FINITE ELEMENT ANALYSIS OF FATIGUE CRACK GROWTH IN ADVANCED METAL ALLOYS

N. Aruna Kumari
Research Scholar
Shri JJT University
Rajasthan

Dr. Ganga Dhar Rewar
Professor
Shri JJT University
Rajasthan

Dr. Sudhir Baijnath Ojha
Associate Professor
SSGB College of Engineering - Bhusawal

ABSTRACT

The analysis of fatigue crack growth is carried out within the framework of the classical linear elastic fracture mechanics. The preprocessing and the finite element analysis of the cracked structure are done using the commercial software system Abaqus. Elastic-perfectly plastic constituents, each with random yield strength, modeled by a normal probability density function are used. Complex constituent relations for material behavior under monotonically increasing load or cyclic loading are replaced by a probabilistic approach in the developed finite element program. The software system ProCrack, a Python-based tool for the automated finite element simulation of fatigue crack growth in arbitrarily loaded three-dimensional components is presented. The Bauschinger effect is found to be related to the mean and the hardening coefficient to the variance of the distribution. The decomposed global stiffness matrix is updated using an efficient technique instead of reformulating it after each boundary change. The applied strains are so important that diffuse damage phenomena are visible as a network of micro-cracks in front of the major crack. A sub-modelling technique in combination with the interaction integral techniqueis used to calculate the range of the stress intensity factors. Fatigue crack growth behavior in weld nugget zone (WNZ) of friction stir welded (FSWed) dissimilar aluminum alloys joints were investigated comparing to that of the FSWed similar aluminum alloy joints and the base materials.

Keywords: *Fatigue crack growth, weld nugget zone (WNZ), aluminum alloys, cracked structure, plastic constituents, micro-cracks, materials.*

INTRODUCTION

The extended finite element method (XFEM) is an alternate way to predict the SIFs using computational methods. In general, the initiation and propagation of cracks must be associated to the SIFs in a complicated state. The extended finite element method proposed has been widely used in recent studies. It is based on the standard finite element framework and uses a special displacement feature to allow discontinuities to occur, overcoming the need to re-mesh continuously throughout the crack tip expansion process. To evaluate the SIFs, XFEM was used to perform crack growth analysis without updating the mesh. Extensive work was undertaken to develop efficient models to evaluate the fatigue crack growth (FCG) and fatigue life in order to overcome fatigue failures. The phenomena includes cyclic hardening, ratcheting, mean stress relaxation, Bauschinger effect, loading history, mean stress dependant hardening, strain range and non-proportional hardening. There are several proposed experimental models, but they become prohibitive both in terms of cost and time. Most components are subjected to fatigue loading. Different strategies were proposed by researchers to estimate the plastic zone dimension and formation at crack tip. A critical literature review was performed to study the previous research. Paul has states that the character of cyclic plastic deformation and harm on a fatigue crack tip is expressible. In cyclic plastic zone, low cycle fatigue is the dominant cyclic deformation mode during symmetric load cycling. It is done by the thorough analysis of plastic core region under different loading conditions. The crack tip for orthotropic and isotropic materials for small scale yielding on different loading conditions was determined.

LITERATURE REVIEW

Durga Prasanth Dude (2023) Many structural components and devices in combustion and automotive engineering undergo highly intensive cyclic thermal and mechanical loading during their operation, which leads to low cycle (LCF) or thermomechanical (TMF) fatigue crack growth. This behavior is often characterized by large scale plastic deformations and creep around the crack, so that concepts of linear-elastic fracture mechanics fail. The finite element software PROCRACK PLAST has been developed at TU Bergakademie Freiberg for the automated simulation of fatigue crack growth in arbitrarily loaded three-dimensional components with large scale plastic deformations, in particular under cyclic thermomechanical loading. PROCRACKPLAST consists of a bundle of Python routines, which manage finite element pre-processing, crack analysis, and post-processing in combination with the commercial software ABAQUS.

Yahya Ali Fageehi (2021) This study presents computational modeling of a crack growth path under mixed-mode loadings in linear elastic materials and investigates the influence of a hole on both fatigue crack propagation and fatigue life when subjected to constant amplitude loading conditions. Though the crack propagation is inevitable, the simulation specified the crack propagation path such that the critical structure domain was not exceeded. ANSYS Mechanical APDL 19.2 was introduced with the aid of a new feature in ANSYS: Smart Crack growth technology. It predicts the propagation direction and subsequent fatigue life for structural components using the extended finite element method (XFEM). The Paris law model was used to evaluate the mixed-mode fatigue life for both a modified four-point bending beam and a cracked plate with three holes under the linear elastic fracture mechanics (LEFM) assumption.

Rajesh P. Verma (2021) The AA6061-T6 aluminium alloys are hugely applicable, where strength of material is predominant. Majority of the failure of such type of structure is due to fatigue. The observation of fatigue crack enhancement is very useful to predict the remaining existence of the shape and therefore, to understand the fatigue failure. It helps to save time and material that is required in experimentation to evaluate crack growth rate. Many researchers investigated the dependency of fatigue crack enhancement on plastic area size; however there may be restricted literature to be had on plastic area dedication of AA6061-T6 aluminium alloys. In this research study the fatigue crack enhancement rate of AA6061-T6 aluminium alloys experimentally and plastic zone is determined at crack tip when crack advances using ANSYS.

Diogo M. Neto (2020) Fatigue crack growth (FCG) has been studied for decades; however, several aspects are still objects of controversy. The objective here is to discuss different issues, using a numerical approach based on crack tip plastic strain, assuming that FCG is driven by crack tip deformation. ΔK was found to control cyclic plastic deformation at the crack tip, while K_{max} has no effect. Therefore, alternative mechanisms are required to justify models based on ΔK and K_{max}. Below crack closure, damage occurs during unloading while during loading the crack tip deformation is elastic. However, if the maximum load is decreased below the elastic limit, which corresponds to the transition between elastic and elasto–plastic regimes, there is no crack tip damage. Finally, the analysis of FCG after an overload with and without contact of crack flanks showed that the typical variation of da/dN observed is linked to crack closure variations, while the residual stresses ahead of crack tip are not affected by the contact of crack flanks.

Rina Sakai (2018) Patients suffering from severe degeneration may find that a posterior stabilized (PS) knee prosthesis, equipped with a post cam mechanism, aids in achieving intrinsic stability. In many retrieval experiments, the polyethylene tibia post was shown to be completely ineffective. We believe that catastrophic failures were mostly caused by the large burdens that people had to cope with on a daily basis. The purpose of this work was to provide a specification for the tibia posts of posterior-stabilized knee prosthesis inserts by investigating the mechanical forces produced by these posts using finite element analysis and a compression test. Following impingement, the tibias in the front and back were mimicked. Results from the FE analysis and the compression test showed that the surface pressure levels were the same.

Fracture toughness characterization of metal sheets

There are a number of well-established fracture mechanics testing methods that may be used to characterize the toughness of metal fractures. Standard thin sheets used in the automotive industry, usually with a thickness of 1-4 mm, do not meet the requirements of ASTM E1820. In spite of all the benefits they provide, industry standards for screening materials do not yet include fracture toughness assessments. One non-standard approach to determining fracture toughness for thin metal sheets is the Essential Work of Fracture (EWF) technique. The process is made easier. When describing the fracture resistance of sheet materials, engineers employ the EWF method. Alloys of several materials fall into this category. Among them are plastic, aluminum, steel, and countless more. High strength aluminum alloys and thin AHSS sheets may now have their fracture toughness evaluated correctly because to Eureka's meticulous improvements to this approach.

Finite element analysis software

Finite element analysis (FEA) is a computerized method for predicting how a product reacts to real-world forces, vibration, heat, fluid flow and other physical effects. Finite element analysis shows whether a product will break, wear out or work the way it was designed. It is called analysis, but in the product development process, it is used to predict what's going to happen when the product is used. FEA works by breaking down a real object into a large number (thousands to hundreds of thousands) of finite elements, such as little cubes. Mathematical equations help predict the behaviour of each element. A computer then adds up all the individual behaviours to predict the behaviour of the actual object.

Finite Element Analysis

Finite Element Analysis (FEA) is a computerised technique that predicts how products and materials will react to forces in the real world, alongside other physical effects like vibrations, heat, and fluid flow. This is important as it demonstrates whether a product will work or fail. If a product won't hold up to the conditions in which it will need to function, then it needs to be reviewed – the advantage of using finite element analysis is that you won't already have produced the product at this stage.

Importance of Finite Element Analysis

Time and money are precious commodities to any business, and the advantages of the finite element analysis method mean that time, effort and costs are saved. In addition, improved efficiency and accuracy prevent expensive mistakes; without using this method, there is the risk that products will not perform as they are intended. For other guidance on using the right products for your application, have a look at our previous blog post on selecting the right rubber compound and step-by-step guide to choosing anti-vibration mounts.

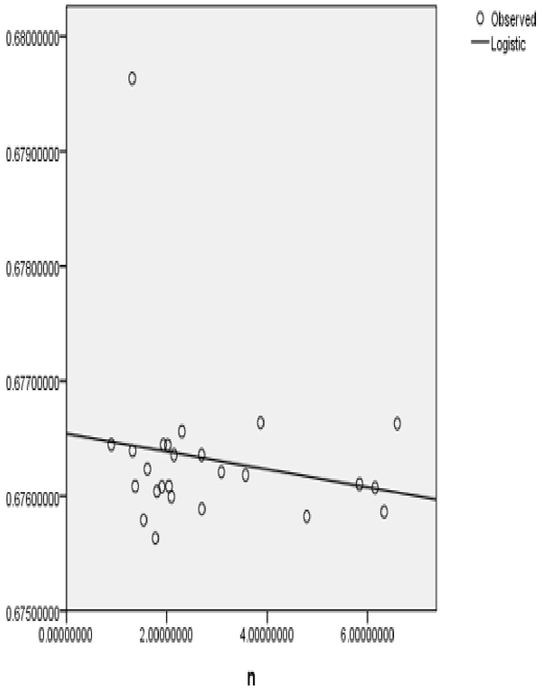
RESEARCH METHODOLOGY

The specimens used for this research were made of aluminum alloy 7075-T76. Aluminum is a lightweight material that is very attractive for airplanes because it is so strong and it can be strengthened by additional treatment. Other attractive characteristics are high-quality corrosion and stress resistance and high strength-to-density ratio coupled with other fundamental properties necessary for applications in aircraft construction. Although, the strength-to weight ratio is high in aluminum. Aluminum is a material that can be fabricated into a diverse range of shapes. This is why it is common practice to evaluate displacements by either using an "effective" plastic load vector or by modifying the stiffness matrix coefficient after each load increase. Learn how elastic plastic materials react to in-situ stress and how the FEM equations related to incremental plasticity work in this section. Prior to plastic yielding, any investigation into elastic plasticity may make use of the finite element equations pertaining to elasticity. The material so popular with the aerospace industry because of high strength and durable characteristics like moderate toughness that are necessary under the harsh conditions airplanes and other aerospace products need to withstand. The basic composition of aluminum alloy 7075 is aluminum, zinc, magnesium and copper plus chromium. The radius of curvature of the arm junction measured 28 mm. A hole was drilled into the center of the specimens that were also notched by an electro-discharge method.

RESULTS AND DISCUSSION

It was necessary to do the regression analysis after the FEM analysis. A trend line was generated in logarithmic mode, and the result graph was shown between U and n. All the essential facts are provided in the following diagram and analysis of the curves. Testing the 6063 Al Alloy in an application setting with the generalized model yielded the findings. This concept, put out is supported by the results, which demonstrate a linear dependence between Keff and UŔK. The concept of a universal Paris law was thus established.

3003 Al Alloy for ΔP= constant



Graph 1: Assuming ΔP remains unchanged, the 3003 Al regression results

Alloy 5052 Al with ΔP set to constant

Model Summary

R	R Square	attuned R Square	Std. Error of the Estimate
.485	.235	.203	.000

The sovereign changeable is n.

ANOVA

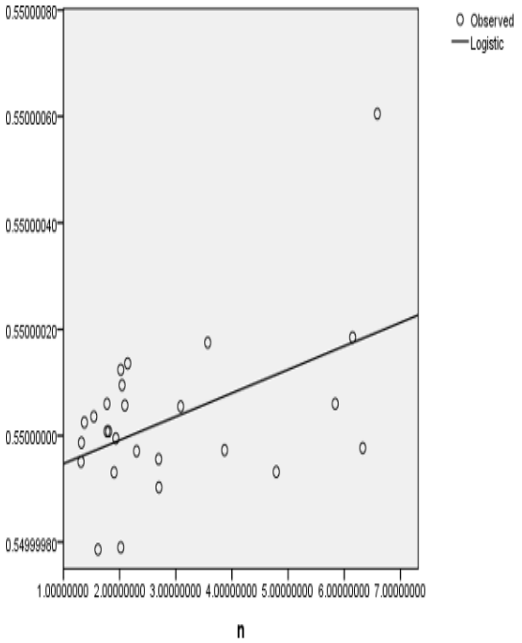
	Sum of Squares	df	Mean Square	F	Sig.
Regression	.000	1	.000	7.382	
Residual	.000	24	.000		
Total	.000	25			

The auto nomouspatchy is n.

Coefficients

	Unstandardized Coefficients		Standardized Coefficients	t	Sig.
	B	Std. Error	Beta		
N	1.000	.000	.616	3.381E7	.000
(Constant)	1.818	.000		1.023E7	.000

The needy variable is ln (1 / U).



Graph 2: Findings from the Regression Analysis of 5052 Al Alloy with ΔP=Konstanz

Table 1: Summary of the Model for Regression Analysis on 6351 Al Alloy with ΔP=Constant

R	R Square	Adjusted R Square	Std. Error of the Estimate
.055	.003	-.022	.007

Table 2: ANOVA

	Sum of Squares	Df	Mean Square	F	Sig.
Regression	.000	1	.000	.122	.729
Residual	.002	40	.000		
Total	.002	41			

CONCLUSION

The integration of FEA with sensor technology and real-time data processing will pave the way for predictive maintenance procedures. Because of this, structural health may be monitored continuously and crack advancement can be predicted based on in-service conditions. Two possible implications of quantum computing potential to revolutionize finite element analysis (FEA) are enhanced model fidelity and faster fracture stability analysis in metal alloys. The displacement extrapolation method uses to evaluate the SIFs, and the maximum circumferential stress theory uses to calculate the crack growth angles. The crack propagation of two case studies is simulated with the help of the developed program, which uses an adaptive finite element mesh generation approach. The predicted values of the stress intensity factors agreed closely with the available numerical results. During the crack propagation, a particular criterion of the Crack Growth increment utilizes the magnitude of the crack increment. Finite element analysis (FEA) is a powerful and efficient tool for analyzing failures. By using finite element analysis, a failure analyst may get a better understanding of the failure's causes from a quantitative and qualitative standpoint. If the analyst doing the finite element analysis (FEA) is well-versed in the system and has sufficient data at their disposal, the results may be believed. However, it is essential to constantly verify FEA findings using experimental or real-world data.

REFERENCE

1. *Yahya Ali Fageehi (2021), "Fatigue Crack Growth Analysis with Extended Finite Element for 3D Linear Elastic Material",Metals,ISSNno:2075-4701, Vol.11(3),Pages.397.https://doi.org/10.33 90/met11030397*
2. *Rajesh P. Verma (2021), "FEA based fatigue crack growth analysis",Materials Today: Proceedings,ISSNno:2214-7853,Vol.46,Pages.10575-10581.https://doi .org/10.1016/j.matpr.2021.01.319*
3. *Durga Prasanth Dude (2023), "ProCrackPlast: a finite element tool to simulate 3D fatigue crack growth under large plastic deformations", International Journal of Fracture,ISSNno:1573-2673,Vol.243,Pages.65-90.https://doi.org/10 .1007/s10704-023-00732-9*
4. *Fernando Ventura Antunes (2023), "Fatigue Crack Growth in Metallic Materials",Materials (Basel),ISSNno:1996-1944,Vol.16(1),Pages.11.doi:10.33 90/ma16010011*
5. *Diogo M. Neto (2020), "Revisiting Classical Issues of Fatigue Crack Growth Using a Non-Linear Approach",Materials,ISSNno:1573-4838,Vol.13,Pages.55 44.doi:10.3390/ma13235544*

6. *Rodolfo Lauro Weinert (2021),"An Introductory Note on Finite Element Problems Based on the Eddy Current Testing Approach", Journal of Electro-magnetic Analysis and Applications, ISSNno:1942-0749,Vol.13, No.11, Pages.145-159.*
7. *Ruining Wei (2018),"Finite Element Simulation Analysis of Compressed Material in Intelligent Compaction", Engineering, ISSNno:1947-394X,Vol.10, No.4,Pages.173-186.*
8. *Rutooj Deshpande (2011),"Liquid Metal Alloys as Self-Healing Negative Electrodes for Lithium Ion Batteries", Journal of The Electrochemical Society, ISSNno:1945-7111,Vol.158(8),Pages.A845-A849.DOI:10.1149/1.3591094.*
9. *Rahul N. Yerrawar (2012),"Finite Element Analysis of Dynamic Damper for CV Joint",Energy and Power Engineering,ISSNno:1947-3818,Vol.4,No.4, Pages.241-247.*
10. *Rina Sakai (2019),"Compression Test and Finite Element Analysis for Failure Criterion of the Tibial Post of a Posterior-Stabilized Knee Prosthesis",Journal of Biomedical Science and Engineering,ISSNno:1937-688X,Vol.12 No.5, Pages.*