CONTOUR INTEGRAL EVALUATION OF CENTRE CRACKED RECTANGULAR PANEL UNDER REMOTE TENSILE STRESS AND THERMAL STRESS: PARAMETRIC STUDY

Mahanthesha.T.[1], Mohammed Imran[2], Mohamed Haneef[3]

[1]*PG Student, Department of Mechanical Engineering, Ghousia College of Engineering, Ramanagaram, Karnataka, India*

[2]*Assistant Professor, Department of Mechanical Engineering, Ghousia College of Engineering, Ramanagaram, Karnataka, India*

[3]*Principal & Professor, Department of Mechanical Engineering, Ghousia College of Engineering, Ramanagaram, Academic Senate member-VTU, Karnataka, India*

Abstract

The Linear Elastic Facture Mechanics (LEFM) approach applies to fracture of materials which are naturally elastic up to failure: glass, ceramics, plastics, and high strength metals with limited amount of ductility before getting fracture. LEFM cannot be prolonged in situations having higher plastic deformation before fracture. Hence LEFM is abandoned & therefore Elastic Plastic Fracture Mechanics (EPFM) is embraced. Specifically the non-linear energy release rate designated as J is analyst's choice and J evaluation is the subject of the study. The J-line integral properties with thermal stress application is determined. Similarly a superposition technique is offered for problems having thermal stress fracture that also includes crack surface tractions. In this technique, J-integral path should have crack surface segments. Finite element modeling well-defined in this technique is the examiner's decision of materials models (constitutive law and failure criteria), constraint equations, finite elements, analysis procedures, meshes, governing matrix equations and methods for solving, specific pre and post-processing choices presented in a selected commercial Finite Element Analysis program (ABAQUS software) for the evaluation of J for a rectangular panel with a meridional crack. The Finite Element Model developed using ABAQUS and evaluation using J-integral method in ABAQUS is validated using a benchmark namely center cracked panel under remote tensile stress & thermal stress for which target solutions are available in the NAFEMS document. Extensive numerical results of a parametric study are presented to show the effect of crack length & material models.The approach is authenticated and validated utilizing benchmarks, a conventional typically investigated problems by means of well-known target results.

Keywords: *Finite Element Analysis, Ductile Fracture, Elastic Plastic Fracture Mechanics, ABAQUS, Thermal Stress, Nonlinear Energy Release Rate (J); Material Model, Crack Analysis.*

--***--

1. INTRODUCTION

Fracture is a failure mode due to unstable crack propagation under different applied stress. Fracture mechanics affords a procedure for the prediction, prevention& control of fracture in different materials, components & structures subjected to static, dynamic& sustained loads.

LEFM is useableif non-linear material deformation is limited to a lesserareaadjacent to crack tip. In numerous materials it's difficult to describecrack behaviour through LEFM. Limited plasticity at the crack tip was accounted for through plasticity correction. The approach is more than sufficient for a wide variety of problems, notably Fatigue Crack Growth(FCG) prediction for which the maximum stress due to applied loads are less than 30 percent of the yield stress. If the load exerted on the ductile metals exceeds elastic range, the initial response of linear stress will provide path to a complex non-linear response. Hence LEFM is abounded and EPFM is embraced for intended analysis of computational fracture mechanics.

1.1 Elastic Plastic Fracture Mechanics

EPFM relates to those material which exhibits time-independent, plastic deformation (i.e.,non-linear behaviour). EPFM consists of two parameters CTOD and non-linear energy release rate (J). Both these parameter defines crack-tip surroundings in elastic plastic materials, and theymay be used as fracture measure. Critical no's of CTOD or J provides nearly size independent quantity of fracture toughness, even incomparatively largecrack-tip plasticity values. There are limited J and CTOD applicability, however these limits are far less limiting than validity necessities of LEFM.

1.2 Non-linear Energy Release Rate (J)

Non-linear elastic body'senergy release rate which contains crack is known as non-linear energy release rate (J). In 2D plane problems J is evaluated as a contour integral as shown in Fig-1.

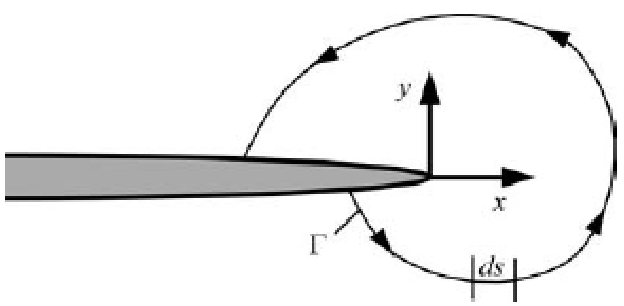


Fig-1: Arbitrary Contour Around Crack-tip.

$$J = \int_{\Gamma} \left(w dy - T_i \frac{\partial u_i}{\partial x} ds \right) \quad (1)$$

Where,
w= Strain energy density
T_i= Traction vectorcomponents
u_i= Components of displacement vector
ds= Length increment alongside contour Γ

Strain energy density here will be defined as

$$w = \int_0^{\varepsilon_{ij}} \sigma_{ij} \, d\varepsilon_{ij} \quad (2)$$

where, σ_{ij} and ε_{ij} are tensors of stress and strain respectively. Traction is a stress vector on a given point of the contour i.e., if we construct a FBD of material inside the contour, *Ti* willdescribe the stresses acting near boundaries. The traction vector components will be,

$$T_i = \sigma_{ij} n_j \quad (3)$$

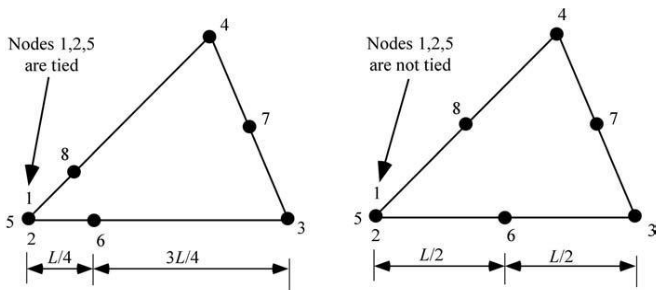
Here n_j is unit vector components acting normal to Γ. Rice showed that significance of J is independent of path of integration near crack. ThereforeJis called aspath-independent integral.

1.3 Computational Elastic-plastic Fracture Mechanics

Best FEA proceduresassumed by design engineers are restricted to linear investigation. Such linear investigationaffords an satisfactoryestimate of real-life features for utmost problems design engineers come across. But, sometimesadded challenging problems arise, and requires non-linear methodology. There are 3 types of non-linearity:

a) Material non linearity,
b) Geometric non linearity and
c) Contact non linearity.

The present analysis mainly focuses on material non-linearity. In elastic analysis, the crack-tip nodes are usually tied, andmiddle-side nodes moved to the quartier point positions as shown in Fig-2(a).Such adjustmentoutcomes in a 1/√r element strain singularity, thatimprovesaccuracy. When a plastic-zone forms, there will be no-more existence of 1/√r singularity near crack-tip. Accordingly, singular elastic elements aren't suitable for elastic-plastic studies. Fig- 2(b)displays an element which exhibits required strain singularity in fully plastic conditions.



a) **b)**

Fig-2: Elastic and Elastic-Plastic analysescrack-tip Elements. Element (a) Produces 1/√r elastic Strain Singularity, while (b) Exhibits a 1/r plastic Strain Singularity

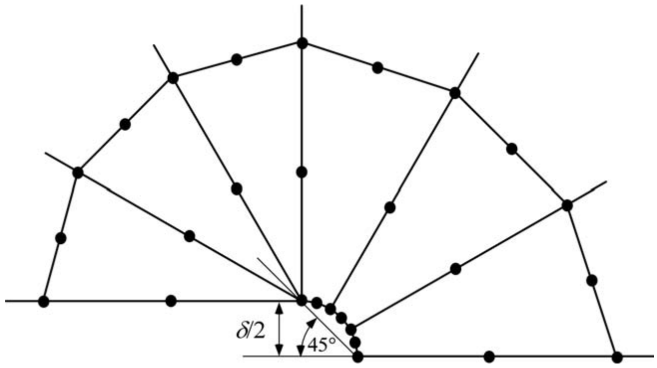


Fig-3: Plastic Singularity ElementsDeformed Shape.

Note that 3 nodes inhabit same point in space. Fig-5 illustrates analogous situation for 3D, where a 20-noded hexahedral Solid element is degenerated into a 15-noded wedge element.

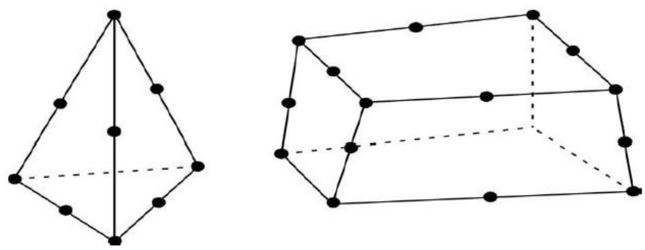


Fig-4: Common Three-Dimensional Continuum Finite Elements: (a) Tetrahedral Element and (b) Brick Element

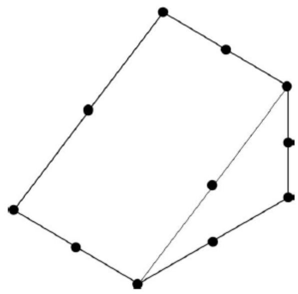


Fig-5: Degeneration of a Brick Element into a Wedge (SPENTA 15)

2. FINITE ELEMENT MODEL

FE Modeling here isexaminer's choice of material models, constraint equations, finite elements,analysis procedures, meshes, governing matrix equations & its methods of solving, specific pre & post-processing choices available inselected commercial FEA software in order to find J. A standard problem having known target results in form of graphs,formulae or tables achieved using analytical methods, experimental techniques, and computational procedures. Used to validate FE modelling for engineering analysis of candidate components & structures using a chosen commercial FEA software ABAQUS.

3. BENCHMARK

A standard test problem having known target results in the form of graphs,formulae or tables achieved using analytical methods, experimental techniques, and computational procedures. Used to validate finite element modelling for engineering analysis of candidate components and structures using a chosen commercial FEA software ABAQUS.

The Benchmark problem selected is Centre Cracked Rectangular Panel under Remote Tensile Stress& Thermal Stress.

A centre cracked rectangular panel of width 2W = 100 mm, length 4W = 200 mm, crack length 2a = 20 mm, (a/w = 0.2) ispresented in Fig-6. The properties material properties of steam generator grade alloy steel are Elastic modulus E=2e5MPa, poisons ratio ν=0.3and yield stress σ_y=271MPa as shown in Fig-7. A plane strain state is assumed and the material is elastic perfectly plastic.

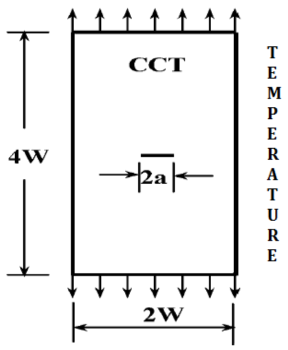


Fig-6: Centre Cracked Rectangular Panel Subjected to Remote Tensile Stress & Thermal Stress at right side of the Panel

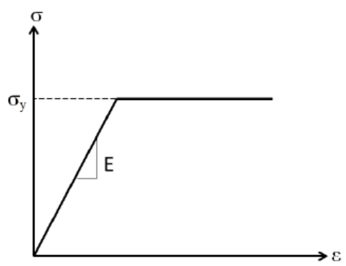


Fig-7: Elastic Perfectly Plastic Materials Stress-Strain Curve

Table-1: Properties of Steam Generator grade Steel

Material Properties	Applied Load
E = 2x10^5 MPa *ν* =0.3 σ_y = 271MPa	*Applied Stress* $\sigma_a = 0.925\sigma_y$ *Temperature* T = 100 DegC

3.1 Finite Element Model

The typical finite element model is presented in Fig-8 and a refined mesh of STRIA 6 elements is generated near the crack tip, and a compatible mesh of QUAD8 elements is used in the rest of the domain. Element type: CPE8R (An 8-node bi-quadratic plane strain quadrilateral, reduced integration.) Number of elements is 7854 and nodes are 39820. Number of elements around the crack-tip = 72.

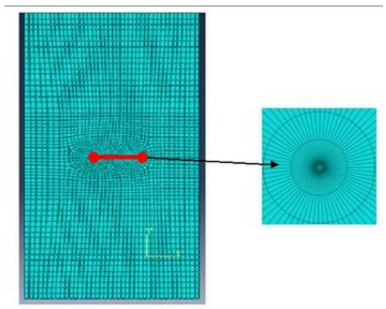


Fig-8: FE Model &crack-tip Singularity Elements.

3.2 Validation

Table-2: J Value for Each Load Step

Sl. No.	Load factor σ/σ_y	J J/mm^2	Normalized J $EJ/(a*\sigma_y^2)$
1	0.1	1.139	0.155
2	0.2	3.599	0.490
3	0.3	5.728	0.780
4	0.4	8.104	1.103
5	0.5	12.160	1.656
6	0.6	16.720	2.277
7	0.7	21.840	2.974
8	0.8	29.090	3.961
9	0.9	40.480	5.512
10	0.925	45.830	6.240

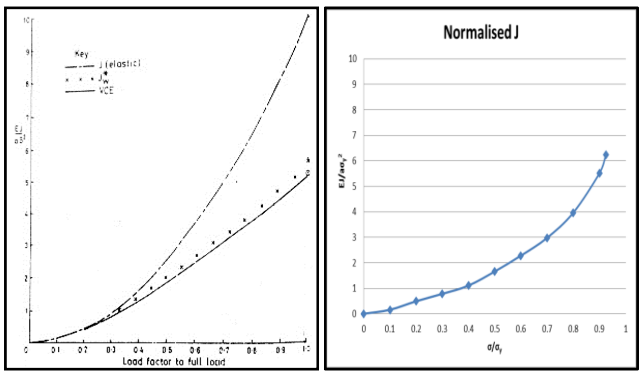


Fig-9: Graph of Normalized J v/s Load Factor Using FEM and Graph of Normalized J v/s Load Factor (NAFEM's Manual)

For each load step, the J is evaluated around the crack-tip and tabulated in table 1. The Normalized J curve is presented in Fig-9 and compared with the master curve in the NAFEMS document [1]. These results are found to closely match with the target solutions reported.

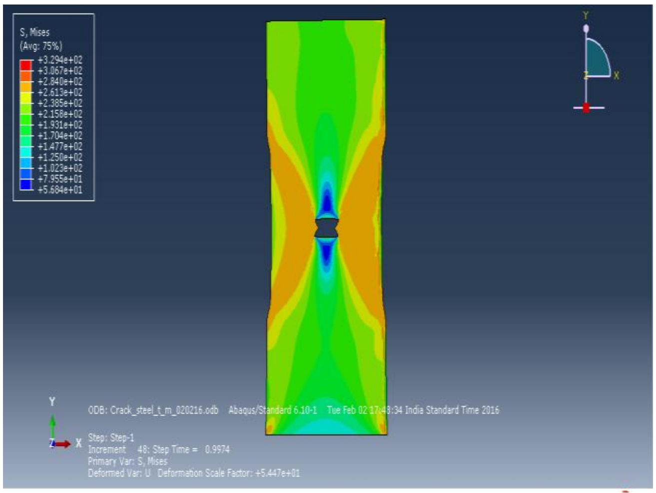


Fig-10: Von-Misses Plot & Line Contours Plot

For a maximum tensile stress of 250.675MPa on performing elastic plastic stress analysis maximum stress at the crack tip and plastic zone shape is shown in Fig- 10.

4. PARAMETRIC STUDY

4.1 By Changing Crack Geometry

To study the effect of crack geometry (a:w ratio) on the behaviour of the panel, computations are performed with the same elastic perfectly plastic material and same loading condition as in the benchmark. As the a:w ratio increases normalized J tends to increase. It was found that as a:w ratio increases plastic failure load tends to decrease.

From the results it is observed that the analysis terminates at a value 216.8MPa for crack geometry a:w = 0.3 which is the maximum value for elastic plastic material.

Table-3: Normalized J value for different a:w ratios for Steel Material

Sl No.	Load factor σ/σ_y	J J/mm^2		Normalized J $EJ/(a*\sigma_y^2)$	
		a:w 0.2	*a:w 0.3*	*a:w 0.2*	*a:w 0.3*
1	0.1	1.139	1.320	0.155	0.120
2	0.2	3.599	3.894	0.490	0.353
3	0.3	5.728	7.111	0.780	0.646
4	0.4	8.104	9.841	1.103	0.893
5	0.5	12.160	13.590	1.656	1.234
6	0.6	16.720	16.180	2.277	1.469
7	0.7	21.840	20.090	2.974	1.824
8	0.8	29.090	25.630	3.961	2.327
9	0.9	40.480	30.950	5.512	2.810
10	0.925	45.830	33.460	6.240	3.037

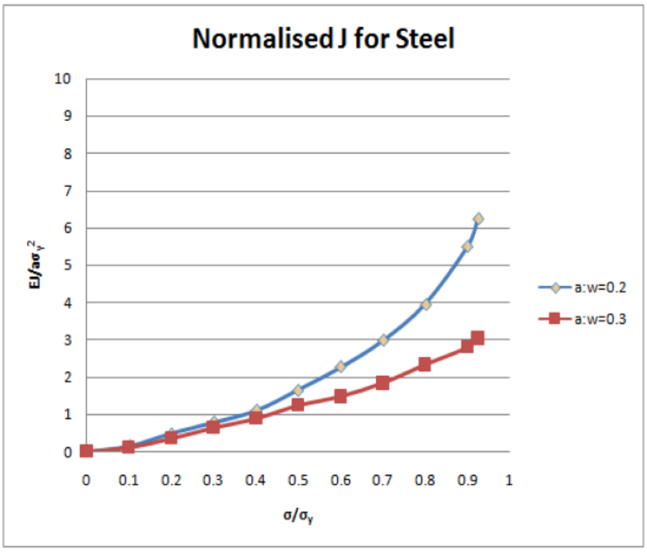


Fig-11: Normalized J v/s Load Factor for Different a:w ratios for Steel Material

4.2 For Aluminium alloy 6061T1 Material

For Aluminium alloy 6061T1 we will analyse the effect of crack geometry for different a:w ratio.Computations are performed considering elastic perfectly plastic material and same loading condition as in the benchmark.

Table-4: Properties of Aluminium alloy 6061T1

Material Properties	Applied Load
$E = 0.7x10^5$ MPa v =0.33 σ_y = 96MPa	*Applied Stress* $\sigma_a = 0.925\sigma_y$ *Temperature* T= 100 Deg C

From the results it is observed that the analysis terminates at a value 75.6MPa for crack geometry a:w = 0.3 which is the maximum value for Aluminium alloy material.

Table-5: Normalized J value for different a:w ratios for Aluminium alloy Material

Sl No.	Load factor σ/σ_y	J J/mm^2		Normalized J $EJ/(a*\sigma_y^2)$	
		a:w 0.2	a:w 0.3	a:w 0.2	a:w 0.3
1	0.1	0.618	0.264	0.235	0.067
2	0.2	1.556	0.995	0.591	0.252
3	0.3	2.711	2.022	1.030	0.512
4	0.4	3.926	3.224	1.491	0.816
5	0.5	5.382	4.857	2.044	1.230
6	0.6	6.982	6.404	2.652	1.621
7	0.7	9.208	7.783	3.497	1.971
8	0.8	11.730	9.142	4.455	2.315
9	0.9	14.220	11.130	5.400	2.818
10	0.925	15.210	12.210	5.776	3.091

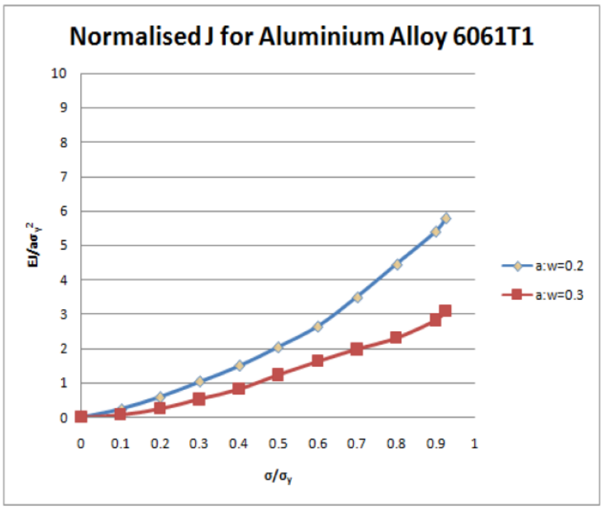


Fig-12: Normalized J v/s Load Factor for Different a:w ratios for Aluminium alloy material

5. CONCLUSION

Based on the above work following are the conclusion drawn

- For the center cracked panel with the elastic perfectly plastic material model, with remote tensile stress and thermal stress, the normalized J value is around 5.7 for Aluminium alloy material&6.24 for Steel material. Results are matching with master curve (NAFEMS India) is 5.
- The location of maximum von-Misses stress at the crack tip & for a maximum tensile stress of 250.675MPa for Steel & 88Mpa for Aluminium alloy for the panel is under fully plastic yielding condition under the influence of temperature loads.
- We infer from the parametric study that there is significant influence of temperature & the crack geometry (a:w ratio) on the stress distribution and nonlinear energy release rate (J) under remote tensile stress & thermal stress.
- The results reported are believed to be accurate as converged solutions are considered.
- The future work consists of studying the variation of J-parameter along the crack front for a pressurized steam generator tube with a meridional crack.
- A pressurized steam generator tube with a inclined crack may be studied.
- A pressurized steam generator tube with a circumferentialthrough wall thickness crack may be studied.
- Also a CFD analysis is needed for the complete parametric study of the behaviour of steam generator having axial crack, inclined crack, circumferential crack, etc.,

ACKNOWLEDGEMENTS

I am whole heartedly thankful to my parents, friends & whoever helped me a lot in completing this paper.I also extend my faithful thanks for the moral support and help provided at times of need.

REFERENCES

[1] E. Hinton (Editor), NAFEMS Introduction to Non Linear Finite Element Analysis, NAFEMS, Glasgow, UK, 1992.

[2] T. L. Anderson, Fracture mechanics: Fundamentals and applications, Taylor and Francis group, CRC Press, Boca Raton, 3rd ed., 2005.

[3] ABAQUS 6.10, GUI Toolkit User's Manual Volume I: A-H, Dassault Systemes, France, 2011.

[4] Zahoor A., Ductile fracture handbook, Axial through wall crack, Electric Power Research Institute report NP-6301-D, Vol. II, 580-589, 1989.

[5] J.H. Lee,Y.W. Park , H.D. Chung , Y.H. Choi, S. Majumdar, Investigation on Behavior of Trapezoidal Through-Wall Cracks Under Constant Pressure Loading, International Journal of Pressure Vessels and Piping, 85,2002, 606– 611.

[6] S. Aoki, K. Kishimoto, M. Sakata, Elastic-plastic analysis of crack in thermally-loaded structuresEngineering Fracture Mechanics 12/1982:405–413.

[7] V. Kumar, B.I. Schumacher, M.D. German, Effect of thermal and residual stresses on the J-integral elastic-plastic fracture analysis, Computers &Structures 01/1991: 487–501.

[8] W.K. Wilson, I.W. Yu., The use of J-integral in thermal stress crack problems, International Journal of Fracture (07/1979):377-387.

BIOGRAPHIES

Mr. Mahanthesha T., PG Student (Machine Design), Department of Mechanical Engineering, Ghousia College of Engineering, Ramanagaram, Karnataka, India.

Mr. Mohammed Imran, Asst. Prof., Department of Mechanical Engineering, Ghousia College of Engineering, Ramanagaram, Karnataka, India

Dr. Mohamed Haneef, Principal & Professor, Department of Mechanical Engineering, Ghousia College of Engineering, Ramanagaram, Karnataka, India