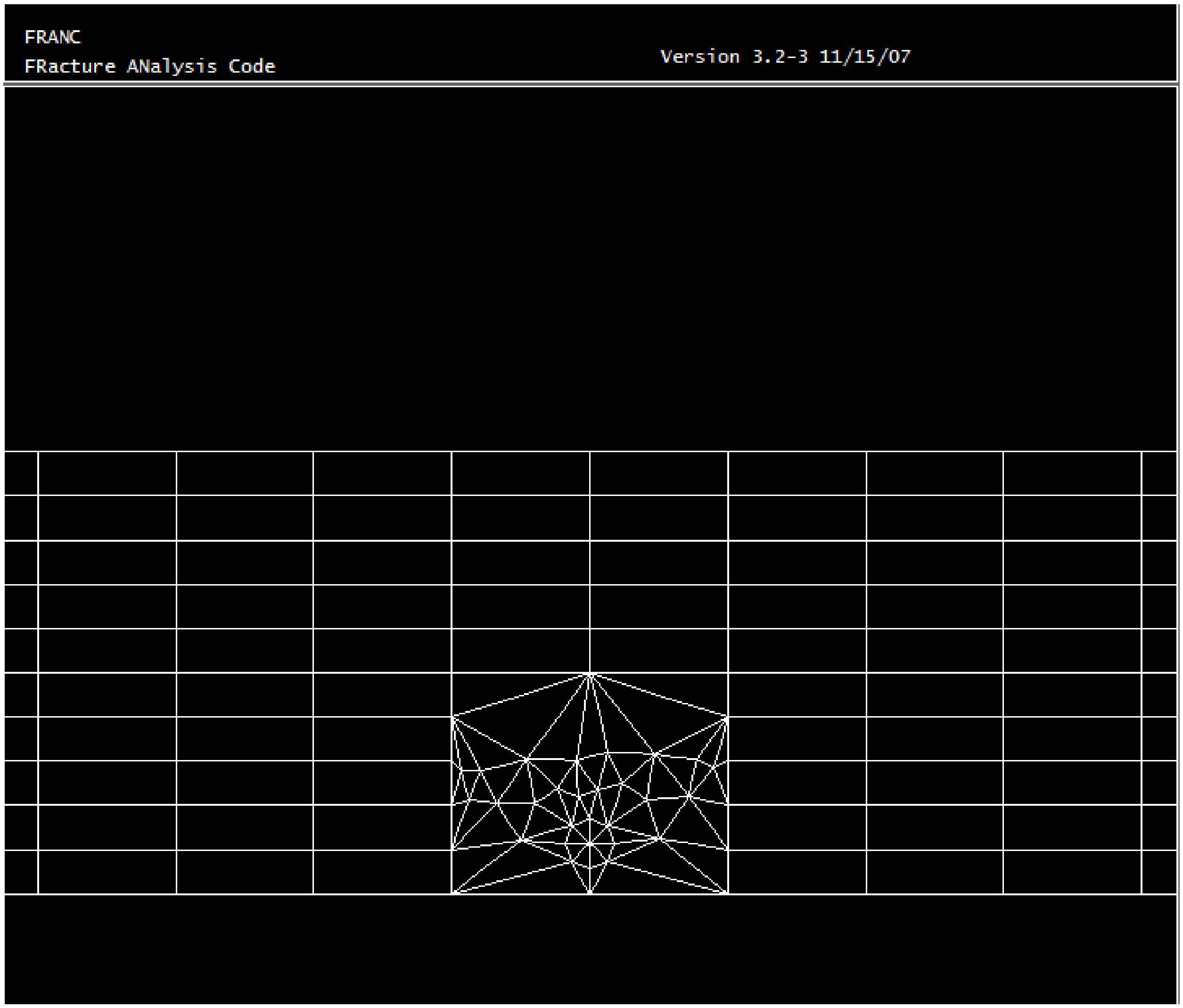

Fig. 8 Crack Initiation

11. For this new structure, new analysis was performed by selecting ANALYSIS→LINEAR→DIRECT STIFF. A report on the model was generated.
12. Then FRACT MECH option was selected and the stress intensity factors were computed using displacement correlation technique (DSP CORR SIF), J integral technique (J-

INTEGRAL), and modified crack closure integral technique (MD CRK-CLOS). The three techniques gave the similar values.

13. Now the crack was propagated from the crack tip. This was done by entering MODIFY→MOVE CRACK→AUTOMATIC. To give the amount of crack growth at each step CRACK INCR option was chosen and crack increment value per step was specified. STEPS option was then used to set the no. of propagation steps. Then PROPAGATE option was selected to begin crack propagation.

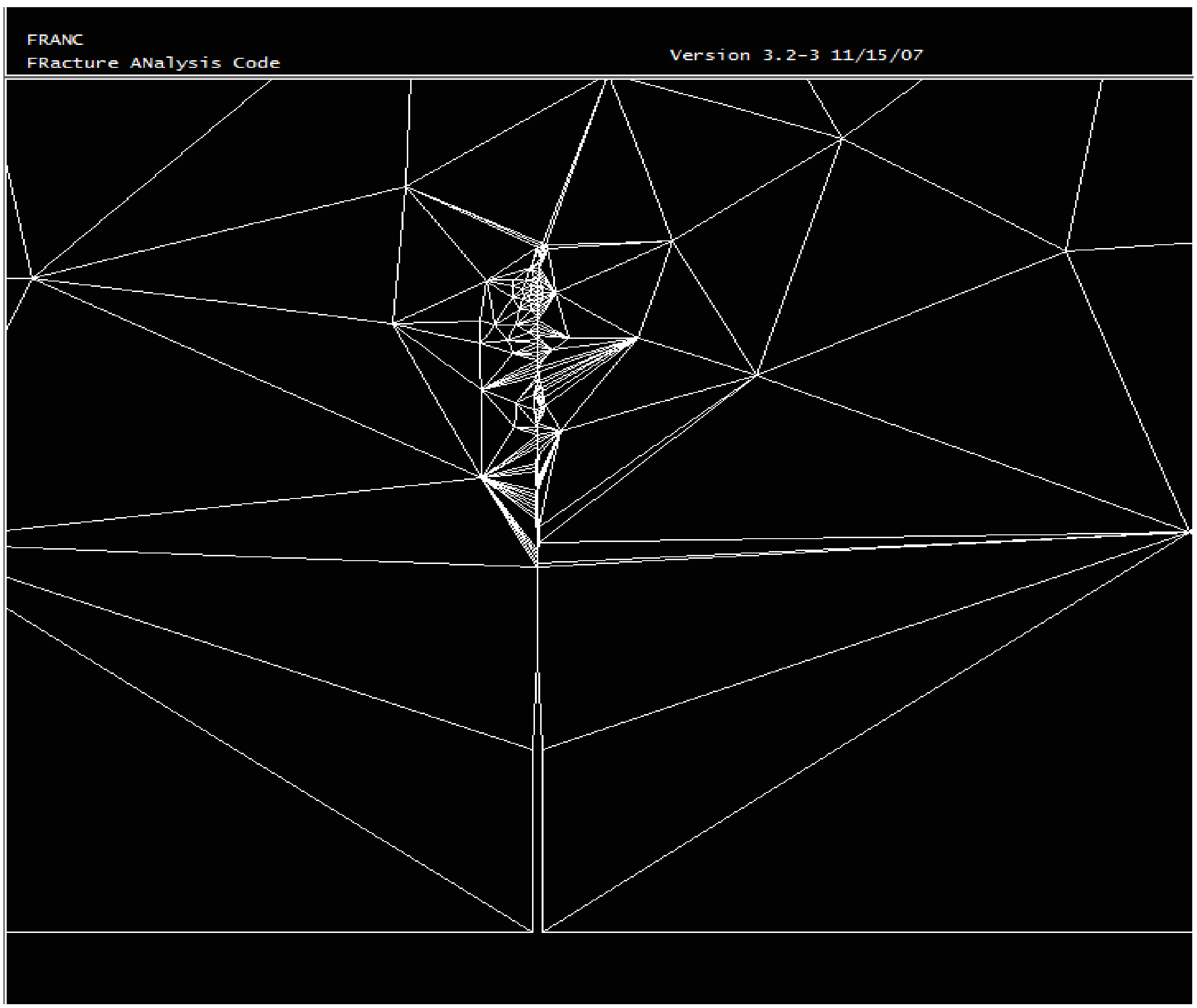

Fig. 9 Crack Propagation

14. Then the file was saved using WRITE option.
15. Now the fatigue crack growth analysis was done by selecting POST –PROCESS and FRACT MECH options. The stress intensity factor history was found using SIF HISTORY option. A KI vs. crack length graph was generated.
16. Now, by using the FATIGUE PLTS option, fatigue analysis was done. Since it is based on Paris model, constants C and m are provided using SET C and SET m option. Then the CYCLE PLOT option created a plot of the number of load cycles as a function of crack length.

CHAPTER 4

RESULTS AND DISCUSSIONS

4.1 Stress Intensity Factor from FRANC2D

The variation of Stress intensity factor with respect to crack length was plotted by FRANC2D for the beam dimensions found by method 1 is shown in following figure.

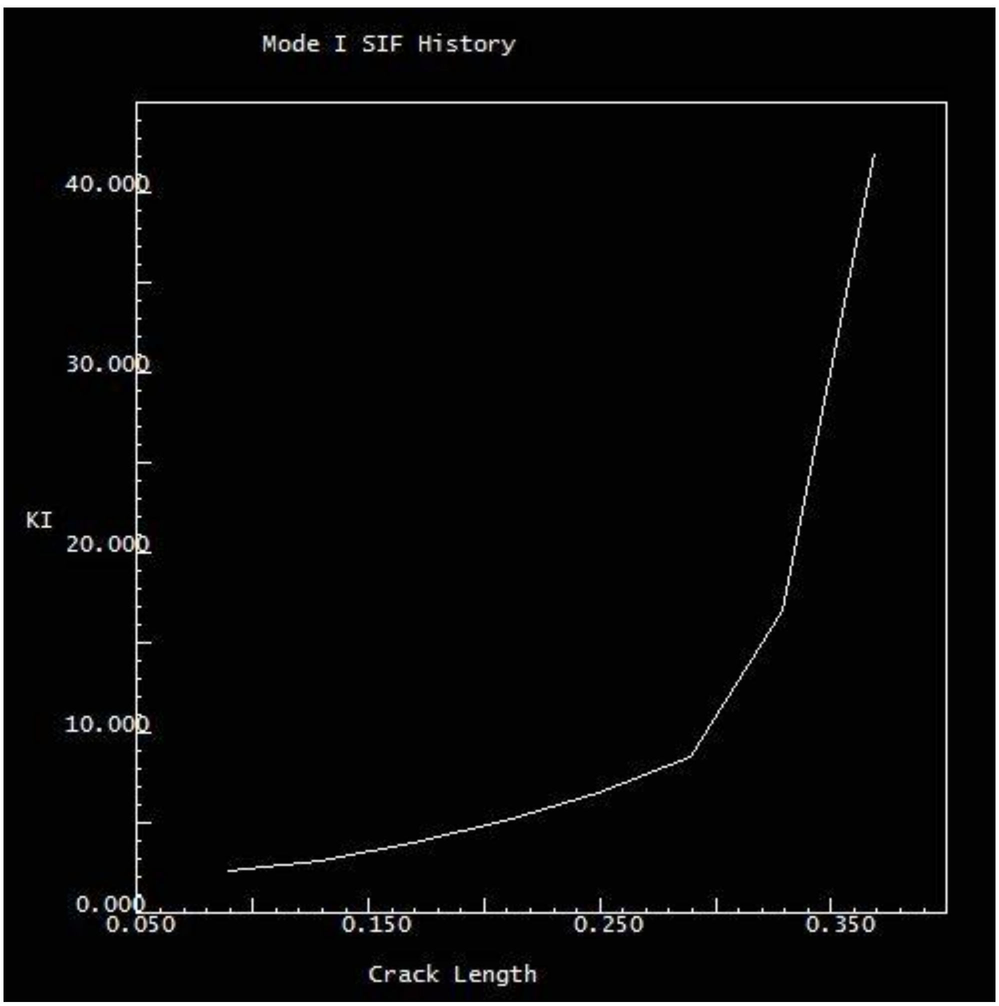

Fig. 10 Stress intensity factor history for beam converted by method 1

The variation of Stress intensity factor with respect to crack length was plotted by FRANC2D for the beam dimensions found by method 2 is shown in following figure.

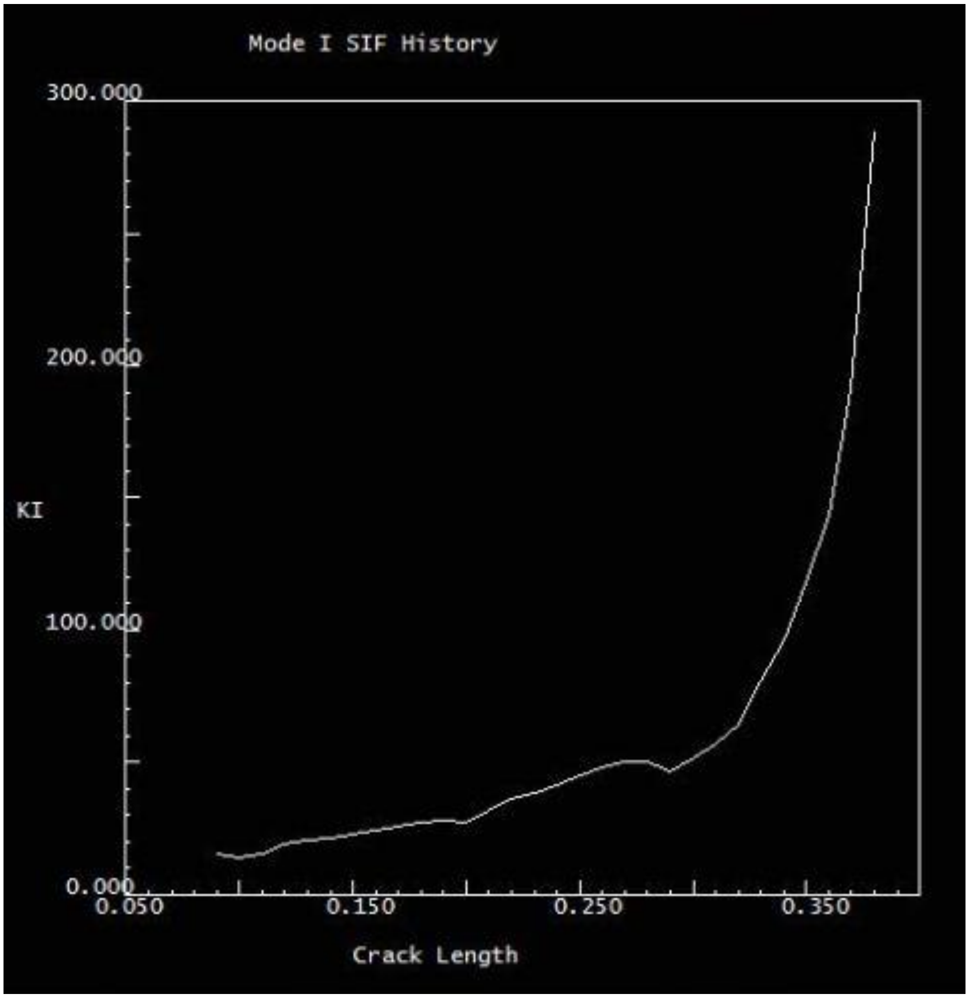

Fig. 11 Stress intensity factor history for beam converted by method 2

Units of crack length and KI in above plots are inch and Ksi√in. as all the units taken in simulation process were inch, Kips and Ksi.

4.2 Cycle Plot from FRANC2D

Using FRANC2D software the number of cycles was plotted as a function of crack length. Plot shows the variation in the number of cycles during the course of crack propagation and it gives the number of cycles required for fatigue fracture of the model. Fig. 13 shows the graph plotted in FRANC2D for beam converted by method 1 and Fig. 14 shows that for beam converted by method 2.

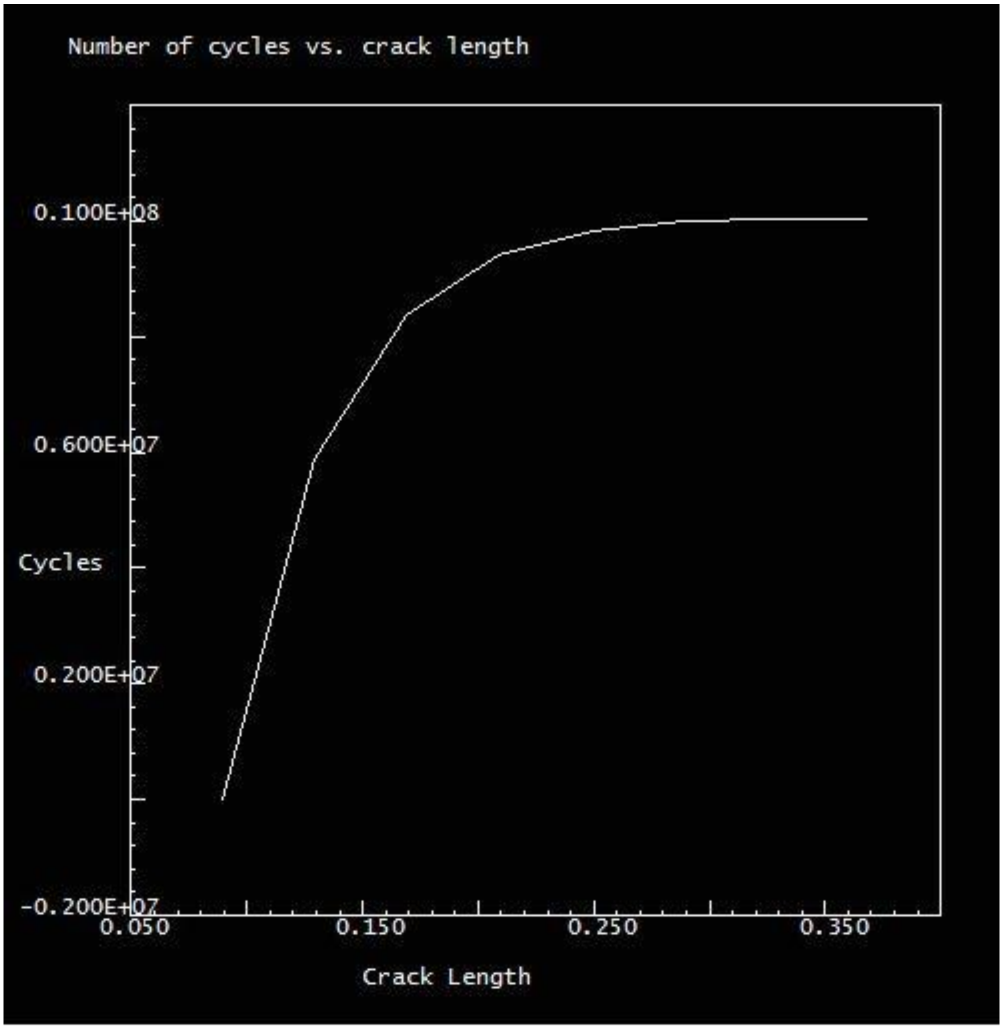

Fig. 12 Cycle Plot for beam converted by method 1.

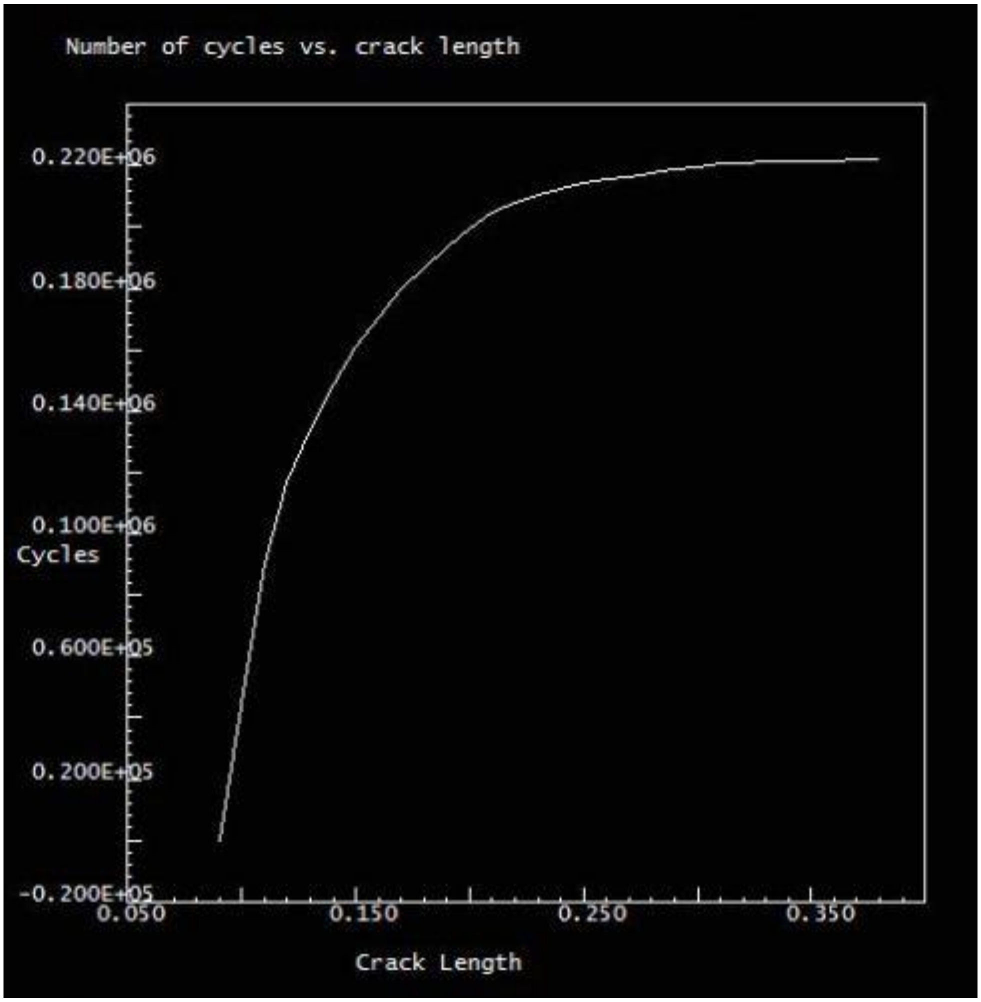

Fig. 13 Cycle plot for beam converted by method 2

Crack length in above graphs is in inch.

Table 4. Comparison of simulation results for method 1 and method 2

Sl. No.	Crack Length(mm)	SIF for method 1 conversion (MPa√m)	SIF for method 2 conversion (MPa√m)	No. of cycles for method 1 conversion ($x10^5$)	No. of cycles for method 2 conversion ($x10^5$)
1	2.28	2.63	15.38	0.10	10
2	2.54	3.30	13.18	0.13	12
3	3.81	3.85	21.98	1.40	70
4	5.08	5.95	26.37	2.0	74
5	6.35	7.14	43.95	2.12	90
6	7.62	10.99	50.94	2.16	91

CHAPTER 5

CONCLUSION

From graphs plotted in FRANC2D for both the methods it was found that nature of curves for SIF vs. crack length and number of cycles vs. crack length are satisfactory. Due to different assumptions taken in the two methods quantitatively, the graphs are different for the two methods. However the data from method 2 were found to be more realistic than those from method 1, because of the more practical SIF values. Still, these data need to be verified with experimental data.

REFERENCES

1. Anthony Andrews and Peter Folger: Nuclear Power Plant Design and Seismic Safety Considerations, Congressional Research Service, 2012
2. Sharif Rahman: "Probabilistic elastic-plastic fracture analysis of circumferentially cracked pipes with finite-length surface flaws" Nuclear Engineering and Design, 195 (2000) 239-260.
3. Dietmar Gross and Thomas Seeling: Fracture Mechanics with an Introduction to Micromechanics, Springer Verlag Publication,2006.
4. P. C. Paris and F. Erdogan, Trans. ASME J. Basic. Eng., Vol. 85, pp. 528-534, 1963.
5. Http://www.apesir.com/mechanical-engineering/fatigue-crack-growth-rate-properties.
6. W. D. Pilkey, Peterson's Stress Concentration Factors, 2nd Edition, John Wiley & Sons, 1997.
7. John M. Barsom and Stanley T. Rolfe, Fracture and Fatigue Control in Structures: application of fracture mechanics, 3rd Edition, ASTM, 1999.
8. http://www.afgrow.net/applications/DTDHandbook/sections/page2_2_0.aspx.
9. Toshihiko Ono and Kohichi Tanaka: "Effective of Scribing wheel dimensions on the cutting of AMLCD glass substrates" Technical Information Paper, Corning Incorporated, 306, November 2004.
10. Stress Intensity Factors Handbook, Vol. 1, Committee on Fracture Mechanics, The Society of Material Science, Japan, Pergamon Press (1986).

APPENDIX-1

Calculations

For a = 4.039 mm

$(\Delta K)_{pipe}$ = 12.72 MPa√m   {Calculated}

And $(\Delta K)_{beam} = \Delta\sigma \sqrt{(\pi a)}\, F(\alpha)$

$$\Rightarrow 12.72\times10^{6} = \Delta\sigma\sqrt{\pi\times4.039\times10^{-3}}\times F(\alpha) \qquad \text{(A1)}$$

For a = 4.039 mm and w = 9 mm

α = 0.448 and $F(\alpha)$ = 1.359

Now putting the value of $F(\alpha)$ in eqn. (A1) we get

$\Delta\sigma$ = 83.05 MPa

Since $\Delta\sigma = \sigma_{max} - \sigma_{min}$ and $R = \frac{\sigma_{max}}{\sigma_{min}} = 0.1$ {for considered test R=0.1}

$$\sigma_{max} = \frac{\Delta\sigma}{(1-0.1)} = 92.28\,\text{MPa} \qquad \text{(A2)}$$

From eqn. (2) $$\sigma_{max} = \frac{3P\lambda}{tw^{2}} \qquad \text{(A3)}$$

From (A2) and (A3) $$P = \frac{92.28\times10^{6}\times(15\times10^{-3})\times(10\times10^{-3})^{2}}{3\times\lambda}$$

And $\lambda = 125 \text{ X } 10^{-3}$ m $\Rightarrow P = 369.12N$