

Fatigue Crack Propagation Analysis on Aluminium Alloy-6062 Using Extended Finite Element Modeling

Madhavan P ^[1], Venkatesan T ^[2], Rajendran R ^[3]

Department of Mechanical Engineering, B.S.Abdur Rahman University, Chennai, Tamilnadu, India ¹

Assistant Professor (sel.Gr.) Department of Mechanical Engineering, B.S.Abdur Rahman University, Chennai, Tamilnadu, India ²

Professor, Department of Mechanical Engineering, B.S.Abdur Rahman University, Chennai, Tamilnadu, India ³

ABSTRACT: A general methodology is proposed in this paper for fatigue crack propagation simulation using Extended Finite Element Analysis (XFEM). The aim of this study is to characterize the fatigue behavior of specimen and to develop a model for fatigue life prediction. Numerical XFEM models used in the fatigue test were developed with the aim to predict the fatigue behavior of Aluminium Alloy-6062 specimen. Experimental test is carried out on Aluminium Alloy-6062 specimen to validate the XFEM model. The whole of the analysis has been carried out using fully developed Abaqus. A parametric code is developed for which the user could provide input values to get the analysis result automatically. The comparison between experimental and design values shows a variation of 8%.

KEYWORDS: Fatigue crack propagation, XFEM, S-N Curve, Aluminium Alloy-6062

I. INTRODUCTION

Fatigue is defined as the progressive and localized structural damage that occurs when a material is subjected to cyclic loading. Many components of modern world are subjected to fluctuation loads and they may fail through fatigue, in fact failure through fatigue is so common that more than 80% failures are caused by fluctuating load. Fatigue analysis is essential to find out the life time of the specimen and its breaking properties. XFEM allows to study crack growth along an arbitrary, solution-dependent path without need to remesh your model. Modeling stationary discontinuities, such as a crack, with the conventional finite element method requires that the mesh conforms to the geometric discontinuities. The XFEM software which is used for this study is Abaqus. The Abaqus is unified Finite Element Analysis (FEA) product suite which offers powerful and complete solutions for both routine and sophisticated engineering problems. The input for design analysis is taken from the experimental result. The load input value is divided into 8 values which improves the accuracy of the result. A parametric code is developed for which the user could provide input values to get the analysis result. The main objective of the study is to determine the bending stress between experimental and design values.

II. MATERIALS & METHOD

2.1 Materials Used

Aluminium Alloy-6062 is the material used for this study. The chemical composition of the Aluminium Alloy-6062 is listed in table 1.

Table 1: Chemical Composition of Aluminium Alloy-6062

Element	Composition (wt %)
Silicon	0.4-.08
Iron	0.7
Copper	0.15-0.4
Manganese	0.15
Magnesium	0.8-1.2
Chromium	0.04-0.35
Zinc	0.25
Titanium	0.15
Aluminium	Allowance

2.2 Specimen Preparation

Aluminium Alloy-6062 is the specimen prepared with the standard dimension based on American Society for Testing and Materials (ASTM) E-606 which is a low cycle fatigue. The standard dimensions are shown in figure 1.

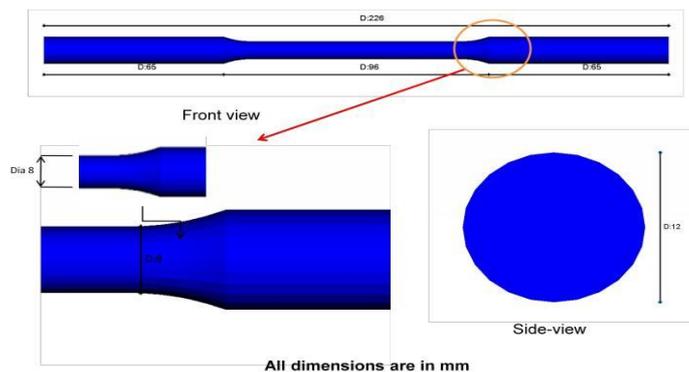


Figure 1: Dimension of Specimen

2.3 Fatigue Test

The equipment used for fatigue test is Rotary Bending Machine. The machine parameters are listed in table 2.

Table 2: Parameters of Machine

Maximum Bending Moment	200 kg.cm
Load (Adjustable)	5-40 kg
Gripping Diameter of the Specimen	12 mm
Testing Diameter of the Specimen	8 mm
Length of Test Specimen	226 mm
Rotating Speed Applied to Test Specimen	4200 rpm
Accuracy of Applied Bending Moment	±1%

The S-N curve is generated from the experimental results. The stress is calculated by using the formula

$$\sigma = \frac{P}{A} \quad \text{---}$$

Where,

P = load in N

A = area of the specimen in mm²

2.4 XFEM

The Abaqus is unified Finite Element Analysis (FEA) product suite which offers powerful and complete solutions for both routine and sophisticated engineering problems. The formula used to calculate the load input value is

$$F = W * \sin(2\pi\omega t)$$

Where,

F = force acting on the specimen in N/m

W = weight acting on the specimen in Kg

t = time taken in seconds

ω = frequency (1/t) in Hertz

The load value is divided into 8 input values which improves accuracy of the result. Load acting on a node taken at 8 consecutive points during a cycle of specimen (i.e., workpiece is stationary while the load is rotating around the workpiece). Force acting on the specimen in the 1st interval of one cycle:

$$F_1 = W * \sin(2\pi\omega t_1)$$

Where,

F_1 = force acting on the specimen during 1st cycle in N/m

t_1 = time interval in first cycle (t/8) in seconds

III. RESULTS & DISCUSSION

3.1 Experimental Results

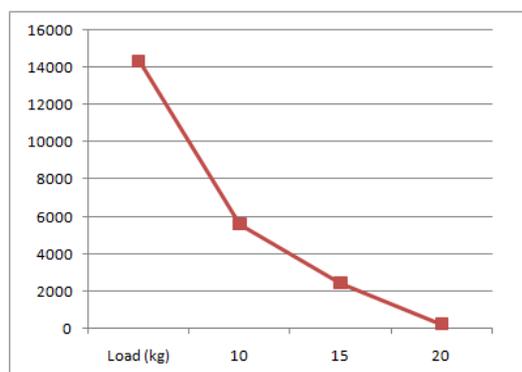
The readings are noted from the experiment such as number of cycles, time with respect to change of load. The experimental results are listed in table 3.

Table 3: Experimental Observation

S.No.	Load (kg)	No. of Cycles	Time (sec)
1	10	14342	172
2	15	5597	78
3	20	2426	42
4	40	236	3

The relation between the load and the no. of cycles are shown in graph 1.

Graph 1: Load vs. No. of Cycles



3.2 Design Results

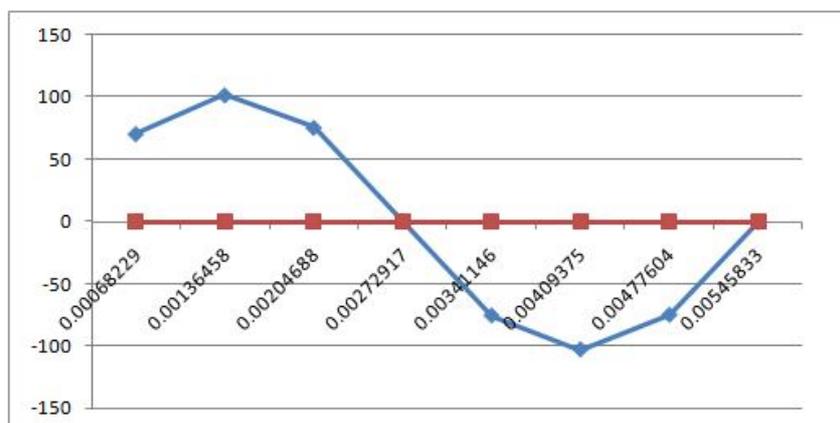
To get accurate result one cycle of fatigue loading condition is divided into 8 intervals. For particular loading conditions we have 8 force values to be entered into the loading inputs. The 8 input values are listed in table 4.

Table 4: Calculation for Force and Time

S.No.	Force (F _x) [N]	Time (t _x) [sec]
1	70.66	0.00068229
2	101.87	0.00136458
3	75.7	0.00204688
4	0.17498	0.00272917
5	-75.53	0.00341146
6	-102.87	0.00409375
7	-74.907	0.00477604
8	-0.3499	0.00545833

The 8 load input values tends to form a full cycle. The relation between the force and time cycle are shown in graph 2.

Graph 2: Force vs. Time Curve

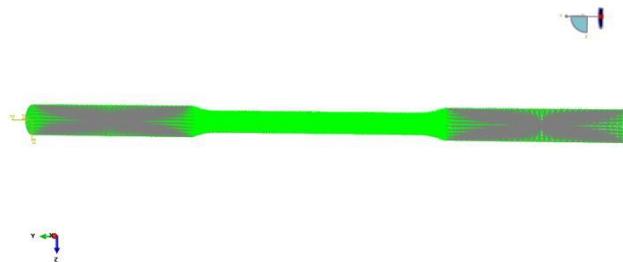


◆ Force [N]
■ Time [sec]

The kinematic coupling is fixed at both the ends of the specimen. The load condition is made to set, the input is made to given as input in the software which tends to give the output. The

maximum bending stress is taken as the result and compared with the experimental result. Kinematic coupling at both ends of the specimen are shown in figure 2.

Figure 2: Kinematic Coupling



The load input is given to the specimen, and then the program is made to run which it shows the left end loading and right end loading.

Figure 3: Left End Loading



Figure 4: Right End Loading



The maximum principal stress is noted from the result which tends to give the result.

Figure 5: Maximum Principal Stress

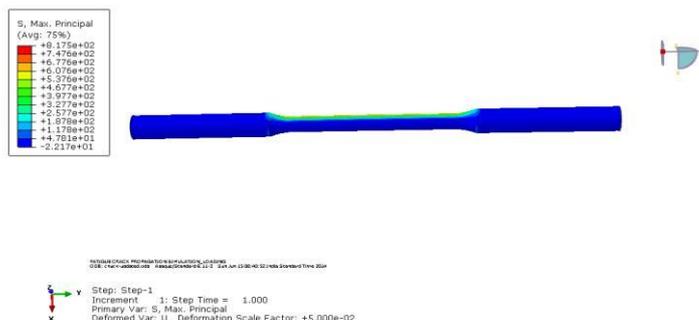


Figure 6: Crack Propagation at Step Time 3

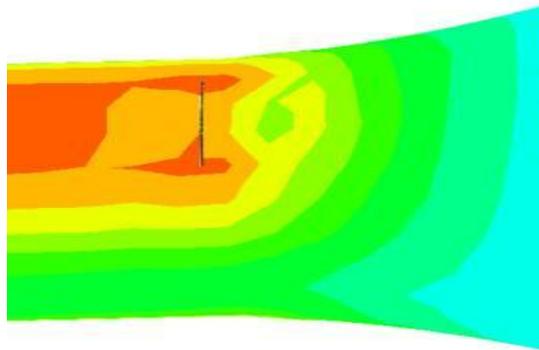


Figure 7: Crack Propagation at Step Time 4

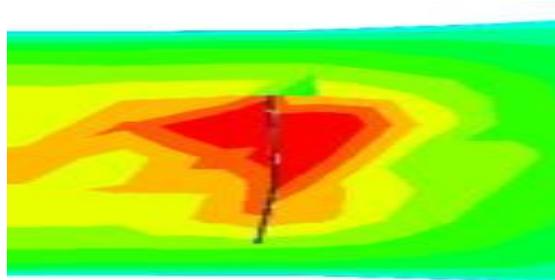


Figure 9: Crack Propagation at Step Time 6

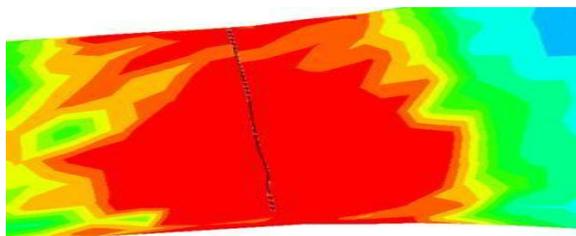
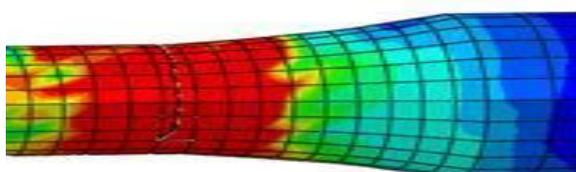


Figure 11: Final Crack Propagation



3.3 Comparison of Results

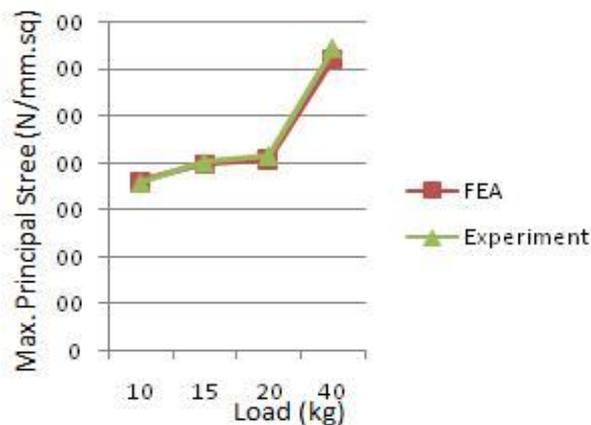
Aluminium Alloy-6062 specimen with crack producing from the edge during cyclic loading. The geometry was

imposed by plane strain conditions and edge load (σ) applied under more loading conditions. The experimental and design results are compared in table 5.

Table 5: Maximum Principal Stress between FEA & Experimental Results

S.No.	Load (kg)	Maximum Principal Stress (N/mm ²)	
		FEA	Experiment
1	10	724	717
2	15	795	804
3	20	817	832
4	40	1245	1289

Graph 3: FEA vs. Experiment Results



III. CONCLUSION

In this study the fatigue behavior characterization was carried out on Aluminium Alloy-6062 specimen using rotary bending machine test. The fatigue crack path, fatigue life and stress intensity factor along the crack length of Aluminium Alloy-6062 specimen were predicted using codes of Abaqus. The results of the developed program have been successfully validated through direct comparisons with experimental and design results. The result between experimental and design has an average error of 8%. This study is ideally suited for predicting data for fatigue life calculation in components and structures.

REFERENCES

- [1] George E. Dieter. Mechanical Metallurgy, McGraw-Hill Book Company, SI Metric Edition.
- [2] Barsoum, R. S. On the use of isoparametric finite elements in linear fracture mechanics. Int. J. for Num. Meth. in Eng., 10, 25-37(1976).
- [3] William D.Callister, Jr. Material Science and Engineering, John Wiley & Sons, Inc., 2007.
- [4] Baragetti S, Gelfi M, La Vecchia GM, Lecis N. Fatigue resistance of CrN thin films deposited by arc evaporation process on H11 tool steel and 2205 duplex stainless steel. Fatigue Fract Eng M 2005

International Journal of Innovative Research in Science, Engineering and Technology

An ISO 3297: 2007 Certified Organization,

Volume 4, Special Issue 4, April 2015

National Conference on Trends in Automotive Parts Systems and Applications (TAPSA-2015)

On 20th & 21st March

Organized by

Sri Krishna College of Engineering & Technology, Kuniathur, Coimbatore-641008, Tamilnadu, India

- [5] Chiou, Y.J., Lee, Y.M., Jowtsay, R. Mixed mode fracture propagation by manifold method. Int. J. of Fract. 114, 327–347(2002).
- [6] Guan, Z.Q., Song, C., Gu, Y.X. Recent advances of research on finite element mesh generation methods. J. of Comp. Aided Design and Comp. Graph. 15 (1), 1–14(2003).
- [7] Zienkiewicz, O., Taylor, R., Zhu, J. The finite element method: its basis and fundamental. 2005.
- [8] S. Baragetti, G.M. La Vecchia, A. Terranova, Fatigue Behavior and FEM Modelling of Thin-Coated Componentes
- [9] Erdogan, F. and G. Sih, 1963. On the crack extension in plates under plane loading and transverse shear. Journal of Basic Engineering 85: 519–27.
- [10] Paris PC, Erdogan F. A critical analysis of crack propagation laws. J Basic 1963.