ISSN: 0974-2115
www.jchps.com Journal of Chemical and Pharmaceutical Sciences

Computation of stress intensity factor on different crack geometry in Al7020 using finite element method

Jegan Dhas P, Vel Murugan, Leo Bensingh, Ashok rugul E
Department of Mechanical Engineering, Jeppiaar Engineering College, Chennai.
***Corresponding author: E-Mail:jegandhas749@gmail.com**

ABSTRACT

The study is a preliminary effort which investigates the effect of variations of crack geometry on the stress intensity factor on a simple thin plate subjected to a constant, uniform, uniaxial tensile load. The geometry of the central crack is varied to produce models ranging from central to sharp cracks tip. Due to symmetry, the models are reduced to quarter symmetric, meshed and refined at the edges of the cracks for clearer, detailed analysis and solution accuracy. Finally, a uniform pressure is applied at the top and the model is constrained with symmetric boundary conditions at the left and bottom. These numerical results are compared to those obtained from analytic fracture mechanics procedures and are found to be consistent

KEYWORDS: MIG (Metal Inert Gas), ANSYS

1. INTRODUCTION

Failure of the engineering structures is caused by cracks, which is depending on the design and operating conditions that extend beyond a safe size. Cracks present to some extent in all structures, either as a result of manufacturing defects or localized damage in service. The crack growth leads to a decrease in the structural strength. Thus, when the service loading to the failure of the structure. Fracture, the final catastrophic event takes place very rapidly and is preceded by crack growth, which develops slowly during normal service conditions.

2. METHODS & MATERIALS

Methodology: The methodology used to investigate the mechanics of crack propagation consists of the following steps:

- material selection
- joining the metals
- Flaw implementation
- Load apply
- Crack propagation
- Calculation of stress intensity factor
- Interpretation of results

Welded Aluminium Alloy 7020 Plate: The welded aluminum cracked body is given below, here W is width of the weld beam, L is full length of the body and t is thickness of the body and crack is mentioned at the weld beam.

The aluminium alloy 7020 base metal is welded by using MIG welding process. Usually the MIG welding is best suitable for weld the all alluminium alloy. The weld beam and the heat affected zone area properties are changed and it will be depend upon the selection of filler metal. The MIG welding process can be done 2 mm to 5 mm alluminium sheet. The dimension of the base plate is does not change after welding. At the solidification or tempering (ageing) of weld beam defects will be happened such as porosity, cracks due to shrinkage.

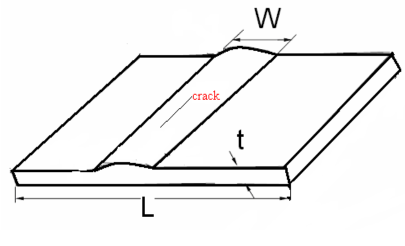

Figure.1.3D model of welded plate aluminium alloy 7020

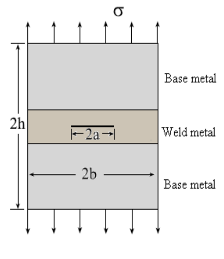

Figure.2.Finite Plates With A Centre Through Crack Under Tension

Finite Element Modeling Of Cracked Body: The finite element analysis the area of base metal and weld metal joined by GLUE option. Crack meshes usually use 8-node quadratic elements for two dimensional problems.

The creation of meshing in finite element models is the most important step in the entire analysis since the decision made at this point will affect the accuracy and the economy of the solid model. Figure.5 shows a meshed model of a quarter symmetric rectangular plates. The meshing was manually setup manually by picking up lines and discretizing them, and finally an Automatic free mesh was applied to the entire the model. The spider web configuration as shown in figure .5 below.

And the figure.5 shows a uniform and constant a uni-axial tensile load of 100MPa applied to the top of the rectangular model. This is indicated the arrows. The model was further constraint at the left and the bottom edges with zero structural displacement symmetric boundary conditions. The arrows shown in the figure .5 indicates the zero displacement boundary conditions. This condition is also shown in the figure by smalls yellow to green arrows.

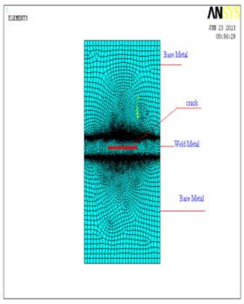

Figure.3. Finite element model of full length cracked body

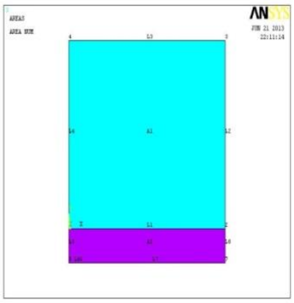

Figure.4. Quarter symmetric model of welded cracked body

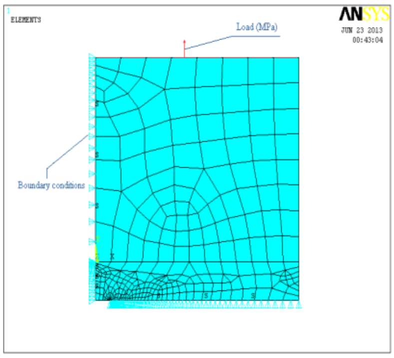

Figure.5.Quarter symmetrical model of welded aluminum alloy 7020 with a crack and uni-axial tensile load of 100 Mpa

The red arrows at the bottom of the rectangular model show the symmetric arrangement of the tensile loads at the edge of the crack. The same procedure for loads and boundary conditions was performed to all the other models used in the study. Finally, all these finite element models were solved.

Using the numerical package Ansys 10, we can determine the value of the stress intensity factor K_I for the same geometry. This is computed using finite elements on a mesh with quadratic triangular elements on the vicinity of the crack tip, and quadratic rectangular elements everywhere else. Quarter point elements, formed by placing the mid-side node near the crack tip at the quarter point, were used to account for the crack singularity.

Table.1. Comparison of numerical value of base metal and weld metal with theoretical value of stress intensity factor at different loading condition

Loads (MPa)	Numerical value of base metal (MPa√m)	Numerical value of weld metal (MPa√m)	Stress intensity factor of both metal(MPa√m)	Theoretical value (MPa√m)
25	2.88	0.26	3.14	2.94
50	5.09	0.50	5.59	5.88
75	8.09	0.74	8.83	8.82
100	11.07	1.01	12.08	11.76
125	14.02	1.23	15.25	14.70
150	16.19	1.56	17.75	17.64
175	19.95	1.83	21.78	20.57
200	22.92	2.10	25.02	23.51
225	25.89	2.37	28.26	26.45
250	28.85	2.65	31.5	29.39
275	31.81	2.92	34.73	32.33
300	34.78	3.19	37.97	35.27
325	37.75	3.46	41.21	38.21
350	40.71	3.74	44.45	41.15

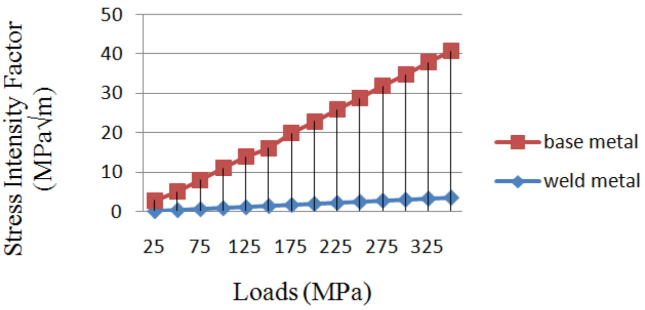

Figure.6. Comparison of numerical value of stress intensity factor of weld metal and base metal at various loading condition

By adding the stress intensity factor of base metal and weld metal, we can get the total stress intensity factor of welded metal. The comparison of numerical value with theoretical value of stress intensity factor is given below by graph.

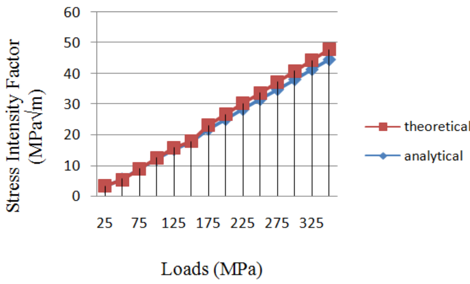

Figure.7. Comparison of theoretical and numerical value of stress intensity factor of welded metal at various loading condition

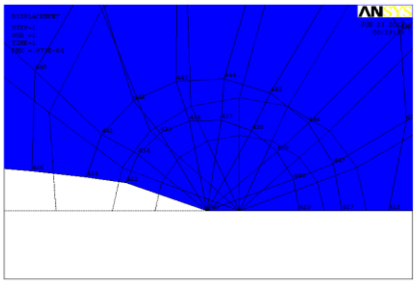

Figure.8. Displacement diagram of welded aluminium alloy 7020

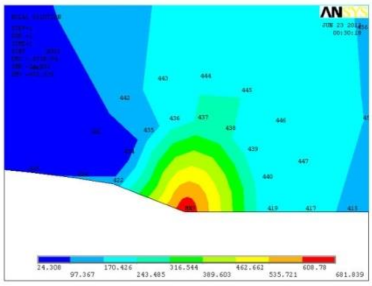

Figure.9. Nodal solution of alluminium alloy 7020 at 100 MPa

The different colours Indicates the minimum to maximum stress intensity of the entire cracked body
The maximum stress intensity is at the vicinity of the crack tip, it is denoted by the red colour, and the minimum stress intensity is denoted by the blue colour.

Extrapolation Method To Determine Stress Intensity Factor: Two dimensional problem of fracture mechanics are solved through finite element method, in general by using triangular, quadrilateral or isoparametric elements. The elements may be 3 node or 6 node triangle, 4 node quadrilateral or 8 node isoparametric elements.

$\sigma x = (k_1 \div \sqrt{(2\times\pi\times r)})\ \cos(\theta\div 2)\ [1-\sin(\theta\div 2)\ \ \sin(3\theta\div 2)]$ (7)

$\sigma y = (k_1 \div \sqrt{(2\times\pi\times r)})\ \cos(\theta\div 2)\ [1+\sin(\theta\div 2)\ \ \sin(3\theta\div 2)]$ (8)

$\tau xy = [k_1 \div \sqrt{(2\times\pi\times r)}]\times\sin(\theta\div 2)\cos(\theta\div 2)\cos(3\theta\div 2)$ (9)

K_1 is determined through σ_x, σ_y, τ_{xy} various element of center cracked specimen.
σ_x = x component applied stress (MPa)
σ_y = y component applied stress (MPa)
τ_{xy} = xy applied shear stress (MPa)

Table.2. Stress values of one finite element model PRNSOL command

	σ_x	σ_y	τ_{xy}
1	169.29	252.89	-13.843
2	118.84	247.89	16.881
3	89.718	216.43	12.588
4	60.058	192.4	5.5266
5	43.83	163.19	3.0990
6	34.537	146.84	0.74605
7	23.632	130.07	-2.9350

Very near the crack tip the finite element solution tends to be inaccurate due to its inability to model singular nature of model well. An improved evaluation of K_1 can be done from the plot of K_1 vs r. thus the K_1 curve can be extrapolated to find K_1 at r=0 for a fixed θ.

Calculation

$\sigma x = [K_1 \div \sqrt{(2\times\pi\times r)}]\cos(\theta\div 2)\ [1-\sin(\theta\div 2)\ \ \sin(3\theta\div 2)\ ]$

Here $\theta = 30^0$

r = 0.0002,……………,0.0014 (m)

The values of stress intensity factor K1 given below the table,

Table.3. Stress intensity factor of different node stress solution

σ_x (MPa)	Stress intensity factor K_1 (MPa√m)
169.29	10.046
118.84	9.979
89.718	9.22
60.058	7.13
43.83	5.819
34.537	5.022
23.632	3.71

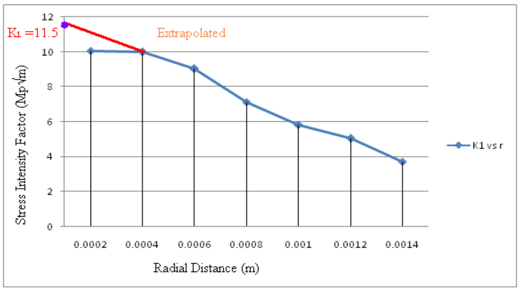

Figure.10. Stress intensity factor displacement extrapolation method

The value of stress intensity factor K_1=11.5 MPa√m

3. COMPARISON OF RESULTS

Model Calculation (Theorical)

Applied stress σ = 100MPa

Crack length a = 4.4 mm

Width b= 62.5 mm

The theoretical value of stress intensity factor,

$K_I = \sigma\sqrt{(\Pi a)}(1-0.1(a/b)^2+0.96(a/b)^4)\ \sqrt{1/\cos(\pi(a/b))}$

$K_I = 100\sqrt{(\pi\times0.0044)}(1-0.1(0.0704)^2+0.96(0.0704)^4)\ \sqrt{1/\cos(\pi(0.0704))}$

K_1 = 11.75 MPa√m

The discrepancy is,

$e = K_I^{(theoritical)}-K_I^{(ansys)}/K_I^{(theoritical)}$

e = 11.75 – 12.9407/11.75

e=10%

Stress intensity factor through displacement method K_1 = 11.70MP√m

4. CONCLUSION

The stress intensity factor is a most useful parameter for determining the effects of cracks on fracture and fatigue. The finite element computational results give good predictions of the expected normal stress intensity factors compared to the analytical results.

Stress intensity is critically important fracture mechanics parameters used by materials engineers and designers. We saw that there are a lot of factors that determine fracture of a material. K_I is an unique material property, that is used by engineers to design and manufacture products for durability and safe operation

This work clearly demonstrates the robustness of the finite element method in handling real life problem. The numerical results obtained using the finite element meshed are in good agreement with theoretical work on done on the crack geometry.

REFERENCES

Al-Mukhtar AM, Henkel S and Biermann H, A Finite Element Calculation of Stress Intensity Factors of Cruciform and Butt Welded Joints for Some Geometrical Parameters, Institute of Materials Engineering, Technische Universität, Germany.

Ansys Tutorial, 2-D Fracture Analysis, ANSYS Release 7.0, University of South Alabama.

Carlos Bloem and Mary Vergara, Aluminium 7020 Alloy and Its Welding Fatigue Behaviour, Universidad de Los Andes, Venezuela.

Jesus Toribio, Javier Ayaso, Beatriz Gonzalez, Juan-Carlos Matos and Diego Vergara, Critical stress intensity factors in steel cracked wires, Department of Materials Engineering of the University of Salamanca, Spain.