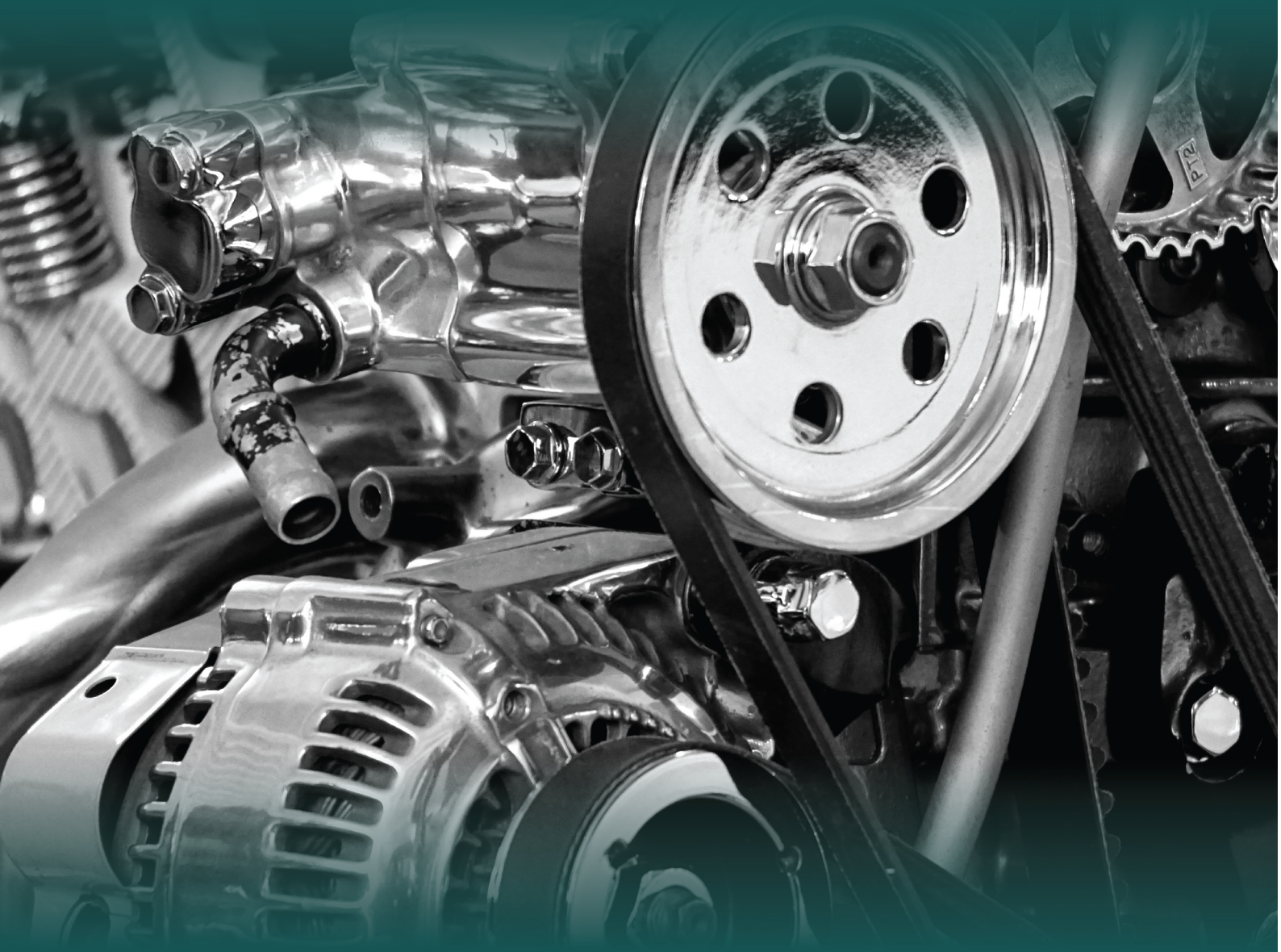


RECENT TRENDS IN MECHANICAL AND AUTOMOTIVE ENGINEERING TECHNOLOGY

**THE 6TH SYMPOSIUM ON DAMAGE
MECHANISM IN MATERIALS AND STRUCTURES
(SDMMS 2022)**



**NASRUL AZUAN ALANG
AIZAT ALIAS
SHAMIL SHAARI
SALVINDER SINGH KARAM SINGH**

RECENT TRENDS IN MECHANICAL AND AUTOMOTIVE ENGINEERING TECHNOLOGY

**THE 6TH SYMPOSIUM ON DAMAGE
MECHANISM IN MATERIALS AND STRUCTURES
(SDMMS 2022)**

RECENT TRENDS IN MECHANICAL AND AUTOMOTIVE ENGINEERING TECHNOLOGY

**THE 6TH SYMPOSIUM ON DAMAGE
MECHANISM IN MATERIALS AND STRUCTURES
(SDMMS 2022)**

**NASRUL AZUAN ALANG
AIZAT ALIAS
SHAMIL SHAARI
SALVINDER SINGH KARAM SINGH**

Penerbit
Universiti Malaysia Pahang Al-Sultan Abdullah,
Kuantan
2026

All right reserved.

Apart from fair dealing for the purpose of study, research, criticism or review, as permitted under the Copyright Act, no part of this book may be reproduced, stored on a retrieval system, or transmitted, in any form or by any means, electronic, mechanical, photocopying, recording or otherwise, without the prior written permission from Penerbit Universiti Malaysia Pahang Al-Sultan Abdullah, Lebu Persiaran Tun Khalil Yaakob, 26300 Kuantan, Pahang Darul Makmur.



Cataloguing-in-Publication Data

Perpustakaan Negara Malaysia

A catalogue record for this book is available
from the National Library of Malaysia

EISBN 978-629-7917-18-4

Director Penerbit : Mr. R
Chief Editor : M. Azli
Editor : Adli
Proofreader : Adli
Graphic Designer : R. W. Chamie
Administration : Zue & N. A. Aryan
Sales & Marketing : N. H.

Share | Like | Tag
Online Shop: <https://msha.ke/penerbitump>
Official Page (FB): Penerbit UMPSA
Official IG: Penerbitumpsa

Published By
Penerbit
Universiti Malaysia Pahang Al-Sultan Abdullah,
Lebu Persiaran Tun Khalil Yaakob,
26300 Kuantan, Pahang Darul Makmur.
Email: penerbit@umpsa.edu.my

Printing
PNC Printing
No.2, Tingkat Bawah, Taman Damai Indah,
Peramu, 26600 Pekan, Pahang Darul Makmur
Tel: 09-425 2010

Recent Trends in Mechanical and Automotive Engineering Technology (RTMAET) swiftly and informally disseminates the latest developments and technology in Mechanical and Automotive. The core of RTMAET comprises original research presented in proceedings and colloquiums. Published volumes in RTMAET comprehensively cover all aspects, subfields, and emerging challenges within mechanical and automotive engineering technology.

Topics in the series include:

- Fatigue Damage
- Fatigue Crack Initiation & Propagation
- Life Prediction Techniques
- Dynamic Fracture
- Damage Mechanics & Assessment
- Non-Destructive Test (NDT)
- Concrete Failure Assessment
- Failure on soil structures
- Structural Durability & Reliability
- Structural Health Monitoring
- Construction Damage Recovery
- Mechanical Structures and Stress Analysis
- Automotive Engineering
- Theoretical and Applied Mechanics
- Manufacturing
- Materials Engineering

For further information about RTMAET, please contact the:

Editor In-Chief

Dr. Shamil Shaari at shamil@umpssa.edu.my

Faculty of Mechanical and Automotive Engineering Technology

Universiti Malaysia Pahang Al-Sultan Abdullah, Pekan, Pahang, Malaysia

Co-Editor In-Chief

Associate Professor Ir. Dr. Mohd Hafizi Zohari at hafizi@umpssa.edu.my

Faculty of Mechanical and Automotive Engineering Technology

Universiti Malaysia Pahang Al-Sultan Abdullah, Pekan, Pahang, Malaysia

**Proceedings of
The 6th Symposium on Damage Mechanism in
Materials and Structures
(SDMMS 2022)**

**16th – 17th August 2022
Pahang, Malaysia**

PREFACE

The 6th Symposium on Damage Mechanism in Materials and Structures (SDMMS2022) is organised by the Faculty of Mechanical and Automotive Engineering Technology (FTKMA), Universiti Malaysia Pahang Al-Sultan Abdullah and Department of Mechanical and Manufacturing Engineering, Faculty of Engineering and Built Environment, Universiti Kebangsaan Malaysia. The conference also is co-organised by Ningxia University, China. The conference is held on 16-17th August 2022 at Swiss-Garden Resort, Kuantan, Pahang. This conference is the continuity from the success previous events which was held virtually on March 2021, August 2019, December 2018 and August 2017. The proceeding book of this conference comprises a collection of 37 articles, which covers the fundamental topics and techniques in SDMMS2022, such as durability, fatigue, creep, fracture mechanics, failure analysis, damage mechanism and assessment, material and structural behaviour, and various topics in structural integrity assessment.

This specific knowledge-sharing session successfully gathers researchers from various universities and industries locally and overseas to meet and discuss the issues related to damage mechanism in material and structures. Hence, it provides the platform and opportunity for researchers to meet and discuss their most recent breakthrough research.

Finally, we would like to express our gratitude to all participants and keynote speakers for taking part in this conference. We are indebted the organizing committee for their continuous efforts to make this conference a success. Also, without the outstanding support from Universiti Malaysia Pahang Al-Sultan Abdullah and Universiti Kebangsaan Malaysia, this conference would not have been possible.

Thank you and do enjoy the valuable knowledge sharing session.

Professor Ts. Dr. Ruzaimi Mat Rejab
Symposium Chair SDMMS2022

CONTENT

	Page
Preface	ix
P01 Cycle Number Estimation Method on Fatigue Crack Initiation using Voronoi Tessellation and the Tanaka Mura Model <i>M. R. A. Rahim, S. Schmauder, Y. H. Manurung, A. Klenk, M. I. M. Ahmad, and K. Dogahe</i>	1
P02 Enhancement on Hemispherical Dimple Core Design using the Fuzzy Triangular Hybrid Decision-Making Method for Damage Behaviour <i>M. K. Faidzi, S. Abdullah, S. S. K. Singh, M. F. Abdullah and A. H. Azman</i>	5
P03 Parametric Studies on Compressive Characteristics of the Spherical-roof Contoured Core (SRCC): A Numerical Study <i>Q. Ma, M. R. M. Rejab, S. A. Hassan, M. Azeem, and Saffirna M.S.</i>	9
P04 Approaches of Finite Element Analysis to Stress Analysis of Rail-Wheel Contact Stress for Light Rail Transit <i>M. N. Tawfik, M.M Padzi, H. Hapaz, and M. Zahar</i>	13
P05 Impact Behaviour of Laminated Bamboo, E-Glass and Hybrid Bamboo/E-Glass Fibre Composites <i>A. Suhaimi, A. Ali, K. Rassiah, and K. Z. K. Ahmad</i>	17
P06 Modelling of Creep Damage Evolution of 316H Stainless Steel using C-Shaped Specimen <i>I. U. Ferdous, N. A. Alang, J. Alias, N. Ab. Razak and N. A. Razak</i>	21
P07 Influence of Crack in Composite Plate on Natural Frequency using Fibre Bragg Grating Sensors Array <i>Z. M. Hafizi and E. Vorathin</i>	25
P08 Modeling the Mechanical Behaviour of PolyJet 3D-Printed Resin Sand Material <i>Z. Bo, L. Shaoying, M. R. M. Rejab, L. Jiayuan, and L. Yushi</i>	29
P09 <i>Numerical Simulation of P91 Steel Under Low Cycle Fatigue Loading</i> <i>M. A. A. Roslin, N. A. Razak, N. A. Alang, and N. Sazali</i>	33
P10 Comparison Creep Life Prediction of P91 Steel by Using Omega Method and Larson Miller Parameter Method <i>S.N.A. Rosli, N. A. Razak, and N. A. Alang</i>	37
P11 Efficient Neighbour Search Algorithm in Nonlocal Fracture Simulation <i>S. Kamaludin and P. Thamburaja</i>	41
P12 Impact of Cyclic Operation to Leaked Pipeline Reinforced with Composite Wrapping <i>Z. Yusof</i>	45
P13 Frequency-driven Crack Propagation in Ultrasonically-assisted Bone Cutting <i>D. Qianriui, S. S. Kamaludin, and N. Yusoff</i>	49

P14	Analysis of Stress Intensity Factor via Finite Element Model <i>M. N. M. Husnain, M. R. M. Akramin, Z. L. Chuan, M. S. Shaari, and A. Takahashi</i>	53
P15	$J - \Delta\sigma$ Quantification of Out-of-plane Constraint Loss in Mode-I Fracture <i>N. Yusoff and F. Yusof</i>	57
P16	Von-Mises stress analysis on the faceplates of laser-welded web-core steel sandwich plates <i>J. Xiaoxia, G. Ningnian, Saffirna M. S., Y. Gang, and W. Yaowei</i>	61
P17	A new flat inspection vehicle for detecting belt damage <i>A. Li, B. Zhang, and H. Hao</i>	65
P18	The Effectiveness of the Equal Load Carrying Capacity Joints Design Method Based on the Ideal Elastic-plastic Hypothesis <i>J. Guo, Z. Dong and H. Fang</i>	69
P19	Research on failure and repair of hydraulic support column seal <i>S. Mu, K. An, and Y. M. Qiao</i>	73
P20	Surface Enhanced Raman Scattering Effect of AgNPs/SiO ₂ Prepared by Rapid Thermal Annealing Method <i>Z. Jia, Y. Wang, and Y. Xu</i>	77
P21	Design and simulation analysis of bolt group connection of BS-type flange cast steel right-angle sea valve <i>Z. Wang, J. Yu, Y. Ma and A. Qu</i>	81
P22	Effect of 3.5% NaCl solution on the mechanical properties of 7075-aluminium alloy with RRA regression re-ageing <i>Z. Pei, W. Guan, K. Linyuan, W. Yan, Z. Zekun, and S. Xin</i>	85
P23	Numerical study of random corrosion characteristics of metal surfaces based on the Cellular automata method <i>Z. Zekun, W. Guan, K. Linyuan, Y. Qingrui, Z. Pei, and X. Yiquan</i>	89
P24	Research on localised corrosion and mechanical properties of high-strength aluminium alloys <i>W. Yan, W. Guan, K. Linyuan, Z. Pei, W. Wenhui, Z. Xuejun, and L. Zhiwen</i>	93
P25	Failure analysis of ear socket of gangue retaining bracket for coal mine <i>S. Mu, Y. Qiao, K. An, and J. Wu</i>	97
P26	Tension-tension Fatigue Behaviour Analysis of Glass Fiber Reinforced Polymer Composite <i>M. Loman, M. H. Zohari, and F. Lamin</i>	101
P27	Mechanical Analysis and Constitutive Modeling of Non-linear Behaviour of Ag-based Conductive Ink <i>S. Zulfiqar, A. A. Saad, Z. Ahmad, and Z. Bachok</i>	105
P28	Fatigue Crack Growth Behaviour of AlSi10Mg Material Produced by Direct Metal Laser Sintering Using S-version Finite Element Method <i>B. F. Yung, S. D. Urai, M. S. Shaari, A. Takahashi and M. A. M. Romlay</i>	109

P29	Effects on Hybridization of Interlayer Composites and Self-Reinforced Polypropylene <i>M. I. Ibrahim, M. R. M. Rejab, M. Quanjin, and M. F. Rani</i>	113
P30	Mechanical Characterisation of 3/2 Fibre Metal Laminate Materials <i>M. F. Rani, M. R. M. Rejab, M. I. Ibrahim, and N. K. Romli</i>	117
P31	Fatigue Characterisation for Effective Strain Damage Model on Various Road Load Condition <i>L. Abdullah, S. S. K. Singh, S. Abdullah, A. H. Azman, and A.K. Ariffin</i>	121
P32	Damage Assessment of Stiffened Steel Plates Subjected to Blast Loads Using Finite Element Analysis <i>N. S. A. Razak and A. Alias</i>	125
P33	Numerical Simulation of Temperature Field in Ultra-Narrow Arc Welding of Thin-Walled Steam Turbine Valve Body Material <i>Gouningnian, Shunshuo, Jiangxiaoxia, Yuanyongguang, Wangbangle and Hanxiaotong</i>	129
P34	Failure Analysis of Radiant Tube in Ethylene Furnace <i>M. H. Mohammad</i>	133
P35	Study on Chemical State Model of Reservice Chemical Equipment <i>Z. Nan, D. Wenjie, W. Qiangjing, S. Xinglong and L. Tongman</i>	137
P36	Investigation and Analysis of Pipe Support Weld Cracks <i>M.Z. Kamardin, M.A. Ton Alias, and A.H. Mohamad</i>	143
P37	Combatting Repetitive Failures at Sour Gas Concentration Unit (SGCU) Overhead System <i>M. H. Hasan, R. Jaafar and N. Ali</i>	147
	List of Editors	151

P01

Cycle Number Estimation Method on Fatigue Crack Initiation using Voronoi Tessellation and the Tanaka Mura Model

M. R. A. Rahim^{1,4}, S. Schmauder¹, Y. H. Manurung², A. Klenk¹, M. I. M. Ahmad³, and K. Dogahe¹

¹*Institute for Materials Testing, Materials Science and Strength of Materials (IMWF), University of Stuttgart, Pfaffenwaldring 32, 70569 Stuttgart, Germany.*

²*SMRI and School of Mechanical Engineering, Universiti Teknologi MARA (UiTM), 40450 Shah Alam, Selangor, Malaysia.*

³*Faculty of Engineering and Built Environment, 43600 Universiti Kebangsaan Malaysia (UKM), Selangor, Malaysia.*

⁴*Department of Occupational Safety and Health Sarawak, Level 17, No.9, Jalan Bukit Mata, 93100 Kuching, Sarawak Malaysia*

Tel: +4917641550355 Email: ^aridzwan.rahim@imwf.uni-stuttgart.de,

^bsiegfried.schmauder@imwf.uni-stuttgart.de, ^cyupiter.manurung@uitm.edu.my

ABSTRACT

This paper deals with the short crack initiation of the material P91 under cyclic loading at different temperatures concluded with the estimation of the short crack initiation Wöhler (S/N) Curve. An artificial but representative microstructural model microstructure was generated using the Finite Element Method and the non-uniform stress distribution is calculated accordingly afterwards. The number of cycles needed for crack initiation is estimated on the basis of the stress distribution in the Voronoi tessellation model by applying the physically-based Tanaka-Mura model. Initial results show the number of cycles to generate crack initiation is correlated with temperature.

Keywords: Short Crack, P91, Wöhler (S/N) Curve, Voronoi Tessellation, Tanaka-Mura Model

INTRODUCTION

Power plants have been playing an essential role in generating huge electricity or processing raw material to become a product. Typically the design life for power plants is about 100,000 hours (Nikbin 2013; Ab Razak et al. 2016). The machinery operated at elevated temperatures with some vibration caused by its surrounding is exposed to low cycle fatigue stresses. On the basis of thermodynamics, raising the operating temperature of a power plant is a primary technique to increase its energy efficiency. However, high operating temperatures require outstanding structural integrity.

In assessing the structural integrity of high-temperature power-plant components, it is essential to understand the crack initiation under fatigue, creep and creep-fatigue interaction loadings (Speicher et al. 2013). High-temperature crack initiation and growth behaviour of a grade 91 steel under constant load have been investigated by many researchers (Bassi et al. 2017; Narasimhachary & Saxena 2013).

Research into fatigue mechanisms for high-temperature components under multi-axial stress has been ongoing for several decades, and an appropriate engineering design criterion has emerged. Fatigue describes the situation where the accumulated rate of damage under complex loading differs from the linear summation of the rate of damages produced by cyclic loading (Zhang et al. 2014; Hales 1980). Forward and backward loadings are essential in causing fatigue to occur. The accumulation of dislocations within the material occurs throughout a large number of experienced cycles (Mlikota 2020; Sangid 2013; Polák and Man 2013).

In this study, the cracking initiation stage is quantitatively described using a dislocation model of a double pileup on a single slip band given by Tanaka and Mura (Tanaka and Mura, 1981; Tanaka and Mura, 1982).

Based on the reviewed literatures, it can be revealed that not many researchers investigated the short crack initiation of fatigue with emphasis on the material P91 for a component. Due to these facts, this research was carried out to examine the number of cycles for short crack fatigue initiation using Abaqus with python language scripts by creating an artificial representative microstructure model by Voronoi Tessellation method for various temperatures. As points of conclusion and further recommendation, results from the simulation will be analysed and discussed.

METHODOLOGY

The Tanaka-Mura method can be used to figure out how many cycles are required for crack initiation and to start crack nucleation in a single grain.

RESULTS AND DISCUSSION

Figure 1.1 and Figure 1.2 show the short crack initiation Wöhler curve corresponding to an artificial representative microstructure selected for the numerical analysis of the fatigue crack initiation lifetime. The number of cycles of each point from TMM is nearly similar compared to experimental as illustrated in Figure 1.1. While at elevated temperatures, in Figure 1.2, P91 gives a smaller number of cycles and a massively decreased stress amplitude compared to room temperature conditions due to the tensile strength of material reduced proportionally towards temperature.

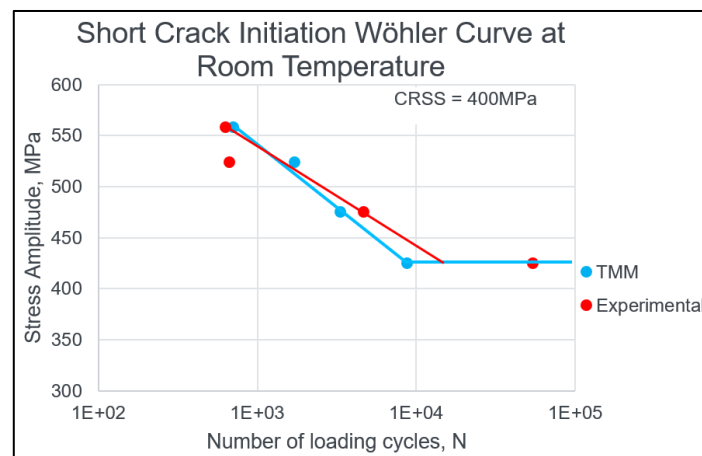


Figure 1.1: Short crack initiation Wöhler curve at room temperature

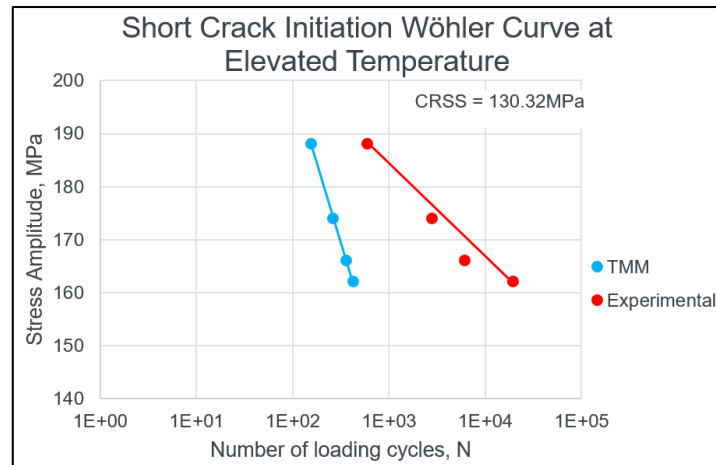


Figure 1.2: Short crack initiation Wöhler curve at 650°C temperatures

CONCLUSION

The Tanaka-Mura model provides a quantitative relation between the slip length, the resolved shear stress along the slip line and the number of cycles to crack initiation. Based on the achieved results, the analysis leads to the following conclusions:

- The process of fatigue crack initiation can be described very well by the Tanaka-Mura model.
- The crack initiation is dependent on the operation temperature as demonstrated in Figure 1.1 and Figure 1.2.
- The Tanaka-Mura model was applied to study the fatigue crack nucleation
- In many cases, Critical Resolved Shear Stress (CRSS) shows an important parameter in giving the highest impact on the fatigue loading and cycle numbers.

REFERENCES

- Ab Razak, N., Davies, C. M. and Nikbin, K. M. (2016) 'Creep-fatigue crack growth behaviour of P91 steels', *Procedia Structural Integrity*, 2, pp. 855–862.
- Hales, R. (1980) 'A quantitative metallographic assessment of structural degradation of type 316 stainless steel during creep-fatigue', *Fatigue of Engineering Materials and Structures*, 3(4), pp. 339–356.
- Mlikota, M. (2020) *Multiscale Modelling and Simulation of Metal Fatigue and its Applications*. Universität Stuttgart.
- Nikbin, N. M. A. M. K. M. (2013) 'Creep crack growth of very long term P91 steel using extrapolated short-term uniaxial creep data', in *Proceedings of the ASME 2013 Pressure Vessels and Piping Conference PVP2013*, pp. 1–6.
- Polák, J. and Man, J. (2013) 'Fatigue crack initiation – The role of point defects', *International Journal of Fatigue*.
- Sangid, M. D. (2013) 'The physics of fatigue crack initiation', *International Journal of Fatigue*, 57, pp. 58–72.
- Speicher, M., Klenk, A. and Coleman, K. (2013) 'Creep-Fatigue Interactions in P91 Steel', in *13th International Conference on Fracture*, pp. 1–10.
- Tanaka, K. and Mura, T. (1981) 'A Dislocation Model for Fatigue Crack Initiation', 48(81), pp. 97–103.
- Tanaka, K. and Mura, T. (1982) 'A theory of fatigue crack initiation at inclusions', *Metallurgical Transactions A*, 13A, pp. 117–123.
- Zhang, X., Tu, S. and Xuan, F. (2014) 'Creep – fatigue endurance of 304 stainless steels', pp. 1–16.

ACKNOWLEDGEMENT

The authors would like to express their gratitude to staff members of the Institute for Materials Testing, Materials Science and Strength of Materials (IMWF) at the University of Stuttgart, Smart Manufacturing Research Institute (SMRI) at School of Mechanical Engineering, Universiti Teknologi Mara as well as Department of Occupational Safety and Health, Malaysia (DOSH). This research is financially supported by Public Service Department of Malaysia with reference number JPA(I)870614145415.

P02

Enhancement on Hemispherical Dimple Core Design using the Fuzzy Triangular Hybrid Decision-Making Method for Damage Behaviour

M. K. Faidzi¹, S. Abdullah¹, S. S. K. Singh¹, M. F. Abdullah², and A. H. Azman¹

¹Department of Mechanical and Manufacturing Engineering, Faculty of Engineering and Built Environment, Universiti Kebangsaan Malaysia, 43600 UKM Bangi, Selangor, Malaysia

²Department of Mechanical Engineering, Faculty of Engineering, Universiti Pertahanan Nasional Malaysia, Kem Perdana Sg. Besi, 57000 W.P Kuala Lumpur, Malaysia

Email: shahrum@ukm.edu.my

ABSTRACT

This paper presents a sandwich panel enhancement by determining the optimum hemispherical dimple core design using the hybrid decision-making method based on the fuzzy triangular elements. The damage occurrence to a sandwich panel is dominated by delamination between the bonding layers when subjected constant cyclic loading. Thus, it is vital to improve the core surface area to increase the delamination resistance. The output of a finite element analysis was analysed using the Analytical Hierarchical Process and the Technique for the Order of Preference by Similarity to an Ideal Solution to determine the optimum hemispherical dimple core design for a sandwich panel. The results showed that a hemispherical dimple core with a diameter of 6 mm and a depth of 3 mm obtained the highest Closeness Coefficient values of 0.9937 and 0.9294 under 50% and 70% constant cyclic loading. This indicated that a small sized hemispherical dimple core design enhanced the bonding strength and damage distribution better than a large sized of dimple core design. The use of fuzzy triangular elements in the hybrid decision-making method increased the accuracy of the optimisation results and reduced the costs and time required in the process design.

Keywords: Delamination, Fuzzy Triangular, Hemispherical Dimple, Optimisation, Sandwich Panel.

INTRODUCTION

In sandwich panel applications, the bonding strength between the interlaminar layers is a crucial aspect as it significantly affects the mechanical performance of a sandwich panel, especially when subjected to constant cyclic loading conditions. Most recent core designs incorporate an open and closed cavity into the body structures such as foam, honeycomb and lattice. Due to this in-built cavity, the performance of a sandwich panel under cyclic loading can vary, which affects the delamination resistance (Faidzi et al. 2021). The performance of a sandwich panel that uses an open or closed cavity core design could deteriorate under extreme conditions like high ballistic impacts. Thus, it is vital to modify the core design configuration to increase the delamination resistance and produce the optimum bonding strength for the sandwich panel.

The optimisation core design process can be long and complicated due to the multiple design criteria that may be involved, such as various sizes and dimensions (Samaie et al. 2019). Therefore, the various core design criteria make it crucial to determine an optimum core design using the Multi-Criteria Decision-Making (MCDM) method to reduce the design process duration. The Fuzzy-Analytical Hierarchical Process (F-AHP) is important in determining the fuzzified weightages for each criterion. It is analysed using the Fuzzy-Technique for the Order of Preference by Similarity to an Ideal Solution (F-TOPSIS) to identify the optimum hemispherical core design for a sandwich panel. Using this established method, a robust design solution without prejudice and bias could be found to reduce the

production costs. and determining the optimum core design, which would enable better damage resistance to the panel bonding layer and structural integrity (Faidzi et al. 2021, Samaie et al. 2019).

In this work, the varied sizes and dimensions of the hemispherical dimple core design for a sandwich panel were determined through finite element analysis (FEA) and analysed using a hybrid MCDM method incorporating the F-AHP and F-TOPSIS to define the optimum hemispherical core design. Geometrical sandwich panels were simulated under four-point bending conditions at 50% and 70% of ultimate strength. The critical criteria contributing to the delamination phenomena were segregated and analysed using the F-AHP and F-TOPSIS. After the analysis, the optimum hemispherical dimple core design for sandwich panel was determined based on the Closeness Coefficient values.

METHODOLOGY

The results of FEA under four-point bending simulation such as stress distribution, deformation and life cycle has been used as input in the MCDM methods; F-AHP and F-TOPSIS, as shown in Figures 2.1 and 2.2. The F-AHP used to determine the weightages for each of the critical criterion and ensure the stability of the PCM analysis. The output from the F-AHP was used in the F-TOPSIS analysis in order to determine the optimum hemispherical dimple core design for sandwich panel using Euclidean distance analysis between the factors.

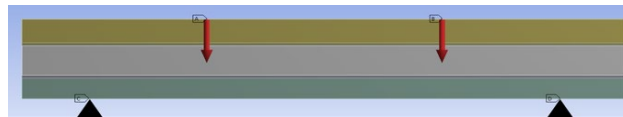


Figure 2.1: The illustration for the four-points bending simulation for the finite element analysis (FEA)



Figure 2.2: The example of stress distribution for geometrical sandwich panel using four points bending simulation.

RESULTS AND DISCUSSION

As shown in Table 2.1, the CR value was less than 0.1, indicating the fuzzified weightages from the PCM analysis were robust and free from inconsistencies (Samaie et al. 2019, Abimbola et al. 2020). The fuzzified weightages were then analysed using the F-TOPSIS to identify the optimum hemispherical dimple core design.

Table 2.1: The checking consistency procedure for all values in F-AHP

Criteria	Ratio of λ to weightage for each criterion	Consistency Ratio (CR)
Max. EVM (MPa)	10.595	$CI = \frac{\lambda_{max} - n}{n - 1}$ $CI = (10.110 - 9) / (9 - 1) = 0.1387$
Max. SS (MPa)	10.674	
TD (mm)	10.186	
Max. EVM on Bonding area (MPa)	10.259	
Max. SS on Bonding area (MPa)	10.034	
Fatigue Life at Core Panel (CAL)	9.763	

Criteria	Ratio of λ to weightage for each criterion	Consistency Ratio (CR)
		Using the CI value, the consistency ratio were determined: $CR = \frac{CI}{RI}$ CR = (0.1387) / 1.45 = 0.096
Max Damage Value on Core (CAL)	10.051	
Fatigue Life at Core Panel (VAL)	9.943	
Max Damage Value on Core (VAL)	9.482	
Average λ Max	10.110	CR < 0.1; Acceptable

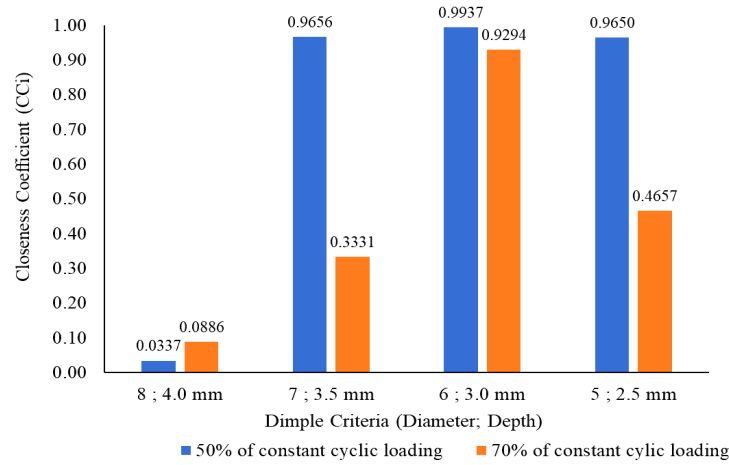


Figure 2.3: The Closeness Coefficient value for varies hemispherical dimple core design under 50 and 70% of constant cyclic loading.

Figure 2.3 shows that the dimple core with a 6.0 mm diameter and a 3.0 mm depth obtained the highest Closeness Coefficient value. The damage distribution on the core panel shows that a large dimple configuration tended to dominate at the region of low cycle fatigue life with the maximum damage value, as shown in Figure 2.4. This finding might relate to the small size and dimension of the dimple core, which contributed to the lower damage distribution and produced an optimum performance for the sandwich panel, compared to the large size configuration.

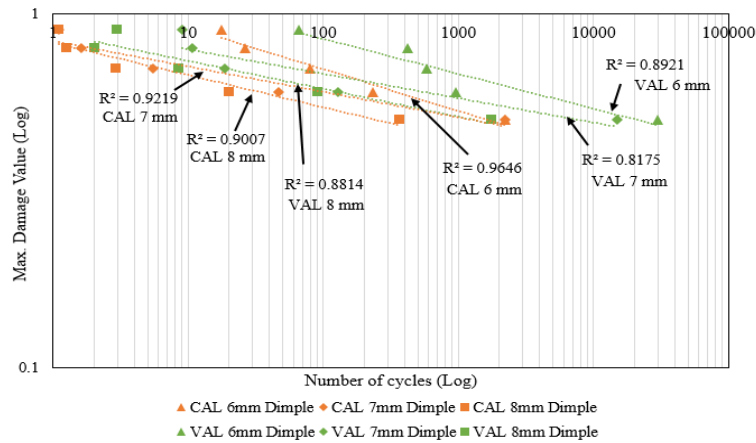


Figure 2.4: The damage plot in logarithmic scale for the maximum damage distribution on the core panel

CONCLUSION

An optimum hemispherical dimple core design for sandwich panel was determined using a hybrid MCDM method. The main critical criteria contributing to sandwich panel failure were obtained through FEA under constant cyclic conditions and analysed using the F-AHP and F-TOPSIS. A hemispherical dimple with a diameter of 6.0 mm and a depth of 3.0 mm could be selected as the optimum core design for the sandwich panel. Finally, the findings reveal a convenient way for optimising the design process with multiple criteria without increasing the production costs.

REFERENCES

- Abimbola, H. A., Bolanle, A. O., Adebayo, O. A. 2020. Performance analysis of fuzzy analytic hierarchy process multi-criteria decision support models for contractor selection. *Scientific African*, Volume 9, e00471.
- Faidzi, M.K., Abdullah, S., Abdullah, M.F., Azman, A.H., Hui, D., Singh, S.S.K. 2021. Review of current trends for metal-based sandwich panel: Failure mechanisms and their contribution factors. *Engineering Failure Analysis* 123: 105302;
- Samaie, F., Meyar-Naimi, H., Javadi, S., Farahani, H. 2019. Comparison of Sustainability Models in Development of Electric Vehicles in Tehran Using Fuzzy TOPSIS Method. *Sustainable Cities and Society*. 53. 101912.

ACKNOWLEDGEMENT

The authors would like to express their gratitude to Universiti Kebangsaan Malaysia (GUP-2021-016) for funding this research

P03

Parametric Studies on Compressive Characteristics of the Spherical-roof Contoured Core (SRCC): A Numerical Study

Q. Ma^{*1,2}, M. R. M. Rejab^{1,2}, S. A. Hassan³, M. Azeem⁴, and Saffirna M. S.¹

¹*Structural Performance Materials Engineering (SUPERME) Focus Group, Faculty of Mechanical & Automotive Engineering Technology, Universiti Malaysia Pahang Al-Sultan Abdullah, 26600 Pekan, Pahang, Malaysia*

²*School of Mechanical Engineering, Ningxia University, 750021 Yinchuan, China*

³*Faculty of Engineering, Universiti Teknologi Malaysia, Skudai, Johor, Malaysia*

⁴*Mechanical Engineering Department, Universiti Teknologi Petronas, Perak, Malaysia*

Tel: +60172667860 Email: neromaquanjin@gmail.com

ABSTRACT

This paper aims to investigate the compressive characteristics of the spherical-roof contoured core (SRCC). The effect of varying the number of unit cells and wall thickness is involved. It was found that increasing the number of unit cells and wall thickness of the core can significantly improve the compressive characteristics. It was observed that the initial peak load of the SRCC cores of 6 plies was 3.48, 3.71, and 3.90 times higher than the SRCC cores of 2 plies. It was exhibited that the fiber tensile and fiber compression damage has mainly occurred around the spherical-roof core dome. It is highlighted that the spherical-roof contoured core has great potential to use in aerospace applications with its excellent compressive characteristics.

Keywords: Sandwich structure, Parametric studies, Spherical-roof contoured core, SRCC, Compressive characteristics.

INTRODUCTION

There has been increasing interest in the lightweight structural design of sandwich panels used in industrial engineering applications, such as aerospace, automotive, marine, military and defence, and civil engineering (Ma et al., 2021, Castanié et al., 2020, Oladele et al., 2020, Ma et al., 2022). Sandwich panels have been continuously studied by developing a novel core design and material type with minimum weight and reasonable strength and structural performance (Huo et al., 2020). In recent years, many studies have investigated the influence on sandwich panels with different core designs, such as honeycomb core, corrugated core (Rejab and Cantwell, 2013), egg-box core (Halder et al., 2018), pyramidal lattice core, and spherical-roof contoured core, etc.

Guangyong Sun et al. studied the low-velocity impact behaviour of honeycomb core sandwich panels through experimental and numerical methods. It was found that a sandwich panel with thin face sheets and a high-density honeycomb core (Sun et al., 2021). Jianjun Zhang et al. conducted a study on the ballistic performance of origami sandwich panels, and deformation localises around the impact area was observed (Zhang et al., 2021). This paper investigates the compressive characteristics of the spherical-roof contoured core (SRCC) using the numerical method. The effect of varying the number of unit cells and core wall thickness is conducted under quasi-static loading. Particular attention is used to characterise the compressive response of the SRCC core, and several evaluation indicators are calculated to quantify the compressive performance.

RESULTS AND DISCUSSION

There are 1×1 , 2×2 , and 3×3 , which are used to investigate the effect of varying the number of the unit cell. Figure 3.1 is plotted the load versus displacement of SRCC models with 2 plies on 1×1 , 2×2 , and 3×3 -unit cells, respectively. It was observed that the response load and energy-absorbing characteristics of the core increased rapidly with the number of the unit cell, which agreed with the results of sandwich panels with different core designs (Haldar et al., 2018, Haldar et al., 2019, Haldar et al., 2016). Here, the response load increases to reach the initial peak load under the elastic deformation stage. Then, the core cell walls gradually collapse, and the response load drops under the plastic deformation stage.

The effect of wall thickness on the compressive response of the SRCC cores is shown in Figure 3.2. Displacement versus load curves of the SRCC core with 2, 4, and 6 plies exhibited similar trends as the load increased to initial peak loading under the elastic deformation stage. It was observed that the response load and energy-absorbing characteristics of the core increased rapidly with increasing the core wall thickness. The loading linearly increases to reach the initial peak load for the elastic deformation stage. Then, the load response becomes the nonlinear trend under the plastic deformation stage, which decreases progressively as the SRCC core between the two compressive platens. It was observed that the initial peak load of the SRCC cores with 6 plies was 3.48, 3.71, and 3.90 times higher than SRCC cores with 2 plies.

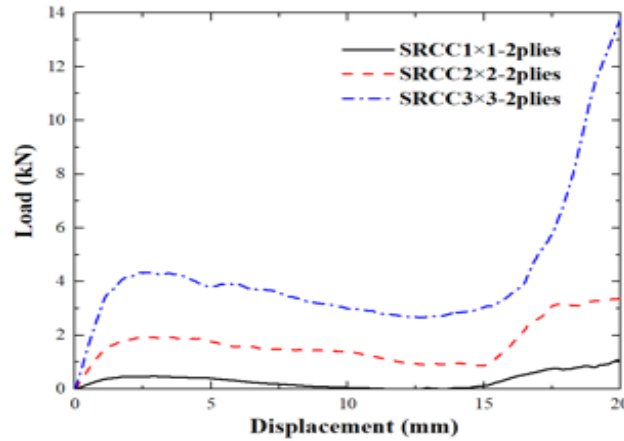


Figure 3.1: Load versus displacement curves of SRCC core models on 1×1 , 2×2 , and 3×3 -unit cells with 2 plies

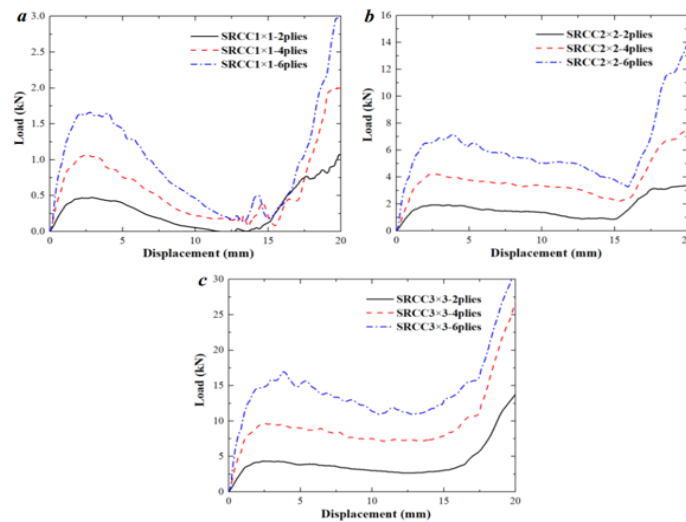


Figure 3.2: Load versus displacement curves of SRCC core models under quasi-static loading: (a) 1×1 , (b) 2×2 , (c) 3×3 -unit cells

Figure 3.3 illustrates the failure behaviour of the SRCC core models with ply 1. It was mainly shown the fiber tensile, fiber compression, matrix tensile, and compression damages, which were the typical failure mechanisms of the core structure (Ma et al., 2020, Quanjing et al., 2020). It is found that the fiber tensile and fiber compression damage mainly occurred around the spherical-roof core dome. The matrix tensile and compression damages also occur around the core dome in the zone 1 area (Ma et al., 2020).

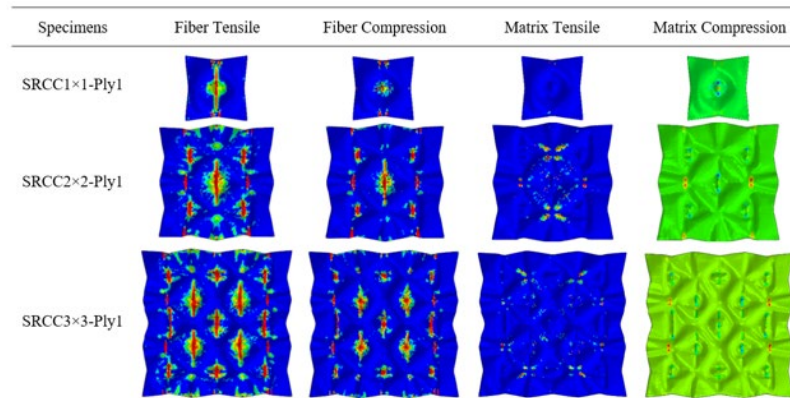


Figure 3.3: Failure behaviour of the SRCC core models with ply 1 using finite element modelling

CONCLUSION

1. For the effect of varying the number of the unit cell, it was found that the response load and energy-absorbing characteristics of the core increased rapidly as increasing the number of the unit cell.
2. For the failure damage mechanism of the SRCC core, it was found that the fiber tensile and fiber compression damage mainly occurred around the spherical-roof core dome. Furthermore, it was shown that the internal energy linearly increased by increasing the compressive displacement.

REFERENCES

- Ma, Q., Rejab, M., Siregar, J. & Guan, Z. 2021. A review of the recent trends on core structures and impact response of sandwich panels. *Journal of Composite Materials*, 55, 2513-2555.
- Castanié, B., Bouvet, C. & Ginot, M. 2020. Review of composite sandwich structure in aeronautic applications. *Composites Part C: Open Access*, 1, 100004.
- Oladele, I. O., Omotosho T. F. & Adediran, A. A. 2020. Polymer-based composites: an indispensable material for present and future applications. *International Journal of Polymer Science*, 2020.
- Ma, Q., Rejab, M., Hanon, M. M., Idris, M. & Siregar, J. 2022a. 3D-Printed Spherical-Roof Contoured-Core (SRCC) Composite Sandwich Structures for Aerospace Applications. *High-Performance Composite Structures*. Springer.
- Haldar, A., Guan, Z., Cantwell, W. & Wang, Q. 2018. The compressive properties of sandwich structures based on an egg-box core design. *Composites Part B: Engineering*, 144, 143-152.
- Rejab, M. & Cantwell, W. 2013. The mechanical behaviour of corrugated-core sandwich panels. *Composites Part B: Engineering*, 47, 267-277.
- Sun, G., Huo, X., Wang, H., Hazell, P. J. & Li, Q. 2021. On the structural parameters of honeycomb-core sandwich panels against low-velocity impact. *Composites Part B: Engineering*, 216, 108881.
- Zhang, J., Lu, G., Zhang, Y. & You, Z. 2021. A study on ballistic performance of origami sandwich panels. *International Journal of Impact Engineering*, 156, 103925.
- Haldar, A., Managuli, V., Munshi, R., Agarwal, R. & Guan, Z. 2021. Compressive behaviour of 3D printed sandwich structures based on corrugated core design. *Materials today communications*, 26, 101725.
- Haldar, A., Cantwell, W., Langdon, G., Wang, Q. & Guan, Z. 2019. The mechanical behaviour of spherical egg-box sandwich structures. *Polymer Testing*, 78, 105954.

- Ma, Q., Rejab, M., Kang, S., Idris, M. & Zin, M. 2020. The energy-absorbing characteristics of single spherical-roof contoured-core (SRCC) cell with composite materials. *International Journal of Automotive and Mechanical Engineering*, 17, 8265–8273.
- Quanjin, M., Rejab, M., Idris, M., Merzuki, M. & Shaoliang, H. Compressive properties on the spherical-roof contoured-core cell with different amounts of diamond-shaped notches. *IOP Conference Series: Materials Science and Engineering*, 2020. IOP Publishing, 012049.

ACKNOWLEDGEMENT

The authors would like to acknowledge the Ministry of Education Malaysia: FRGS/1/2019/TK03/UMP/02/10 and the Faculty of Mechanical & Automotive Engineering Technology, Universiti Malaysia Pahang Al-Sultan Abdullah (UMPSA) for funding this research: PGRS180319. The first author (Quanjin Ma) is thankful to receive the DRS (Doctoral Research Scheme) Scholarship from the Institute of Postgraduate Studies, Universiti Malaysia Pahang Al-Sultan Abdullah, Malaysia. This research work is strongly supported by the *Structural Performance Materials Engineering (SUPREME) Focus Group* and the *Human Engineering Focus Group (HEG)*.

P04

Approaches of Finite Element Analysis to Stress Analysis of Rail-Wheel Contact Stress for Light Rail Transit

M. N. Tawfik¹, M.M Padzi¹, H. Hapaz¹, and M. Zahar²

¹*Department of Mechanical Engineering, Malaysia France Institute, University Kuala Lumpur, 43650, Bangi, Selangor, Malaysia.*

²*Dept. of Train Ops, Rapid Rail Sdn Bhd - Amp. Line.
Tel: +603-8913 2800, Email: tawfik.badrudin@s.unikl.edu.my*

ABSTRACT

The objective of this research is predicting the performance for rail as under repeatedly over stressing due of axle load. The finite element analysis (FEA) has been developed to study the fatigue damage comes from rail-wheel contact stress. The 3-Dimensional of rail-wheel was designed using a software and the analysis was performed under static loading stress. The material was R260 as a steel. The factor of safety was calculated based on the stress numbers of 20kN applied to the rail. The results show that the high stress contributes to impact for the fatigue damage on rail surface. This is one of ways to study fatigue and causes the part of rail break off in certain time. Furthermore, this phenomenon has not yet reported and discussed in previous studies for light rapid transit.

Keywords: Finite Element Analysis, Rail-Wheel Contact, Contact Stress, Fatigue of Rail

INTRODUCTION

In today's railway engineering, the current demand has been dramatically increased of passengers and train frequency for heavier axle loads, train speed and greater traffic density. It is reflecting to contribute for fatigue failure of rails become most serious to derailment as able to treat us. Fatigue can be described as the component receives a force either in continuous loads, more thousands or millions of times either in variable or constant can be developing into cracks and increase a percentage to failure of components.

These failures always appear in gradually, locally and permanently with relying on repeated stress come out of dynamic stresses at critical area when the number of stresses subjected are less than the ultimate tensile strength (Cui, 2002). The term of rolling contact fatigue (RCF) have been used in rail and wheel contact stress which popular words in today researchers. RCF able to appear at surface or subsurface originated and rely on the types of materials and the stress distribution (Kim et al., 2014).

In order to predict the fatigue of rails, it vital to measure the accurate stress analysis for rail-wheel and wheel. The two types approaches can use for rail stress analysis as with based on Hertz contact theory and finite element analysis (FEA). Generally, the Hertz contact theory main backbone to gain rolling contact stress predictions and sometimes this method has less accuracy for results of rolling contact stress when the plastic deformation develops in large number of stresses.

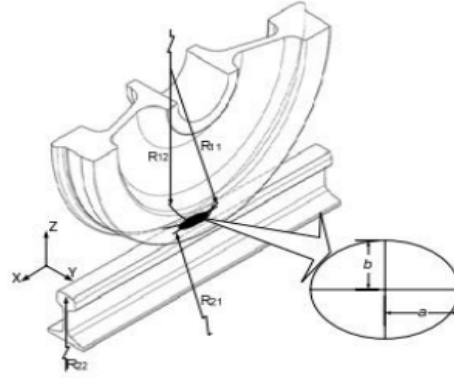


Figure 4.1: Rail-wheel contact of Elliptical
(Soemantri *et.al*, 2010)

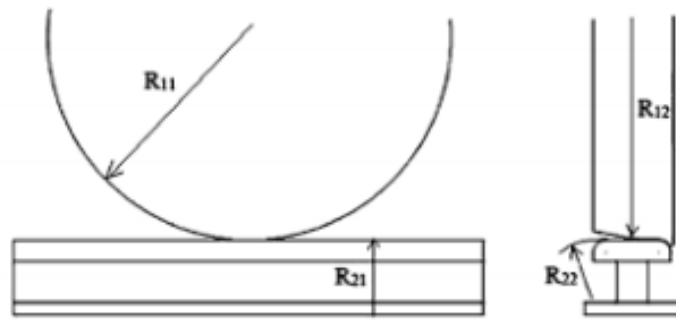


Figure 4.2: Left: Actual dimensions in millimetres; Right: Damaged Specimen of R260 for LRT

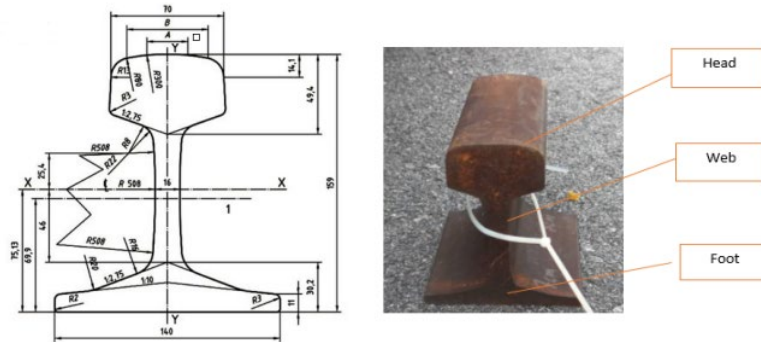


Figure 4.3: Rail-wheel contact of Elliptical
(Srivastava *et.al*, 2014)

In order to calculate a number of contact area and pressure distribution that it essential to understanding the main formula by Hertz Theory (Kim *et al.* 2014). Kim *et al.* (2014) mention the hertz contact theory consider the radii of curvature of rail and wheel profiles in contact with assumed as the constant and no plastic deformation in contact area. To find the value of normal pressure distribution $p(x,y)$ expressed as below:

$$p(x,y) = \frac{3W}{2\pi ab} \sqrt{1 - \left(\frac{x}{a}\right)^2 - \left(\frac{y}{b}\right)^2} \quad (1)$$

the representative of formula as follows:

a= half width contact area for x longitudinal direction

b= half width contact area for y lateral direction

W= total number contact force

$$a = m \sqrt[3]{\frac{3\pi}{4} \frac{W(k_1 + k_2)}{(A+B)}}, \quad (2)$$

$$b = n \sqrt[3]{\frac{3\pi}{4} \frac{W(k_1 + k_2)}{(A+B)}}. \quad (3)$$

METHODOLOGY

i. Design of rail model

The actual rail profile has been obtained from urban rail company. The type of rail was well known of R260 as standard rail and has been used for light train in urban network. The 3-d model was built in solid-work and imported into inventor for finite element analysis.

ii. Selection material properties

Table 4.1: Material Properties

Properties	Rail
Standard	EN 13674
Material Type	R260
Young's modules, E	210×10^3 MPa
Yield stress, σ	528 MPa
Poisson's ratio, ν	0.27

iii. Meshing

The meshed for rail was generated in inventor. The selections of meshing size were checking and verifying with current available in inventor.

RESULTS AND DISCUSSION

Figure 4.4 indicates the contour of Von-Mises stress distribution, it was show that the high stress contact has found in the between wheel and rail of 20kN under static analysis. The total deformation was subjected of rail as rely on the value of stress and factor of safety of rail may be decreased, respectively to the increase of value for vertical load as shown in Figure The rail was in good condition of safety as load up to 20kN.

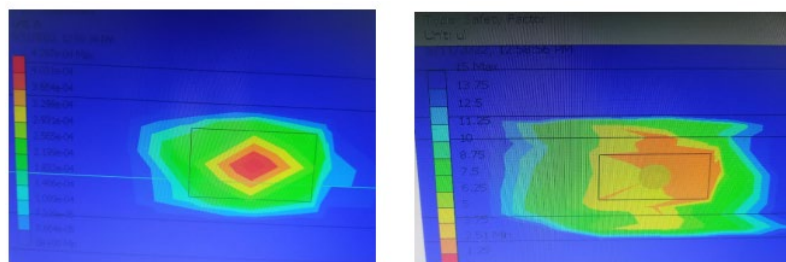


Figure 4.4: Maximum von Mises stress distribution, Maximum Factor of safety

CONCLUSION

This research study was to investigate the stress analysis of rail under static conditions which is vertical axis. The results demonstrate that the factor of safety for rail can be accepted. The value of simulated loads that up to 20KN. The findings of current study able to use for researchers in order to predict the failures of rails.

REFERENCES

- Srivastava J., Sarkar P., Ranjan V. 2014. Contact Stress Analysis in Wheel–Rail by Hertzian Method and Finite Element Method, *Journal of The Institution of Engineers (India): Series C Mechanical, Production, Aerospace and Marine Engineering*.
- Soemantri, S.S., Puja W., Budiwantoro B., Parwata M., Schipper D. .2010. Solution to Hertzian Contact Problem between Wheel and Rail for Small Radius of Curvature, *Journal of Solid Mechanics and Materials Engineering*, Volume 4 issue, 6 pages 669-677.
- Sayed H.M, Lotfy M., Zohny H.N, Riad H.S 2018. Prediction of fatigue crack initiation life in railheads using finite element analysis, *Ain Shams Engineering Journal*, Volume 9, Issue 4, pages 2329-2342.
- Zulkifli M A, Basaruddin K.S, Rahim Y.A, Afendi M., Gurubaran P., Ibrahim I., 2018. Three Dimensional Finite Element Analysis on Railway Rail, International Conference on Advanced Manufacturing and Industry Applications, *IOP Conf. Series: Materials Science and Engineering* 429.
- Ashofteh R.S and Mohammadnia A. 2014. Stress Analysis in the Elastic-Plastic Analysis of Railway Wheels, *IJR International Journal of Railway*, Vol. 7, No. 1 / March 2014, pp. 1-7.
- Gurubaran P., Afendi M, Fareisha M.A, Majid M.S, Haftirman, Rahman M.T.A 2018. Fatigue life investigation of UIC 54 rail profile for high speed rail, *IOP Conf. Series: Journal of Physics: Conf. Series* 908.
- Kim T.Y. ,Kim H.Y 2014. Three-dimensional elastic plastic finite element analysis for wheel-rail rolling contact fatigue, *International Journal of Engineering and Technology (IJET)*, ISSN : 0975-4024 ,Vol 6 No 3 Jun-Jul 2014, pp- 1593-pp1600.
- Sirata G.G, Lemu H.G. Wacławski K, Jelila Y.D 2021. Study of rail-wheel contact problem by analytical and numerical approaches, *IOP Conf. Series: Materials Science and Engineering*.
- Sun Y. and Ling L. 2021. An optimal tangential contact model for wheel-rail non-Hertzian contact analysis and its application in railway vehicle dynamics simulation. *Veh Syst Dyn.* 1–29.
- Kong, Y.S., Abdullah, S., Schramm, D., Omar, M.Z. & Haris, S.M. 2019. Optimisation of spring fatigue life prediction model for vehicle ride using hybrid multi-layer perceptron artificial neural networks. *Mechanical Systems and Signal Processing* 122: 597-621.
- Singh, S.S.K., Abdullah, S. & Nikabdullah, N. 2017. The needs of understanding stochastic fatigue failure: A review. *Engineering Failure Analysis* 80: 464-471.

ACKNOWLEDGEMENT

The authors would like to express their gratitude and thanks to MFI, Universiti of Kuala Lumpur, for funding this research under the KPT scheme

P05

Impact Behaviour of Laminated Bamboo, E-Glass and Hybrid Bamboo/E-Glass Fibre Composites

A. Suhaimi¹, A. Ali¹, K. Rassiah², and K. Z. K. Ahmad¹

¹*Department of Mechanical Engineering, Faculty of Engineering
Universiti Pertahanan Nasional Malaysia (UPNM), National Defense University Malaysia (NDUM)
Kem Sg. Besi, 57000, Kuala Lumpur, MALAYSIA.*

²*Department of Mechanical Engineering,
Politeknik Melaka Malaysia, No2 Jalan PPM 10, Plaza Pandan Malim, 75250, Melaka MALAYSIA
E-mail: aidyali@upnm.edu.my*

ABSTRACT

Natural fibres are currently used as a mutually reinforcing fibre foundation. One of their prospects, particularly that which is said to mechanical reinforcement, is frequently seen in bamboo fibre. As a result, in order to discover the most effective qualities of bamboo fibre, E-Glass fibre and hybrid bamboo/E-Glass fibre composites, this research focuses on how the mechanical properties of woven bamboo and E-Glass performed under impact loading. In the test, laminated hybrid bamboo was hit by a hard object named Impactor, it is then, analysed between the hard object and the laminated hybrid bamboo shows the impact properties that optimal. Configuration of 6,12,18 layers of woven bamboo and 6,12,18 layers of woven E-Glass, in order of thicknesses 9 mm, 18 mm, 22 mm and 4 mm, 7 mm and 10 mm, respectively were fabricated by hand lay-up. The thickness of each and individual strips of woven bamboo and E-Glass used in the range of 1.5-2.5 mm thicknesses. The results showed that the impact performance of the hybrid bamboo/E-Glass more than the non-hybrid bamboo/E-Glass.

Keywords: Woven bamboo, Glass fibre, Hybrid bamboo/E-Glass, Laminated Composite, Mechanical Properties.

INTRODUCTION

Bamboo composites are widely used and studied by researchers all over the world. In Malaysia, there are an abundance of abandoned resources. A review on thermomechanical behaviour of bamboo was detailed in Rassiah & Ahmad (2013). (Rassiah et al., 2014) had successfully identify the mechanical properties of the bamboo in 3D views that would allow the researchers to acquire the best performance of bamboo parts in terms of its mechanical properties. Moreover, the research conducted had explored the developing bamboo in form of strips. The strips of bamboo was then woven onto each other to produce a layer of bamboo (Rassiah et al., 2014). Prior to this, Ali et al., (2016) also had successfully investigated the fatigue and fracture of the woven bamboo strip.

Such understanding is important in the industry whether a bamboo composite can be a new construction material. Recent advancements made by (Ali et al., 2019) successfully improved on the effect of woven bamboo arrangement towards increasing the ballistic limit. The idea is to combine hybrid woven bamboo with woven E glass fibre to tackle the ballistic impact from a travelling bullet. The work shows experimentally that laminated hybrid bamboo composites had survived and pass the ballistic impact limit test known as V50 and NIJ tests, the results showed that the developed composites was qualified to be registered as the combat armour panel. Therefore, this paper will target investigation of woven bamboo/epoxy composite vs strip bamboo/polyester composite and compare the mechanical properties. Finally evaluate the most effective sorts of bamboo between matrix and therefore the reinforcement.

It is crucial to understand the response to impact, impact energy and impact force absorbed by the material before its failure in order to produce an effective design for a structure (Razali et al., 2014). For more in-depth research, this study implemented impact test using weight-impact test machine which is to characterise and analyse the energy absorption, performance, and mechanical properties of the composites.

MATERIALS AND METHODS

The previous success method done by previous researcher will be used in developing the woven bamboo/E-Glass fibre reinforced polyester hybrid composites which are suitable for its resistance to impact applications. Woven bamboo and E-Glass fibre are chosen as fibre materials reinforcing the polyester.

RESULTS AND DISCUSSION

Table 5.1 shows the tabulated data observed in the test. Figure 5.1 shows the testing configuration and the machine used. Figures 5.2, 5.3 and 5.4 present the velocity and energy consumed by the composite during test.

Table 5.1: Highest Peak of Force (N), Velocity (V) and, Impact Energy (J)

Series	Force (N)	Velocity (V)	Impact Energy (J)
6WB	2894.86	3.19	28.78
6WG	6571.94	5.84	94.99
12WB	5209.75	4.60	59.19
12WG	14806.91	7.93	173.85
18WB	3919.42	5.49	84.67
18WG	21826.77	10.64	312.24
18WG	21597.79	11.90	390.54
4WB			

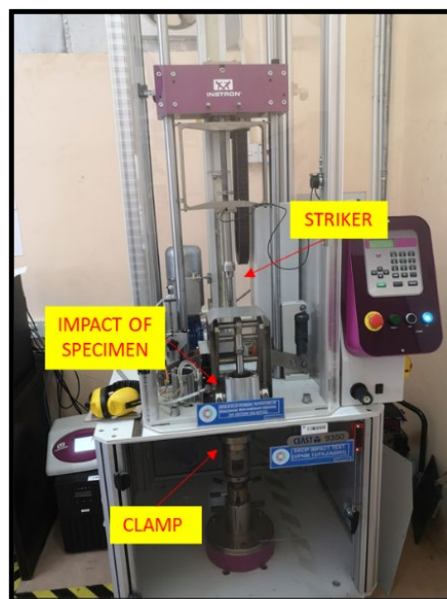


Figure 5.1: CEA3T 9350 Drop Tower Impact System for impact testing the hybrid laminated bamboo composites.

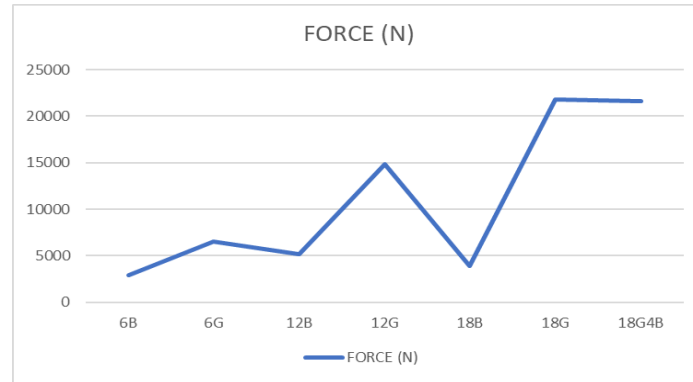


Figure 5.2: Graph of Force (N) against composites

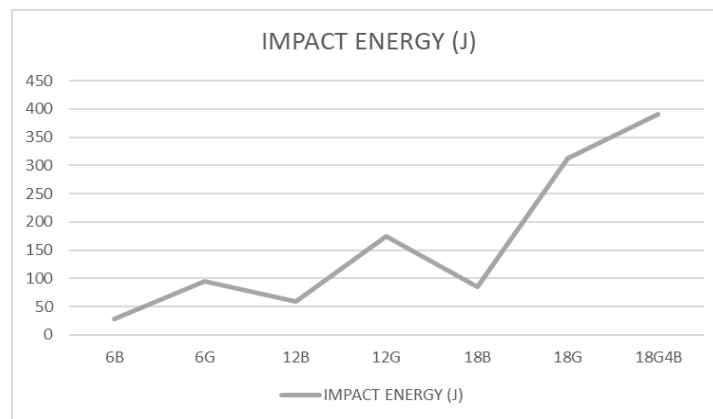


Figure 5.3: Graph of Impact Energy (J) against composites

CONCLUSIONS

In conclusions, in one hand, it is observed that, the impact strength of thicker layer of bamboo composites is obtained, the optimal layer recorded was 18, which consists on bamboo alone. On the other hand, the impact performance of hybrid bamboo/E-glass composites register 4 folds stronger than the non-hybrid bamboo.

REFERENCES

- Aidy Ali, Adawiyah, R, Rassiah, K., Ng, W. K., Arifin, F., Othman, F., and AHMAD, M. M. (2018). Ballistic impact properties of woven bamboo-woven E-glass-unsaturated polyester hybrid composites. *Defence Technology*, 15(3), pp. 282-294.
- Ali, A., Rassiah, K., Othman, F., Pueh, L. H., Earn, T. T., Hazin, M. S., & Megat Ahmad, M. M. H. (2016). Fatigue and fracture properties of laminated bamboo strips from *Gigantochloa scortechinii* polyester composites. *BioResources*, 11(4), 9142–9153.
- Rassiah, K., & Ahmad, M. M. H. M. (2013). Bamboo, thermoplastic, thermosets, and their composites: A review. *Applied Mechanics and Materials*, 330, 53–61.
- Rassiah, K., Megat Ahmad, M.M.H., And Aidy Ali. (2014) Mechanical Properties of Laminated Bamboo Strips From *Gigantochloa Scortechinii* / Polyester Composite. *Materials & Design*, 57 pp. 551-559.
- Rassiah, K., Megat Ahmad, M.M.H., Aidy Ali., Abdullah, A.H., and Nagapan, S. (2018) Mechanical properties of layered laminated woven bamboo *Gigantochloa scortechinii*/epoxy composites. *Journal of Polymers and the Environment*, 26(4), pp. 1328-1342.

ACKNOWLEDGEMENT

The authors acknowledge Universiti Pertahanan Nasional Malaysia (UPNM) for supporting this research work, Grant UPNM/2020/GPJP/TK/2, for Prof. Dr. Aidy Ali, and Politeknik Melaka (PMK) for Dr. Kannan Rassiah, and as well as The Coordinator of Composite Engineering Laboratory (FKP/UTeM) for granting permission to use all the available equipment

P06

Modelling of Creep Damage Evolution of 316H Stainless Steel using C-Shaped Specimen

I. U. Ferdous¹, N. A. Alang¹, J. Alias¹, and N. A. Razak¹

¹*Faculty of Mechanical and Automotive Engineering Technology, Universiti Malaysia Pahang Al-Sultan Abdullah, 26600, Pekan, Pahang, Malaysia*

**Tel: +60174636568, *Email: azuan@umpsa.edu.my*

ABSTRACT

Cracks are commonly found in high-temperature pipes in power plant that are subjected to creep. Understanding the crack growth behaviour is essential to estimate the creep lifetime of such components. This paper presents the numerical modelling of creep crack growth (CCG) behaviour in 316H stainless steel using C-shaped tension, CS(T), specimens. Finite element analyses, incorporating ductility exhaustion damage models were employed to simulate the crack growth. The influences of plane stress and plane stress condition on crack growth behaviour were investigated. The results showed that the model with plane stress condition is underestimated, resulting in a shorter crack growth than the plane strain. A comparable result with experimental data was found for the case of plane strain. The results suggested that 2-D model with plane strain condition is appropriate to predict the creep crack growth life.

Keywords: Creep Crack Growth, C-Shape Specimen, State of Stress, 316H Stainless Steel.

INTRODUCTION

Creep crack growth in high-temperature components is inevitable. After an extended period of operation, cracks are frequently found to form and propagate, reducing the load-bearing capability of the components. Therefore, accurate assessment methods are necessary to evaluate whether the components are fit to continue the service or not. With advances development in finite element (FE) methods, more sophisticated models can be employed in the study of CCG in complex components geometries. To date, several works on crack growth modelling of creep fracture have been reported. Hayhurst et al. (1984) employed a Kachanov-type damage variable in finite element simulations to simulate the progression of creep damage. Pandey et al. (2019), on the other hand, used a damage mechanics technique as well as the extended FEM to simulate crack growth behaviour in type 316 stainless steel. In addition, Kim et al. (2016) employed coupled models between plastic and creep damage to improve the prediction in P91 steel.

In this paper, a strain-based damage variable is used to predict CCG in a CS(T) specimen within an FE framework. The influence of plane stress and plane strain conditions is examined.

FINITE ELEMENT MODELLING

CCG modelling was performed in 316H stainless steel using CS(T) specimen at 550°C. A 2-dimensional (2D), plane stress (PS) and plane strain (PE) FE analysis was employed. The dimensions of the specimen (FE model) are tabulated in Table 6.1. The CS(T) specimen with a sharp crack was modelled and four node continuum PS/PE elements were assigned using commercial ABAQUS v6.13 software. A detailed finite element mesh applied to the models is shown in Figure 6.2. Uniform sizes of 50 µm elements have been assigned along the symmetry plane of the crack path. The material is modelled as an isotropic elasto-plastic material. A loading pin is modelled as a rigid body, and the

concentrated force of $P = 8250$ N is applied at the centre of the pin. The symmetrical condition was applied for computational efficiency. The creep constants (of Norton's power law equation) employed in FE analysis are $A=7.24 \times 10^{-32}$ and $C=10.62$ (Yatomi et al. 2008). To simulate the progression of the crack, the damage parameter, ω at each time interval is estimated according to Eq. (1). The load carrying capacity at each gauss point of the element is switched to 1 MPa (to simulate the damage) when the cumulative damage equals one ($\omega = 1.0$).

$$\omega = \int_0^t \dot{\omega} dt = \int_0^t \frac{\dot{\epsilon}_c}{\dot{\epsilon}_f} dt \quad (1)$$

In the present study, a user-defined field (USDFLD) subroutine in ABAQUS is employed to define the creep damage (as in Eq. (1)) and reduce the load-carrying capacity of damaged elements.

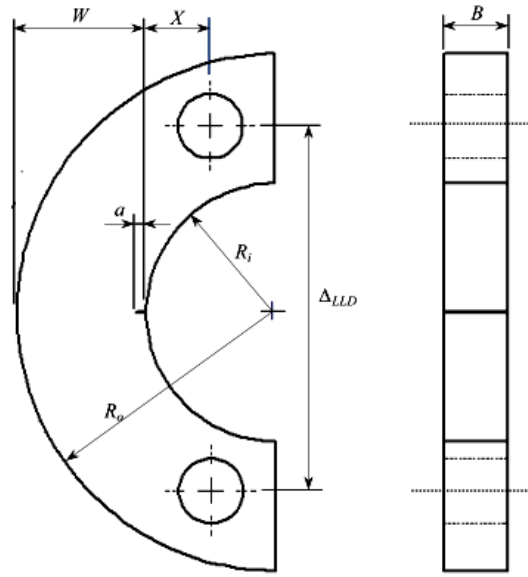


Figure 6.1: Definition of C-Shaped specimen

RESULTS AND DISCUSSION

Figure 6.3 compares the results of crack length against rupture time between FE analysis and experiment data (Davies et al. 2006). As shown in this figure, the PS condition predicts a higher crack length compared to PE for a given time. Despite the fact that the plane stress condition results in lower material constraint at the crack tip than the plane strain, creep strains can be substantially high, resulting in a larger load line displacement and faster crack propagation (Oh et al. 2011). Clearly, the C-Shaped specimen modelled using the PE condition predicts well the experiment crack growth results. The creep damage contour plot for PS conditions is shown in Figure 6.4.

Table 6.1: Details dimension of C-Shaped specimen (in millimetre)

Width, W	X	Inner Diameter, ID	Outer Diameter, OD	Thickness, a/W B
12.5	12.5	50	100	12.5 0.234

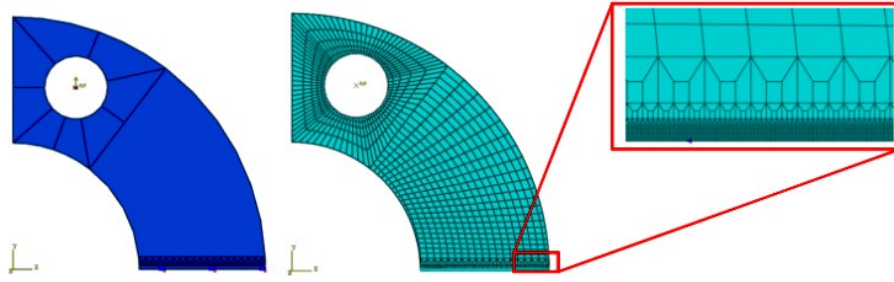


Figure 6.2: Detailed finite element mesh

CONCLUSION

The FE analysis based on the ductility exhausted damage model was performed to investigate the creep crack growth behaviour in 316H stainless steel using a CS(T) specimen. Adopting a plane stress condition leads to an underestimated creep crack growth life. In contrast, the plane strain condition shows better results as compared to experimental data.

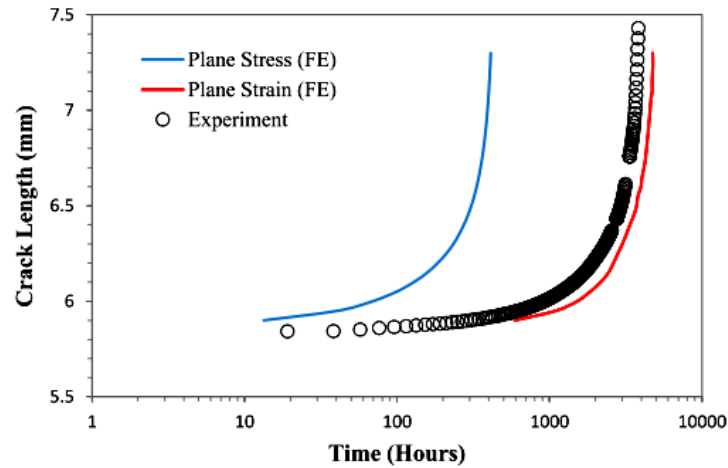


Figure 6.3: Comparison between FE prediction and experimental results

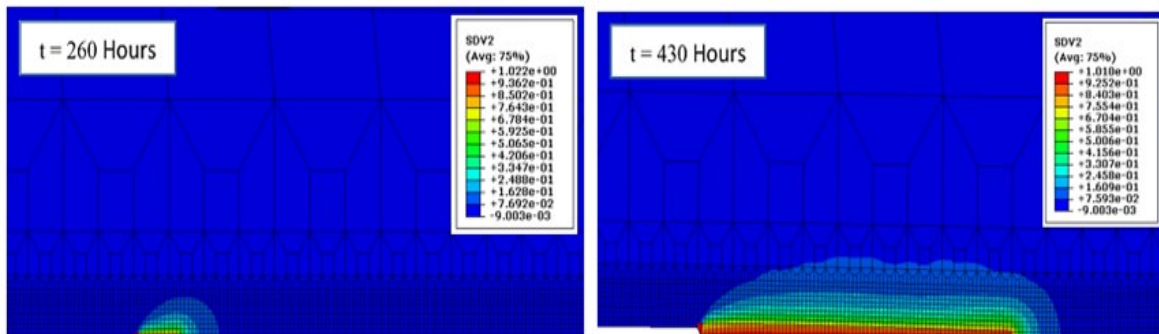


Figure 6.4: Creep damage contour

REFERENCES

- Hayhurst, D.R., Dimmer, P.R. and Morrison, C.J. 1984. Development of continuum damage in the creep rupture of notched bars. *Phil. Trans. R. Soc. Lond. A* 311:103-129.
- Kachanov, L.M., 1986. Introduction to Continuum Damage Mechanics. *Kluwer Academic Publishers, Dordrecht*.
- Pandey V.B et al. 2019. Creep crack simulations using continuum damage mechanics and extended finite element method. *Int. J. of Damage Mech.* 28(1):3–34.
- Kim D.J et al.2016. Finite element simulation of creep crack growth using combined plastic-creep damage model. *Proc. Struc. Integ.*, 2:825-831.
- Yatomi M, Davies CM, Nikbin KM. 2008. Creep crack growth simulations in 316H stainless steel. *Eng. Fract. Mech.* 75:5140–50.
- Davies C et al. 2006. Analysis of creep crack initiation and growth in different geometries for 316H and carbon manganese steels. *J. ASTM Int.* 3:1–20.
- Oh C-S et al. 2011. Creep failure simulations of 316H at 550°C: Part I – A method and validation. *Eng. Fract. Mech.* 78:2966– 77.

ACKNOWLEDGEMENT

The authors would like to express gratitude to the Ministry of Higher Education (MOHE) Malaysia for funding this research under the Fundamental Research Grant Scheme FRGS/1/2019/TK03/UMP/02/2 (University reference of RDU1901107).

Influence of Crack in Composite Plate on Natural Frequency using Fibre Bragg Grating Sensors Array

Z. M. Hafizi^{1,*} and E. Vorathin¹

¹*Advanced Structural Integrity and Vibration Research (ASIVR),
Faculty of Mechanical & Automotive Engineering Technology, Universiti Malaysia Pahang Al-Sultan
Abdullah, 26600 Pekan, Pahang, Malaysia.
Email: *hafizi@ump.edu.my*

ABSTRACT

This paper demonstrated that the presence of crack can alter the natural frequency values of a composite plate structure. An array of three fibre Bragg grating (FBG) sensors is bonded on the composite plate. A crack is formed close to one of the FBG sensors and an impact is given at the centre of the plate. The natural frequency retrieved from the FBG sensor near the crack is 68% difference from FBGs that are away from the crack.

Keywords: Fibre Bragg Grating (FBG) Array, Composite Plate, Crack and Natural Frequency.

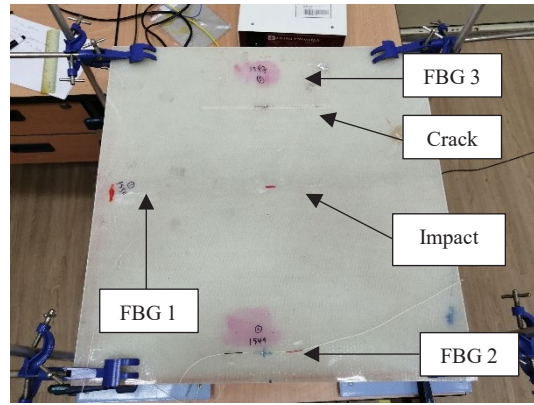
INTRODUCTION

Crack in composite material often being analysed through vibration parameters such as natural frequencies, mode shapes, damping and stiffness (Babu et al., 2015). According to Chaudhari et al. (2014), the presence of crack can distort and alter the natural frequency values. Various researchers (Duvnjak et al., 2021, Chen et al., 2021, Rekatsinas et al., 2021) have reported on the use of conventional electrical sensors in analysing the vibration parameters. However, this can increase the cost of an acquisition device as multiple electrical sensors are required in performing the vibration analysis (Yu et al., 2018).

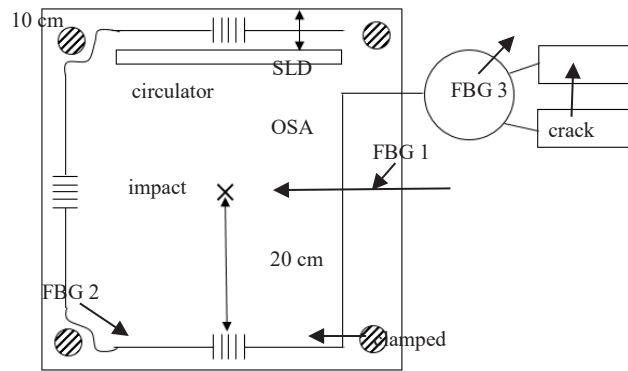
In order to overcome this issue, an array of three fibre Bragg grating (FBG) sensors was utilised. FBG is an optical fibre sensor that has the advantages of immunity to electromagnetic interference, suitable for long range monitoring and multiplexing capability (Vorathin et al., 2018). The FBG sensors array was used to capture the natural frequency and investigate how the presence of crack affect the natural frequency values.

EXPERIMENTAL SETUP

A ten layers hand lay-up composite plate was fabricated with the dimension of 45 cm (length) × 45 cm (width) × 0.3 cm (thickness) as shown in Figure 7.1. The composite plate consists of a through crack with the length and size of 17.5 cm and 0.2 cm. The crack was located close to FBG 3. An array of FBG sensors which are FBG 1, FBG 2 and FBG 3 were surface bonded at top, left and bottom of the plate. The FBG sensors array was illuminated by a Superluminescent Diode (SLD) light via an optical circulator. The reflected Bragg wavelengths from the sensors array was viewed by using an optical spectrum analyser (OSA). FBG 1, FBG 2 and FBG 3 have the reflected Bragg wavelengths of 1550.0323 nm, 1549.3869 nm and 1547.7630 nm.



(a)



(b)

Figure 7.1: (a) The picture of the experimental setup (b) The illustration of the experimental setup.

In order to excite the natural frequency, three trials of impacts were introduced at the distance of 20 cm away from all the sensors. The captured impact signals were then being analysed through Fast Fourier Transform (FFT) and the natural frequency values were compared.

RESULTS AND DISCUSSION

Figure 7.2 shows the frequency spectrums captured by the FBG sensors array for trial 1. From the results, clearly seen that the FBG sensors array capable to retrieve the natural frequency spectrums simultaneously. From the comparison, it was observed that Mode 1 natural frequency spectrum for FBG 3 was dissimilar as compared to FBG 1 and FBG 2. For trial 1, the natural frequency values for FBG 1, FBG 2 and FBG 3 were retrieved at 478.36 Hz, 479.45 Hz and 153.12 Hz. Table 7.1 shows the summary of natural frequency values for all the three trials.

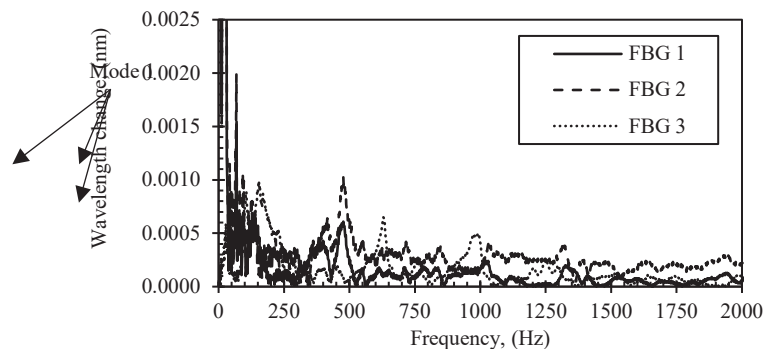


Figure 7.2: Frequency spectrums captured by FBG sensors array for trial 1

Table 7.1: Summary of Mode 1 natural frequency values for all trials

	Mode 1 frequency values (Hz)			
	Trial 1	Trial 2	Trial 3	Average
FBG 1	478.36	482.00	478.00	479.45
FBG 2	479.45	476.00	482.00	479.15
FBG 3	153.12	154.00	154.96	154.03

From Table 7.1, the average natural frequency values for FBG 1, FBG 2 and FBG 3 were retrieved at 479.45 Hz, 479.15 Hz and 154.03 Hz. The natural frequency values difference between FBG 1 and FBG 2 were relatively insignificant at 0.06%. However, as compared to FBG 1 and FBG 2, the difference of more than 68% was obtained for FBG 3. The difference in natural frequency values captured by all the FBGs justified that the presence of crack has altered the wave propagation (Chaudhari et al., 2014). Therefore, resulted in dissimilar natural frequency values.

CONCLUSION

As a conclusion, this study has successfully proven that the presence of crack has resulted in dissimilar natural frequency values. The natural frequency value near the crack at FBG 3 was more than 68% difference as compared to natural frequency values that were away from the crack.

REFERENCES

- Babu, R. K. P., Kumar, B. R. & Rao, K. M. 2015. Vibration analysis of low pressure steam turbine blades with crack. *Canadian Journal of Basic and Applied Sciences*, 3, 8-28.
- Chaudhari, C. C., Gaikwad, J. A., Bhanuse, V. R. & Kulkarni, J. V. Experimental investigation of crack detection in cantilever beam using vibration analysis. 2014 First International Conference on Networks & Soft Computing (ICNSC2014), 2014. IEEE, 130-134.
- Chen, Y., Mendoza, A. S. E. & Griffith, D. T. 2021. Experimental and numerical study of high-order complex curvature mode shape and mode coupling on a three-bladed wind turbine assembly. *Mechanical Systems and Signal Processing*, 160, 107873.
- Duvnjak, I., Damjanović, D., Bartolac, M. & Skender, A. 2021. Mode shape-based damage detection method (MSDI): Experimental validation. *Applied Sciences*, 11, 4589.
- Rekatsinas, C. S., Chrysochoidis, N. A. & Saravanos, D. A. 2021. Investigation of critical delamination characteristics in composite plates combining cubic spline piezo-layerwise mechanics and time domain spectral finite elements. *Wave Motion*, 106, 102752.
- Vorathin, E., Hafizi, Z. M., Ghani, S. a. C., Siregar, J. P. & Lim, K. S. 2018. FBGs real-time impact damage monitoring system of GFRP beam based on CC-LSL algorithm. *International Journal of Structural Stability and Dynamics*, 18, 1850075.
- Yu, M., Fu, S., Gao, Y., Zheng, H. & Xu, Y. 2018. Crack detection of fan blade based on natural frequencies. *International Journal of Rotating Machinery*, 2018.

ACKNOWLEDGEMENT

This research received the financial support from Ministry of Higher Education Malaysia under the Fundamental Research Grant Scheme FRGS/1/2019/TK03/UMP/02/7 @ RDU1901116

Modelling the Mechanical Behaviour of PolyJet 3D-Printed Resin Sand Material

Z. Bo¹, L. Shaoying², M. R. M. Rejab³, L. Jiayuan¹, and L. Yushi¹

¹School of Mechanical Engineering, Ningxia University, 750021, Yinchuan, Ningxia, China

²State IJR Center of Aerospace Design and Additive Manufacturing, School of Mechanical Engineering, Northwestern Polytechnical University, 710072, Xian, Shaanxi, China

³Faculty of Mechanical & Automotive Engineering Technology, Universiti Pahang Malaysia Al-Sultan Abdullah, Pekan, Malaysia

Tel: +8613895605796 Email: zhangb@nxu.edu.cn

ABSTRACT

High resolution additive manufacturing (AM) greatly releases the design freedom and promotes the development of advanced structural design methods. In order to fully take advantages of voxelated forming methods and model the mechanical behaviour of 3D-printed furan resin sand material by Poly Jet, the mechanical evaluation method based on Poly Jet 3D printing technology should cover different printing and loading orientations. This study systematically evaluates the tensile and compressive behaviour of PolyJet 3D-printed casting sand material, predicting the failure behaviour accurately of the fabricated validated structures. The experimental results suggest that the tensile and compressive strength are about 2.33MPa and 6.28MPa, respectively. For the structural testing, the simulation agrees well with the experimental results. In addition, the evaluating method in this paper is also applicable to other additive manufactured materials.

Keywords: Poly Jet, Sand Material, Furan Resin, Failure Behaviour.

INTRODUCTION

Additive manufacturing (AM) greatly releases the degree of freedom of design owing to its powerful ability to fabricate some complex structures. Therefore, AM has been widely used in aerospace, electronics, medical treatment, automobile, mould manufacturing and other industries (Zhu et al., 2016), which reduce the developing cost and cycle, improving the performance of final products. In casting industry, the raw materials are usually sand particles and furan resin, therefore PolyJet (Yuan et al., 2021) is the most appropriate technology to bond small particles due to process advantages. Actually, a variety of commercial Poly Jet equipment have been developed to print sand mould with special surfaces or complex shapes.

However, relatively big sand particles and limited resin bonding make the mechanical properties of the final printed materials very weak. Besides, the failure mode is unstable, which has greatly hindered the use safety in the structure lifecycle. It is urgent to model the mechanical behaviour of 3D printed furan resin sand material by Poly Jet. AM utilizes a layer-by-layer manner to fabricate structures, as a result, some researches have proposed anisotropic mechanical model to describe the material behaviour. Li et al. (Li et al., 2020) adopted the transversely isotropy to establish the constitutive model of SLA and SLS printed materials (Li et al., 2021).

Although this model provides guidance for the mechanical evaluation of 3D printed sand mould, the extremely weak bonding strength and a quantity of micro defects may result in typical brittle characteristics on macro-scale. It is necessary to perform systematic mechanical tests to develop unique strength failure criteria to describe the failure behaviour of casting sand samples fabricated by PolyJet. In this investigation, firstly the off-axis tensile and compressive experiments are designed to obtain

basic strength data. Then the Mohr strength criterion is established to describe the failure of Poly Jet fabricated resin sand material, and the case study structures are tested. Finally, the finite element analysis is performed to predict the structural load capacity.

RESULTS AND DISCUSSION

As shown in Figure 8.1, the tensile and compressive specimens in various building directions are printed for evaluating anisotropic stiffness and strength. Typical experimental curves are plotted in Figure 8.2, and the average tensile strength can be obtained for each specimen. Then the Mohr failure criterion is established to describe the failure of Poly Jet fabricated resin sand material as follow:

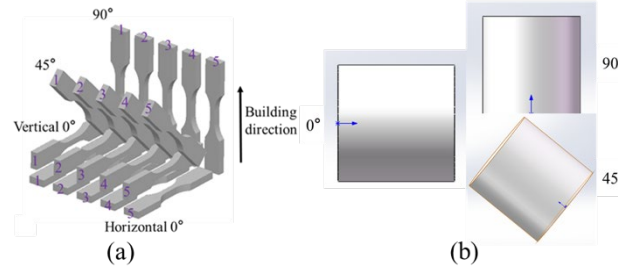


Figure 8.1: Specimens in various building directions: (a) tensile test, (b) compressive test

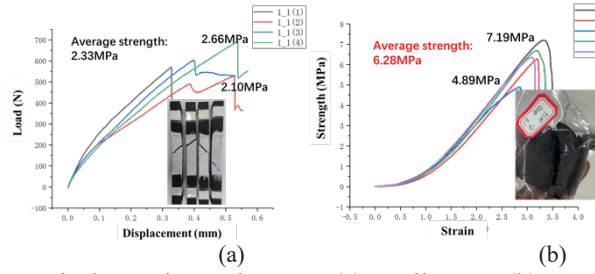


Figure 8.2: Typical experimental curves: (a) tensile tests, (b) compressive tests

$$\sigma_1 - ([\sigma_T]/[\sigma_H]) \sigma_3 \leq [\sigma_T]/n \quad (1)$$

where, σ_1 is the first principal stress, σ_3 is the third principal stress, σ_T is the tensile strength, σ_H is the compressive strength, and n is the safety factor, 2.5.

The cantilever beam in Figure 8.3 is printed to perform the experimental validation. The load capacity of the cylindrical beam is 91 kg under standard gravitational acceleration. Finally, the finite element model is established to calculate the Mohr stress and predict the failure limitation. (see Figures 8.4 and 8.5).

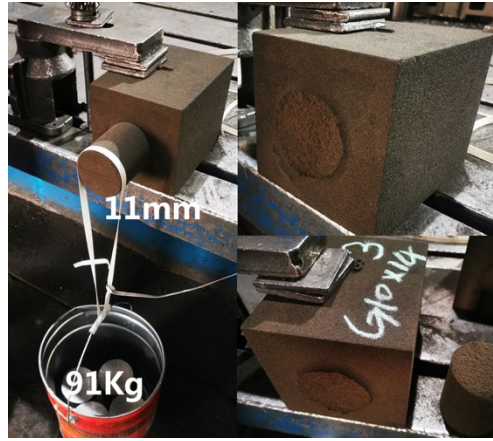


Figure 8.3: The cantilever beam for experiment

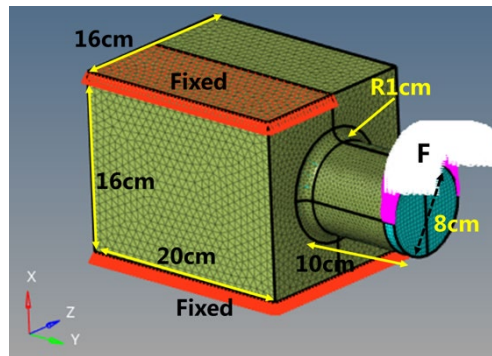


Figure 8.4: The finite element model for the beam

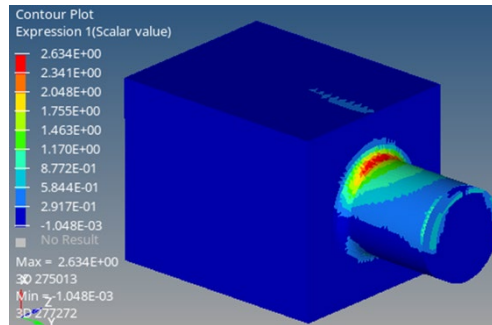


Figure 8.5: The calculated results (Mohr stress)

CONCLUSION

It can be found from the experimental curves that the tensile and compressive strength are 2.10–2.66 MPa and 4.89–7.19 MPa, respectively. Then the printed cylindrical beam supports 91 kg under standard gravitational acceleration. The fracture cross-section is relatively flat, which demonstrates strong brittleness for the Poly Jet fabricated resin sand material. After that, the finite element model is established as a physical test, and the maximum Mohr stress is 2.634 MPa and it is within the range of tensile strength, exhibiting the effectiveness of this evaluation method, proving that Mohr failure criterion is appropriate to describe the failure of Poly Jet 3D printed resin sand materials.

REFERENCES

- Li, S., Yuan, S., Zhu, J., Wang, C., Li, J. & Zhang, W. 2020. Additive manufacturing-driven design optimisation: Building direction and structural topology. *Additive Manufacturing* 36: 101406.
- Li, S., Yuan, S., Zhu, J., Zhang, W., Zhang, H. & Li, J. 2021. Multidisciplinary topology optimisation incorporating process-structure-property-performance relationship of additive manufacturing. *Structural and Multidisciplinary Optimisation* 63: 2141-2157.
- Yuan, S., Li, S., Zhu, J. & Tang, Y. 2021. Additive manufacturing of polymeric composites from material processing to structural design. *Composites Part B: Engineering* 219: 108903.
- Zhu, J.-H., Zhang, W.-H. & Xia, L. 2016. Topology optimisation in aircraft and aerospace structures design. *Archives of Computational Methods in Engineering* 23: 595-622.

ACKNOWLEDGEMENT

The authors gratefully acknowledge supports for this work from Project of Natural Science Foundation of Ningxia (Grant No.2022AAC02003), and Key Technology R&D project of Ningxia (Grant No. 2018BFH0300)

P09

Numerical Simulation of P91 Steel Under Low Cycle Fatigue Loading

M. A. A. Roslin, N. A. Razak, N. A. Alang, and N. Sazali

*Structural Performance Materials Engineering Focus Group (SUPREME),
Faculty of Mechanical and Automotive Engineering Technology,
Universiti Malaysia Pahang Al-Sultan Abdullah, Pekan, Pahang.
Tel: +60192832475 Email: norhaida@ump.edu.my*

ABSTRACT

The P91 steel low cycle fatigue behaviour have been analysed using the ABAQUS software. Low cycle fatigue parameters from the experimental data of low cycle fatigue testing of a previous study were taken to develop the material model parameters for the numerical simulation. The combined hardening parameters were developed and the simulation uses strain amplitude varying between 0.3% and 0.6% with a constant strain rate of $0.1\%s^{-1}$ at room temperature. The material exhibits cyclic softening as stress amplitude of the material decreases with increase in the number of cycles. Cyclic softening behaviour is more noticeable with the increase in strain amplitude as it resulted in lower fatigue life. The analysis of the low cycle fatigue P91 steel can give a significant understanding on the usage of P91 steel to the industry applications.

Keywords: Cyclic softening, Low cycle fatigue, P91 steel, Strain amplitude.

INTRODUCTION

Many engineering components in power generation plants, gas turbines and petrochemical industries that operates at high temperatures will undergo creep-fatigue loading conditions which will affect the creep, fatigue and creep-fatigue life of the components (Saad et al. 2016). The introduction of the use of P91 steel greatly improve the industries as it possesses good thermo-physical properties, excellent thermal conductivity and thermal resistance in addition to its great creep strength which is up to $550^{\circ}C$ (Dundulis et al., 2017). However, the power plant operations will directly affect the components of the power plant itself in terms of useful creep, fatigue and components creep-fatigue life (Zhang et al., 2014). The downside of P91 steel are they cannot sustain the cyclic mechanical and thermal loading in power plant operations in addition to the material prone to creep occurrence at high temperature $T > 550^{\circ}C$ (Alang et al., 2016). Prolonged exposure of the material to cyclic loading may resulted in low cycle fatigue which may cause thermomechanical fatigue failure to the components (Matheron et al., 2016).

According to Kannan & Sankar (2013) at higher amplitude, the low cycle fatigue life decreases and the fatigue damage increases. The softening rate of the material increases proportionally with strain amplitude of the material which shows that the cyclic stress response of the material is strongly dependent on the strain amplitude as mentioned by Golanski et al. (2013). The hysteresis loop for the LCF tests at constant strain rate shows that the higher strain amplitudes resulted in higher peak stress on both tension and compression loading as mentioned by Alang et al. (2016).

The low cycle fatigue behaviour of P91 steel may differ for steel with prior deformation compared to steel without prior deformation (Li et al., 2010). Prior creep strain can be used as pre conditioning in engineering components in elevated temperature operations (Kayatzadeh et al., 2017). This shows that the prior deformation can affect the degradation of the material and also the fatigue failure of material in cyclic loading as stated by N.A. Razak (2018). Experimental work can be time and cost consuming

and numerical simulation is another reliable alternative as it can also examine the influence of prior creep damage on P91 steel on the low cycle fatigue behaviour and fatigue life of the material under various strain conditions.

Therefore, this study uses numerical simulation in ABAQUS to investigate the low cycle fatigue behaviour of P91 steel. The material model using combined hardening rules were calculated based on the experimental data for low cycle fatigue tests from a previous study by N.A. Razak (2018). A constant strain rate of $0.1\%s^{-1}$ are used with various strain amplitudes ranging from 0.3% to 0.6%.

RESULTS AND DISCUSSION

The cyclic stress response for various strain amplitude is tabulated from the simulation. Figure 9.1 shows the material exhibits cyclic softening as shown in the decrease of stress amplitude with increasing number of cycles. The higher strain amplitudes, the lower the fatigue life of the material and shows that the cyclic softening and life of material are dependent on the strain amplitudes.

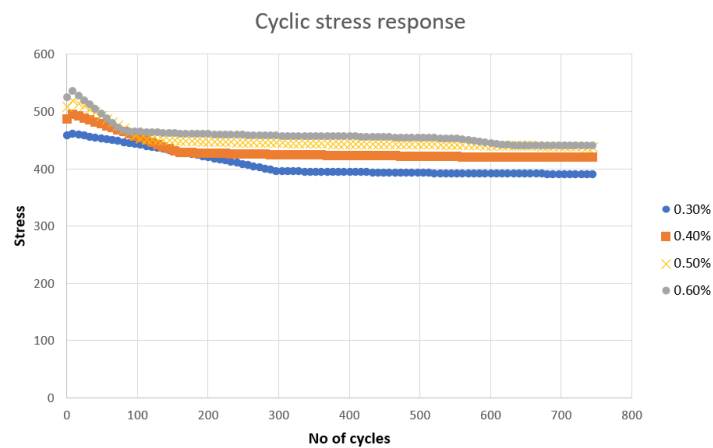


Figure 9.1: Cyclic stress response of low cycle fatigue with various strain amplitudes

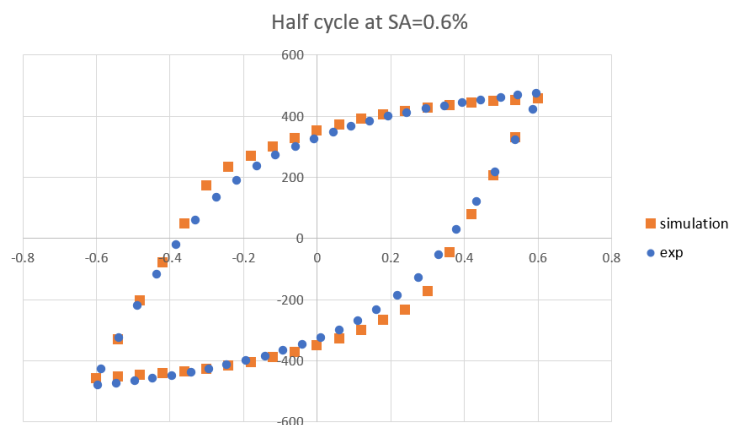


Figure 9.2: Simulation vs experimental hysteresis loop at strain amplitude 0.6% at half cycle

The hysteresis loop obtained from the simulation can correlate well with the experimental hysteresis loop as shown in Figure 9.2.

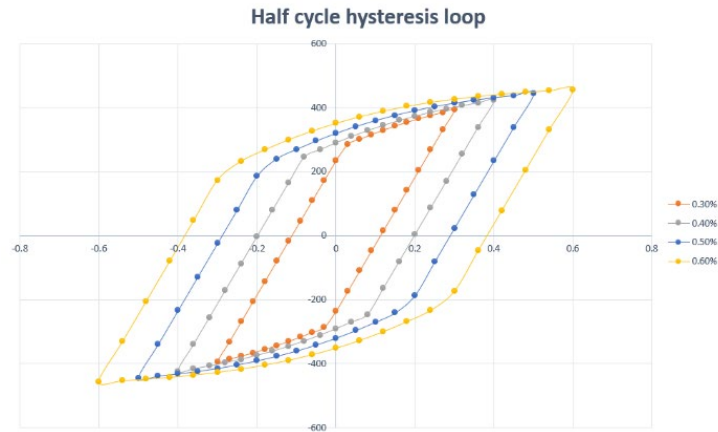


Figure 9.3: Half cycle hysteresis loop for strain amplitude of 0.3%, 0.4%, 0.5% and 0.6%.

Figure 9.3 shows that the higher strain amplitude, the higher the peak stress and plastic strain.

CONCLUSION

The determination of properties from the experimental low cycle fatigue test were used to determine the material model of the simulation by using combined hardening. The simulation results can be seen as successful as the hysteresis loop shows relatively similar with the experimental hysteresis loop. The low cycle fatigue simulation compares the cyclic softening behaviour with different strain amplitudes in a good correlation with the experimental results. Lower strain amplitudes have higher number of cycles compared to higher strain amplitudes. Stress and plastic strain of the material is dependent on the strain amplitude as shown in the simulation.

REFERENCES

- A. A. Saad, Z. Bachok, and W. Sun, 2016. A study on the damage evolution of P91 steel under cyclic loading at high temperature. *International Journal of Automotive and Mechanical Engineering* 13:3564–3573.
- D. F. Li, N. P. O’Dowd, C. M. Davies, and K. M. Nikbin, 2010. A review of the effect of prior inelastic deformation on high temperature mechanical response of engineering alloys. *Int. J. Press. Vessel. Pip.*, 87: 531–542.
- G. Dundulis, R. Janulionis, A. Grybėnas, V. Makarevičius, and R. Dundulis, 2017. Numerical and experimental investigation of low cycle fatigue behaviour in P91 steel. *Engineering Failure Analysis* 79:285–295.
- G. Golański and S. Mroziński, 2013. Low cycle fatigue and cyclic softening behaviour of martensitic cast steel. *Eng. Fail. Anal.*, 35:692–702.
- N. A. Alang, C. M. Davies, and K. M. Nikbin, 2016. Low Cycle Fatigue Behaviour of Ex-Service P92 Steel at Elevated Temperature. *Procedia Struct. Integr.*, vol. 2, pp. 3177–3184.
- N. A. Razak, 2018. Creep and Creep-Fatigue Interaction in New and Service Exposed P91 Steel. Imperial College London.
- P. Matheron, G. Aiello, O. Ancelet, and L. Forest, 2016. Recommendations for joint fatigue coefficients for welded P91 junctions at 550°C. *J. Nucl. Mater.*, vol. 472, pp. 240–246.
- R. Kannan, V. Sankar, R. Sandhya, and M. D. Mathew, 2013. Comparative evaluation of the low cycle fatigue behaviours of P91 and P92 steels. *Procedia Eng.*, vol. 55, pp. 149–153.
- X. Zhang, S. T. Tu, and F. Xuan, 2014. Creep-fatigue endurance of 304 stainless steels. *Theor. Appl. Fract. Mech.*, vol. 71, pp. 51–66

ACKNOWLEDGEMENT

The authors would like to express their gratitude and thanks to Universiti Malaysia Pahang Al-Sultan Abdullah for providing financial support under Research Grant Scheme PGRS2003146. This work was also supported by the Faculty of Mechanical and Automotive Engineering Technology, UMPSA for laboratory facilities.

P10

Comparison Creep Life Prediction of P91 Steel by Using Omega Method and Larson Miller Parameter Method

S.N.A. Rosli, N. A. Razak, and N. A. Alang

*Structural Performance Materials Engineering Focus Group (SUPREME),
Faculty of Mechanical & Automotive Engineering Technology (FTKMA),
Universiti Malaysia Pahang Al-Sultan Abdullah, 26600 Pekan, Pahang, Malaysia.
Tel: +60146299306 Email: atikahrosli65@gmail.com*

ABSTRACT

Martensitic P91 steel is desirable for structural components operating at elevated temperatures. It is extensively used in nuclear power plant boilers, pipelines, reactor pressure vessels, and steam generators due to its high creep strength and corrosion resistance. Predicting the P91's creep rupture life is crucial for safe and efficient operation. Several creep laws have been developed over time to predict deformation, damage propagation, and rupture of materials subjected to the creep phenomenon. There are many methods for creep life prediction, such as Manson-Haferd, Sherby-Dorn, Monkman-Grant, Theta project, Kachanov-Rabotnov, Wilshire, and Sinh model but the Omega method and the Larson Miller Parameter (LMP) are the only two methods approved in API 579-1/ASME FFS-1. Therefore, this paper compares the prediction of life by using the Omega method and the Larson Miller Parameter Method. Creep tests were carried out at 600°C at 160 MPa, 180 MPa, and 190 MPa. To predict the rupture life, both the omega method and LMP method have been employed, which utilised the initial creep strain rate and creep strain. The experimental data have been compared to available literature data for P91 material. The predicted life was always more significant than the experimental result, and it was strongly linked to the predicted value.

Keywords: P91, Creep Behaviour, Omega Method, Larson Miller Parameter.

INTRODUCTION

The most advanced steam power plants, according to Zhang, Y. (2009), achieve efficiencies of around 42 percent with steam temperatures reaching 600°C. The demand for such high thermal efficiency without compromising boiler and turbine component dependability and service life involves the implementation of stronger materials capable of operating under high strains at increasing temperatures. Among the many alloys, modified 9Cr-1Mo ferritic steel (also known as Grade 91 or P91) has the strongest mechanical resistance and is now widely used around the world for headers, steam pipes and generators, fast breeder reactors, and other applications involving temperatures more than 500°C (Boydak et. al. 2010). Despite its excellent creep and corrosion resistance at elevated temperatures, P91 steel may experience failure as a result of microstructural changes caused by elevated temperatures and prolonged service exposure time or due to significant local overheating of the component.

Premature failure in power plants always occurs during operation, which highlights the importance of creep range component life assessment. Numerous life prediction techniques have been developed to ensure safety at elevated temperatures, such as Theta projects (J. Perez et al. 2019), the Kachanov approach (L. An. 2015), Larson Miller Parameter (M. S. Haque 2019), and the Omega method (M. Prager 2000) can analyse these creep curves. However, only the Omega method and Larson Miller (LMP) are the only two methods provided in API 579-1/ASME FFS-1.

The Omega model is an MPC project that was originally proposed by Prager (1995). The Omega model includes a mechanism for determining the component's remaining life in the creep phase at high temperatures and pressures. Strain rate and a multi-axial damage are the Omega method parameter. It is used to forecast the rate of strain accumulation, creep damage accumulation, and estimate the time of the steel to fail as a function of stress state and temperature. It accurately predicts the rupture time of materials at lower temperatures by modelling primary and secondary creep regime deformations (Deweese et. al. 2014).

For steel subjected to the thermally induced creep process of stress rupture, the Larson-Miller parameter describes the equivalent of time and temperature. It calculates the comparable periods for stress rupture to occur at different temperatures. The idea behind this method of extrapolating stress rupture and creep data is that the absolute temperature compensated time function for a specific material should have a unique value. Only the applied stress level should determine this value.

In this work, the objective of this study is to predict the creep life of P91 by using both Omega and LMP methods. The creep tests have been performed at 600°C for 160 MPa, 180 MPa, and 190 MPa.

RESULTS AND DISCUSSION

The experimental data in Figure 10.1 demonstrates that the values Ω of the test data represented by the circle symbol is in line with the available data. In general, the values of Ω increase with decreasing levels of stress and temperature proves that the predicted values are consistent and close to the experimental results. This also implies that the simple and dependable Omega approach can be used to forecast the life of components operating at elevated temperatures.

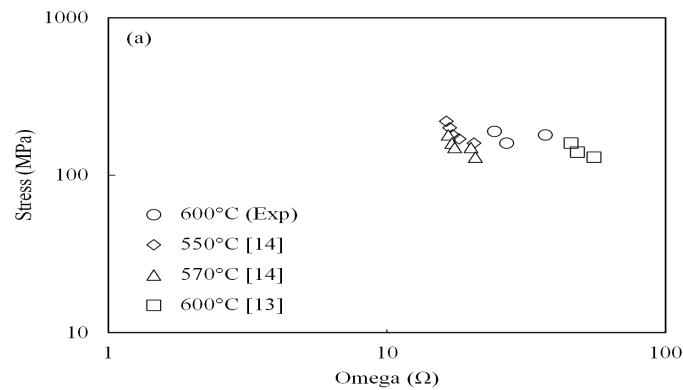


Figure 10.1: P91's stress dependency of omega, Ω

The Larson-Miller equation also demonstrates that it may be used to forecast the creep rupture life of P91 steel under a variety of stress conditions. The relationship between applied stress and the Larson-Miller parameter is fitted and presented in Figure 10.2

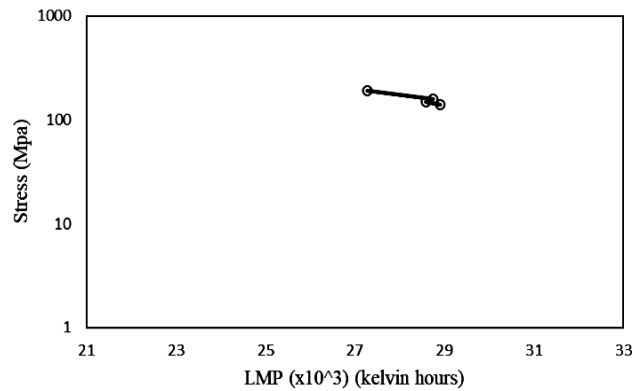


Figure 10.2: Larson Miller Parameter against stress

Table 10.1: The rupture time of experimental data, and predicted rupture time by Omega Law and Larson Miller Parameter

Temperature T/°C	Stress σ /MPa	Rupture Time (t/h)		
		Experimental	Omega Law	Larson Miller Parameter
600°C	190	17.8	19.1	27280
	180	73.8	80.7	27830
	160	860.7	960.1	28750

CONCLUSION

In this paper, creep tests were conducted on the P91 steel under 600°C at various stress levels. The magnitudes of the actual rupture times were compared to the projected creep life values using the Omega Law and the Larson-Miller methods. As a result, the predicted life using Omega Law was more dependable than the Larson Miller Parameter method. The result of Omega law and the experimental result were significantly correlated with the predicted value. In addition, the predicted rupture life values of Omega Law are consistent and close to the experimental result as it calculates the stress applied, dissimilar to Larson Miller Method. The result indicates that the predicted values of the test data by using Omega Law are consistent with the available data and the initial creep strain rate increased linearly with an increase in stress and temperature.

REFERENCES

- An, L., 2015. *The development of advanced creep constitutive equations for high chromium alloy steel (P91) at transition stress range (Doctoral dissertation, University of Huddersfield)*.
- Boydak, Ö., Keskinikılıç, S. and Koçak, M., 2010. Microstructural and Mechanical Characterisation of High Temperature and Creep Resistant Steel Weldments. In *Annual Assembly & International Conference of the International Institute of Welding* (pp. 835-841).
- Deweese, D. and Jain, R., 2014, March. Evaluation of Limit Load Analysis Methods Applied to High Temperature Design: Part 1—Consistent Material Modeling. In *Pressure Technology* (Vol. 40740, pp. 315-326). American Society of Mechanical Engineers.
- Haque, M.S., 2019, July. Modification of the MPC Omega Model to Predict Primary and Tertiary Creep. In *Pressure Vessels and Piping Conference* (Vol. 58974, p. V06AT06A032). American Society of Mechanical Engineers.
- Prager, M., 2000. The omega method—an engineering approach to life assessment. *J. Pressure Vessel Technol.*, 122(3), pp.273-280.

Prager, M., 1995. Development of the MPC omega method for life assessment in the creep range.
Zhang, Y., 2009. *Changes in microstructure and mechanical properties of P91 weld metal during creep* (Doctoral dissertation, University of Nottingham).

ACKNOWLEDGEMENT

Sincerely thanks to Universiti Malaysia Pahang Al-Sultan Abdullah for providing laboratory facilities and financial assistance under grant number FRGS/1/2019/TK03/UMP/03/5 which enabled this research to be carried out. The UMPSA Faculty of Mechanical and Automotive Engineering Technology also participated in this study.

P11

Efficient Neighbour Search Algorithm in Nonlocal Fracture Simulation

S. Kamaludin¹ and P. Thamburaja²

¹*School of Mechanical Engineering, Engineering Campus, Universiti Sains Malaysia, 14300 Nibong Tebal Pulau Pinang, Malaysia.*

²*Department of Mechanical and Manufacturing Engineering, Faculty of Engineering and Built Environment Universiti Kebangsaan Malaysia, 43600 UKM, Bangi Selangor, Malaysia.
Tel: +6013-2224505 Email: sarahkamaludin@usm.my*

ABSTRACT

In non-local simulation, a material point interaction is expanding to other material points that are located within its zone of influence. One of the most important processes in nonlocal simulation is searching the neighbour points within the individual point zone of influence. In finite element simulation, material points are represented by the integration point of the elements. The use of nonlocal approaches is expected to increase the computational cost as time complexity for neighbour search can be as high as $O(n^2)$. Therefore, an efficient neighbour search algorithm is needed to reduce the computational time for neighbour search. In this study, computational time for searching neighbours between brute force method and balanced Kd-tree method is compared for different mesh sizes and a more efficient algorithm is determined.

Keywords: Nonlocal fracture criterion, Finite element framework, Neighbour search algorithm

INTRODUCTION

Reliability of critical structural component depends on the ability to perform an accurate prediction simulation of crack initiation and propagation in material failure. In material damage simulation via finite element method (FEM), several approaches have been introduced to explicitly represent the effect of spatial discontinuities i.e., cracking behaviour. Some of the approaches that are available in the finite element analysis (FEA) commercial software are cohesive zone method (CZM), extended finite element method (XFEM) and element deletion method. However, these approaches are topological and mesh dependant i.e., to obtain an accurate prediction, mesh size and orientation plays a significant role.

One of the methods used to reduce the mesh topological effect while obtaining an accurate fracture simulation is by introducing integral nonlocal fracture criteria. The nonlocal fracture criterion can be introduced within the finite element framework by taking the cumulative average of the fracture criterion within the element zone of influence, which is defined by the internal characteristic length (Khodabakhshi et al., 2016, 2019). Based on the study conducted by Peerlings et. al. 2002, introducing strong nonlocality such as integral nonlocal formulation into fracture models, can minimize localization instabilities and remove singularity effect of damage behaviour. Element deletion method is known as a rudiment method of predicting fracture within finite element framework. However, if used in combination with nonlocal fracture criterion, it has the capability to produce an accurate prediction with minimum effects of mesh topology i.e., mesh sizes, orientation, and shapes (Thamburaja et al., 2019; Sarah et al., 2020).

The most straightforward method to find all the neighbour points in the zone of influence is using brute force or exhaustive algorithm. All material points are systematically enumerated and looped over to check if the condition is satisfied. While this method is simple to implement and if applied correctly can ensure accuracy, the process is time consuming. The time complexity of this method is $O(n^2)$. This

problem is insignificant for small numbers of points and is becoming more apparent with the growing number of points. Tree-based searching algorithm with the worst time complexity of $O(n \log n)$ is more suitable to be used in complex nonlocal problems (Vazic et al., 2020). Binary search tree (BST) offers more efficient data storage and queries. K-dimensional (Kd) tree is a special case of binary tree that can split the data into multiple dimensions and transverse through the tree for efficient data queries. In this study, the comparison of time performance between the straightforward brute force method and Kd tree were tested.

RESULTS AND DISCUSSION

To compare the computing performance between brute force method and balance Kd-tree, several triangles meshes has been built on a 10 mm x 10 mm square plate. The element number used ranges from 1000 to 500000 elements. The characteristic length used is 3Δ (3elements), and the maximum number of elements in the zone of influence (ZOI) is 70 elements. The configuration used for testing is *Intel(R) Core (TM) i7-3770K @ 3.5 GHz, 16 GB RAM, MS Windows 10 x64*. The brute force and balanced Kd tree program are written using *Fortran90* and run using GFORTRAN compiler version 6.3.0. Based on results from this study, this program will be implemented into ABAQUS VUMAT subroutine.

The time comparison for each method used is tabulated in Table 11.1 and plotted in Figure 11.1. As shown in Table 11.1, brute force is the worst performing method, and this problem is becoming more significant with many elements. This method can be used as a testing algorithm on small element numbers and not suitable for complex and large problems especially if the continuous update of the neighbour element inside the zone of influence is needed. On the other hand, the Kd-tree algorithm is more suitable for large problems as it can perform neighbour search in a shorter time.

Table 11.1: Computation time comparison between brute force method and balanced Kd-tree

Number of Elements	Brute Force Method	Balanced Kd-tree Algorithm
1106	0.2656 s	0.0938 s
9252	18.500 s	0.7031 s
48738	503.25 s	4.1875 s
100622	2206.8 s	6.5781 s
504252	53391 s	50.312 s



Figure 11.1: Time comparison for neighbour search for brute force method and balanced Kd tree

CONCLUSION

In this study, the performance of searching the neighbour points in zone of influence in nonlocal simulation using brute force method and balanced Kd-tree searching algorithm is compared. By using a fixed 2D domain size, the element sizes and numbers are varied, and the searching time was determined for zone of influence of 3Δ . Based on the findings, the brute force method is suitable for small size problems and testing purposes. For large problems, the Kd-tree algorithm offers a more efficient method.

REFERENCES

- Khodabakhshi, P., Reddy, J.N. and Srinivasa, A., 2016. GraFEA: a graph-based finite element approach for the study of damage and fracture in brittle materials. *Meccanica*, 51(12), pp.3129-3147.
- Khodabakhshi, P., Reddy, J.N. and Srinivasa, A., 2019. A nonlocal fracture criterion and its effect on the mesh dependency of GraFEA. *Acta Mechanica*, 230(10), pp.3593-3612.
- Peerlings, R.D., De Borst, R., Brekelmans, W.A.M. and Geers, M., 2002. Localisation issues in local and nonlocal continuum approaches to fracture. *European Journal of Mechanics-A/Solids*, 21(2), pp.175-189.
- Thamburaja, P., Sarah, K., Srinivasa, A. and Reddy, J.N., 2019. Fracture of viscoelastic materials: FEM implementation of a non-local & rate form-based finite-deformation constitutive theory. *Computer Methods in Applied Mechanics and Engineering*, 354, pp.871-903.
- Sarah, K., Thamburaja, P., Srinivasa, A. and Reddy, J.N., 2020. Numerical simulations of damage and fracture in viscoelastic solids using a nonlocal fracture criterion. *Mechanics of Advanced Materials and Structures*, 27(13), pp.1085-1097.
- Vazic, B., Diyaroglu, C., Oterkus, E. and Oterkus, S., 2020. Family member search algorithms for peridynamic analysis. *Journal of Peridynamics and Nonlocal Modeling*, 2(1), pp.59-84.

ACKNOWLEDGEMENT

This work is not supported by any research grants.

P12

Impact of Cyclic Operation to Leaked Pipeline Reinforced with Composite Wrapping

Z. Yusof

*Faculty of Mechanical and Automotive Engineering Technology,
Universiti Malaysia Pahang Al-Sultan Abdullah, 26600 Pekan, Pahang.
Tel: +60132903130 Email: zulkifliyusof87@gmail.com*

ABSTRACT

Metal wall loss which results in pipeline leaked has been a major issue in all industries, especially oil and gas industries. It is regardless of external or internal corrosion. To overcome this issue, several repair practices have been introduced in the industries. One of them is composite wrapping which is proven to be cost and time effective. The widely use of this composite wrapping in oil and gas industries has become a major concern by asset owners on its integrity whether it can withstand the internal pressure and flow throughout the time especially for composite wrapping installed at leaked pipeline. Over the time, the cyclic operation may impose a threat to the composite wrapping integrity. In the event of sudden pressure change due to plant trip may also impose a threat to the composite wrapping integrity. It is important to determine whether the composite wrapping can be considered as permanent repair as to solve the pipeline leaked issue. In this study, the author will elucidate the leaked pipeline reinforced with composite wrapping which subject to cyclic operation by simulating it using finite element analysis and validate the results with actual history of site installation.

Keywords: Composite wrapping, Cyclic operation, Leaked pipeline, Permanent repair

INTRODUCTION

The usage of steel pipeline in transporting product such as hydrocarbon either in the form of liquid or gas has been the safest and the most efficient method while operating in harsh environment (V.C. Khan et al. 2017). The pipeline itself may expose to various types of environments which later lead to various damage scenarios such as internal corrosion, external corrosion, erosion, mechanical damage, coating damage and others. This may impact the integrity of the pipeline itself on pressure bearing capability (M.F. Köpple et al. 2013).

In the event of the deterioration of the pipeline integrity due to wall loss either internal or external wall loss, it may lead to loss of product containment to the oil and gas facilities itself as well as imposing safety and health threat to surrounding. For examples the incident due to pipeline leak such as fire incident due to underwater pipeline leaked at Gulf of Mexico in July 2021 (A. Al-Heeti & J. Ryan 2021) and 29,000 gallons of oil spill due to pipeline leaked in Southern California in June 2016 (R Schleeter 2016) have resulted in various consequences especially to the companies and environment.

As to overcome this issue, several repair methods has been introduced in the industry which is based on ASME PCC-2 (ASME 2018). The polymer based laminate or composite wrapping method has been accepted as a method to reinforce the pipeline subject to internal or external wall loss. It is more cost effective in term of installation safety and reduction of risk of unplanned shutdown due to traditional repair method which is cut and replace method. (D. Djahida et al. 2021). Since the composite wrapping has been widely use in the oil and gas industries to overcome various types of metal wall loss to the pipeline, it has been a major concern by the asset owners on the integrity of the composite wrapping itself throughout the time. The concern is related to whether the composite wrapping able to withstand the internal pressure and flow throughout the time especially at leaked pipeline. The pipeline which

undergoes the cyclic operation may impose a threat to the composite integrity as well as the sudden pressure change due to plant trip. This may impact the consideration of using the composite wrapping as a permanent repair method for leaked pipeline by the asset owners.

In this study, composite wrapping specimen which undergo cyclic operation will be evaluate via finite element analysis and later the revalidate the result with experimental study via hydrostatic test based on N. Kumar et al. (2018) as per Figure 12.1 and cyclic test based on ASTM D2143-15 (ASTM, 2015) as per Figure 12.2. The actual historical data from site installation at selected oil and gas plant will also be evaluate in this study.

RESULTS AND DISCUSSION

Based on the historical data taken between year 2005 until 2021, there are 475 numbers of registered temporary repair via composite wrapping. Figure 12.1 shows that out of 475 wrapping, only 22 were performed due to pinhole leaked and balance due to wall thinning. Figure 12.2 shows the distribution of composite wrapping at leaked pipeline per medium type which most of cases happen at hydrocarbon services. Based on Figure 12.3, it shows that the composite wrapping able to withstand the process medium and operating condition throughout the years before permanent replacement can be done. Figure 12.4 shows the actual photo taken during inspection conducted every 6 months after installation of the composite wrapping.

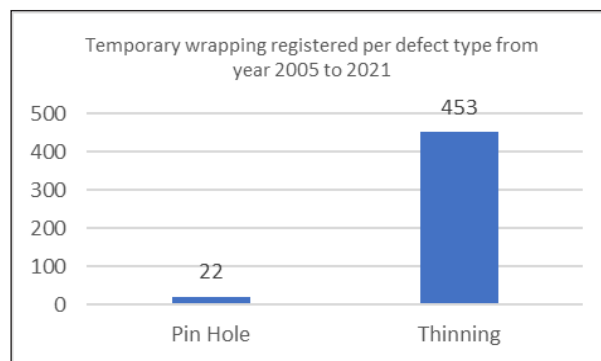


Figure 12.1: Temporary wrapping registered per defect type from year 2005 to 2021

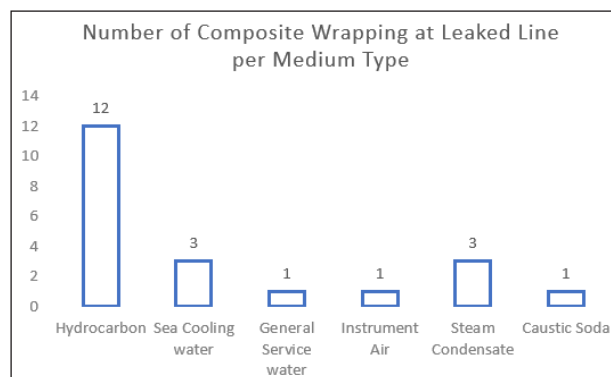


Figure 12.2: Number of Composite Wrapping at Leaked Line per Medium Type

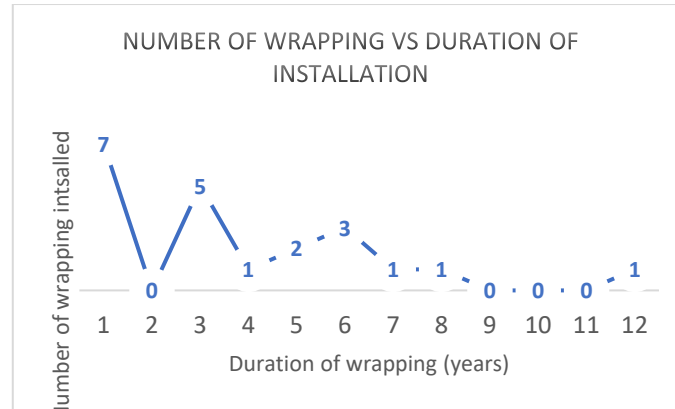


Figure 12.3: Historical composite wrapping data installed at leaked pipeline

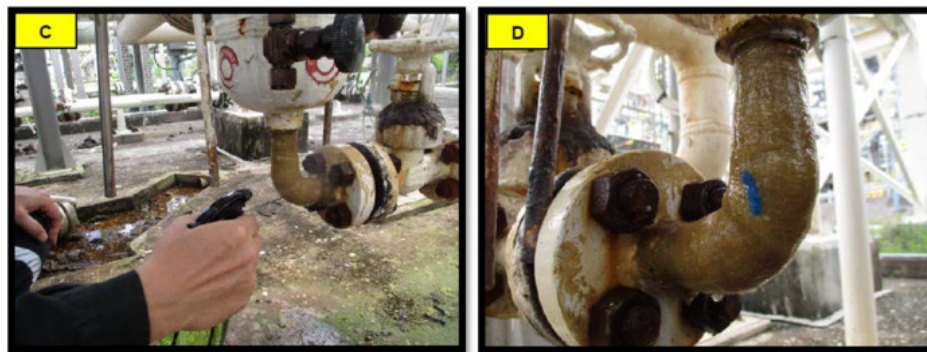


Figure 12.4: Leak inspection at wrapping location

CONCLUSION

Based on this research, it is found out that the composite wrapping able to be used for leaked pipeline. Although the pipeline itself is exposed to cyclic operating condition, the composite wrapping able to sustain the operating pressure from the medium if within design limit of the composite wrapping. Therefore, composite wrapping is suggested to be used as an alternative and cheapest method to resolve pipe leaking issue.

REFERENCES

- VC Khan, G. Balaganesan, A.K. Pradhan, M.S. Sivakumar 2017. Nanofillers Reinforced Polymer Wrap to Repair Corroded Steel Pipe Lines. *Journal of Pressure Vessel Technology*. 139. 10.1115/1.4036534
- M.F. Köpple, S. Lauterbach, W. Wagner 2013. Composite repair of through-wall defects in pipework – Analytical and numerical models with respect to ISO/TS 24817, *Composite Structures*, Vol. 95
- A. Al-Heeti, J. Ryan 2021, July 3rd. The ocean is on fire': Gas leak from underwater pipeline sees flames erupt in Gulf of Mexico. *CNET*. <https://www.cnet.com/news/the-ocean-is-on-fire-gas-leak-from-underwater-pipeline-sees-flames-erupt-in-gulf-of-mexico/>
- R. Schleeter 2016, June 23rd. ANOTHER Oil Spill: Pipeline Leaks 29,000 Gallons in Southern California, Greenpeace. <https://www.greenpeace.org/usa/another-oil-spill-pipeline-leaks-29000-gallons-in-southern-california/>
- ASME 2018. ASME PCC-2-2018 Repair of Pressure Equipment and Piping. *The American Society of Mechanical Engineers*

- D. Djahida, G. Tewfik, M. Witek, M. Abdelghani 2021. Analytical Model and Numerical Analysis of Composite Wrap System Applied to Steel Pipeline. *Materials* 2021, 14, 6393
- N. Kumar, V. Khan, G. Balaganesan, A.K. Pradhan, M.S. Sivakumar 2018. Repair of through thickness corrosion/leaking defects in corroded pipelines using Fiber Reinforced Polymer overwrap. *IOP Conference Series: Materials Science and Engineering*. 346. 012016.
- ASTM 2015. ASTM D2143-15 Standard Test Method for Cyclic Pressure Strength of Reinforced, Thermosetting Plastic Pipe. *ASTM International*

ACKNOWLEDGEMENT

The authors would like to express his gratitude and thanks to Universiti Malaysia Pahang Al-Sultan Abdullah, Malaysia LNG Sdn. Bhd, PETRONAS Group Technical Solutions and Gapura Solution Sdn. Bhd. for assisting in this research especially on the data gathering and knowledge sharing from technical expertise matter.

P13

Frequency-driven Crack Propagation in Ultrasonically-assisted Bone Cutting

D. Qianriui, S. S. Kamaludin, and N. Yusoff

School of Mechanical Engineering, Engineering Campus, Universiti Sains Malaysia, 14300 Nibong Tebal, Penang, Malaysia

Tel: +60146386343 Email: menorwahida@usm.my

ABSTRACT

Ultrasonically-assisted bone cutting with high precision has great benefits for orthopaedic surgeries. However, the brittleness and anisotropy of cortical bones make it easy to produce irregular crack propagation, large fractured chips, and surface damage during the cutting process. These can be minimized by optimizing the operating parameters of the cutting tool; cutting frequency, amplitude, speed, depth, and temperature, with consideration of the toughening mechanism of bone. Therefore, the current study is motivated to investigate the effect of varying frequency on crack propagation in ultrasonically-assisted bone cutting through the means of finite element analysis. The pattern of crack propagation in relation to the variation of frequency was investigated using the extended finite element method (XFEM) in consideration of the bone microstructures. The results showed that crack propagation is effectively controlled when the tool is operated at higher frequencies so that it penetrates directly into the osteon and Haversian canal without being deflected along the cement line, and that the cutting force decreases proportionally.

Keywords: Crack propagation, XFEM, Ultrasonically-assisted cutting, Vibration frequency, Bone microstructure.

INTRODUCTION

Ultrasonically-assisted cutting (UAC) technique that applies high-frequency vibration has been alternatively used in bone resection as it is known for allowing high cutting precision. However, postoperative trauma such as large fractured chips and surface damage may still occur due to the anisotropic property of bone and its hierarchical microstructure that contribute to its intrinsic toughening mechanism. Therefore, the performance and efficiency of the UAC procedure in bone resection strongly depend on the interaction between the tool and bone which can be quantified by looking at the pattern of crack propagation in relation to the varying operating parameters that can be optimised accordingly.

Emphasising on the influence of material anisotropy Shu and Sugita (2020) showed that the application of high vibration frequency increases strain rate that in turn, decreases the toughness of the bone, which consequently controls the direction of crack propagation effectively. The study also observed an upward crack propagation when vibration frequency is over 800 Hz, which was quantified by small chip formation that could prevent large-scale fractures of the bone. At the same time, the operating output such as force should also be monitored critically as the excessive level of cutting force that may lead to damages of surrounding tissues and result in poor surface quality (Shu & Sugita, 2020).

In the present study, the effect of four different UAC operating vibrations ranging from low (0 and 200 Hz) to high frequencies (800 and 2400 Hz) on the pattern of crack propagation in bone resection was numerically examined. A two-dimensional finite element fracture model for bone was constructed in consideration of its microstructure using extended finite element method (XFEM) implemented in the finite element commercial software Abaqus 6.12. The microstructure model of the bone consisted

of four main composite constituents, which were Haversian canal, interstitial matrix, osteon, and cement line, as shown in Figure 13.1.

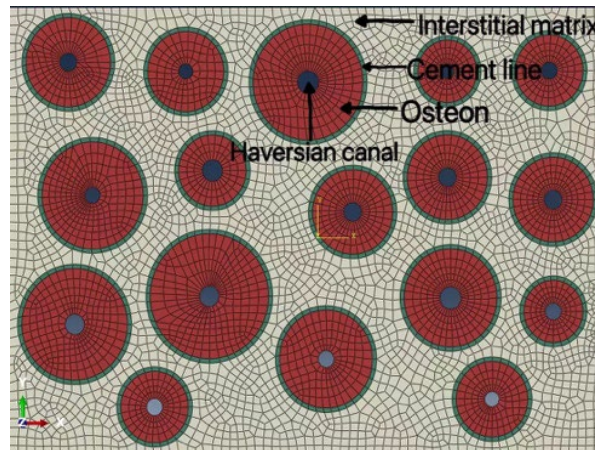


Figure 13.1: Finite element model of bone microstructure.

The material properties of the constituents listed in Table 13.1 were assigned based on experimental data published in Li et al. (2013). The failure criterion was based on a yield strain - set at 0.6% following the flow stress-strain curves of Abdel-Wahab et al. (2011). For simplicity, the blade of the cutting tool was modelled as a rigid body with the feed speed of 5 mm/s. The vibration amplitude was set constant at 20 μm in all analyses.

Table 13.1: Material properties of bone microstructure constituents.

	Osteon	Interstitial Matrix	Cement line
Elastic Modulus [GPa]	12.85	14.12	9.64
Poisson's ratio	0.17	0.153	0.49
Yield strain	0.6%	0.6%	0.6%
Fracture initiation strain	0.65%	0.65%	0.65%
Strain energy- release rate	860	238	146

RESULTS AND DISCUSSION

Figure 13.2 shows the patterns of crack propagation at all frequencies examined. From the figure, it can be observed that the crack directly penetrates the osteon in the direction of the Haversian canal without any major deflection. However, at lower frequencies (Figure 13.2(a) and (b)), a small path irregularity was evident near the cement line, which was more pronounced at zero frequency, representing a conventional cut. With increasing frequency, the crack path becomes smoother as the crack propagates toward the Haversian canal as shown in Figure 13.2(c) and (d). This would greatly reduce the risk of large-scale fractures and result in a smooth surface finish, and thus improve the surface integrity of the bone. It can also be seen that as the frequency increases, the induced stress in the vicinity of the canal decreases and is concentrated only at the tool-bone interface. This would prevent the initiation of a reversing abrupt crack at the canal, and thus allow a straight-line crack trajectory.

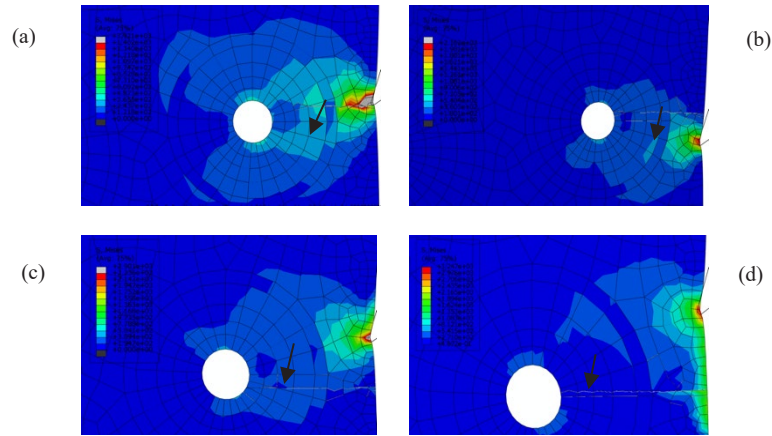


Figure 13.2: The patterns of crack propagation at frequencies (a) 0 Hz, (b) 200 Hz, (c) 800 Hz, and (d) 2400 Hz.

Looking at the operating output of the UAC cutting, the induced forces appear to gradually decrease with increasing vibration frequency as shown in Figure 13.3. This finding is consistent with many studies (Alam et al., 2011; Shu & Sugita, 2020; Shu et al., 2018) that reported the reduction in induced force with the application of mechanical vibration, which would subsequently minimise the postoperative risk.

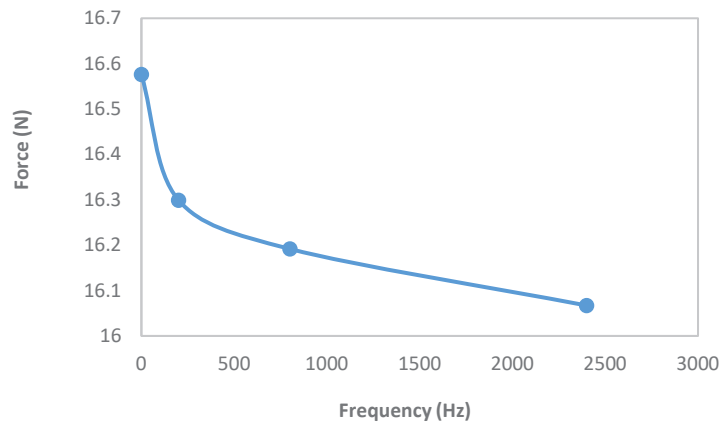


Figure 13.3: The induced force in relation to the imposed frequency.

CONCLUSION

In orthopaedic surgeries, optimised set of UAC cutting parameters should be determined to ensure a precision cutting, which is important for minimising postoperative trauma. In this study, XFEM has been used to examine the effect of varying frequency of UAC on the pattern of crack-propagation in bone resection in consideration of the bone microstructures. The results confirm that the application of high frequency alone may essentially control the tool penetration and crack propagation. The application also causes the induced cutting force to decrease proportionally with increasing frequency.

REFERENCES

- Abdel-Wahab, A. A., Alam, K. & Silberschmidt, V. V. 2011. Analysis of anisotropic viscoelastoplastic properties of cortical bone tissues. *Journal of the Mechanical Behaviour of Biomedical Materials* 4: 807-820.
- Alam, K., Mitrofanov, A. V. & Silberschmidt, V. V. 2011. Experimental investigations of forces and torque in conventional and ultrasonically-assisted drilling of cortical bone. *Medical Engineering & Physics* 33: 234-239.
- Li, S., Abdel-Wahab, A., Demirci, E. & Silberschmidt, V. V. 2013. Fracture process in cortical bone: X-FEM analysis of microstructured models. *International Journal of Fracture* 184: 43-55.
- Shu, L. & Sugita, N. 2020. Analysis of fracture, force, and temperature in orthogonal elliptical vibration-assisted bone cutting. *Journal of the Mechanical Behaviour of Biomedical Materials* 103: 103599.
- Shu, L., Sugita, N., Oshima, M. & Mitsuishi, M. 2018. Design and experimental force analysis of a novel elliptical vibration assisted orthopedic oscillating saw. *Medical Engineering & Physics* 54: 22-31.

ACKNOWLEDGEMENT

The authors would like to acknowledge the support received from the Ministry of Higher Education Malaysia under the Fundamental Research Grant Scheme: FRGS/1/2021/TK0/USM/02/1

P14

Analysis of Stress Intensity Factor via Finite Element Model

M. N. M. Husnain¹, M. R. M. Akramin¹, Z. L. Chuan², M. S. Shaari¹, and A. Takahashi³

¹ *Faculty of Mechanical & Automotive Engineering Technology, Universiti Malaysia Pahang Al-Sultan Abdullah, 26600 Pekan, Pahang Darul Makmur*

² *Faculty of Industrial Sciences & Technology, Universiti Malaysia Pahang Al-Sultan Abdullah, Lebuhraya Tun Razak, 26300 Gambang, Kuantan Pahang Darul Makmur*

³ *Faculty of Science and Technology, Department of Mechanical Engineering, Tokyo University of Science, 2641 Yamazaki, Noda, Chiba 278-8510, Japan.*

Tel: +60132440319 Email: husnainoh94@gmail.com

ABSTRACT

Stress intensity factor is the main parameter in analysis of fatigue crack growth because it brings an impact to the remaining life of structures. The objective of this paper is to predict the safe and failure region of structure during the fracture process based on the SIFs. Lognormal distribution is utilised in this study to predict the SIFs along the crack front. SIFs are generated from one hundred of samples randomly in lognormal distribution by using the BootstrapS-FEM. The safe region is observed when the samples exceed the fracture toughness of 29 MPa. $\sqrt{\text{m}}$ at angle $2\phi/\pi = 0$ to 0.626 with 94% of the samples. The failure region is indicated at the angle $2\phi/\pi = 0.627$ to 1 with 6% of failure samples. The failure region is essential in this analysis to prevent from the unstable crack growth of the structure happened.

Keywords: Stress intensity factor, Lognormal distribution, Fracture toughness.

INTRODUCTION

There are many factors which contribute to the crack growth rate including size, shape, orientation and location of the flaw (Her & Lin, 2014). The factors of crack growth rate occur in a scatter presented in real data (Wilson et al., 2018). The scatter results produce variations in the crack growth parameters such as material properties and crack size. The material properties particularly influence the fatigue parameters; hence, variations of SIF and fatigue behaviour will be generated automatically. The randomness in some of the parameters reveals analysis of the surface cracks to a certain degree of complexity. The prediction of brittle fractures for surface cracks is a unique challenge to researchers (Campagnolo, Berto, Razavi, & Ayatollahi, 2017; Iwashita & Azuma, 2019). The shape, boundary conditions of the structural components, size, location, orientation of the surface cracks, and fatigue loading are taken into account in the analysis.

The SIFs of three-dimensional fatigue crack propagation must be computed along the whole crack front. The three-dimensional fatigue crack propagation analysis of surface cracks is gaining more attention from researchers (K. Shi, Cai, Chen, & Bao, 2014; Zhiping, Zesheng, & Lei, 2017). The assessment of a three-dimensional surface crack is performed during the whole process of fatigue crack growth. The crack growth relationships are applied to study the evolution of crack shapes in industrial failures.

Therefore, an analysis method is required which considers the uncertainties in the structural components in the application environment (Uusitalo, Lehtikoinen, Helle, & Myrberg, 2015). One such method is the probabilistic analysis method. In the probabilistic approach, the uncertainty can be considered with the upper and lower bounds from the generation of random samples. The uncertain parameters are involved in the analysis for instance loads, material properties and crack geometrical

aspects. The uncertainties consider in this analysis for further assuring the safety of structures. Thus, the analysis of fatigue crack growth by probabilistic approach is effective to the prediction of detection period. The prediction is important to ensure defects in the material components will not produce failure during in-service schedule and the lifetime of the material component can be estimated prior to the scheduled detection.

RESULTS AND DISCUSSION

Figure 14.1 shows the deterministic S-FEM not reaching the fracture toughness of $29 \text{ MPa}\cdot\sqrt{\text{m}}$ for Aluminium Alloy 7075-T6. The safe structure was predicted below the critical SIF in elastic region during the fracture process. Thus, the unstable crack growth did not occur based on deterministic S-FEM approach because it only produced a single prediction of SIF. Therefore, BootstrapS-FEM was utilised to predict the SIF value that possibly tend to achieve the fracture toughness. The mean BootstrapS-FEM is presented with upper and lower bounds as shown in Figure 1. The upper bound exceeds the fracture toughness thus indicating a significant probability for unstable crack growth. The failure region shown in the shaded area of Region A exceeds the fracture toughness from one part of the one hundred samples at angle $2\phi/\pi=0.627$ to 1, as depicted in Figure 14.1. Thus, a fracture of the material and the structural component could occur. A quantitative finding for the structural safety was attained with the probability of failure computation from the one hundred samples of data. Thus, the BootstrapS-FEM should be considered in an analysis since it is able to detect the possibility of unstable crack growth happening compared to deterministic S-FEM. Deterministic S-FEM just produces passive results.

Figure 14.2 shows the histogram for the SIF at $2\phi/\pi = 1$ for the tension model. The minimum and maximum stresses 23.43 and $30.91 \text{ MPa}\cdot\sqrt{\text{m}}$, respectively. The mean value is $27.35 \text{ MPa}\cdot\sqrt{\text{m}}$. The minimum and maximum deviations from the mean are 3.92 and $3.56 \text{ MPa}\cdot\sqrt{\text{m}}$, respectively. Although the range between the minimum and maximum deviations was only approximately $7.48 \text{ MPa}\cdot\sqrt{\text{m}}$ it does not produce a major uncertainty for generating one hundred samples. Even though it shows a small range of uncertainty, it is crucial when analysed with a fracture toughness. In this case, the critical SIF was $29 \text{ MPa}\cdot\sqrt{\text{m}}$. Thus, 94% of the samples were in the safe region which was below the critical SIF. Catastrophic failure would not occur in the safe region. Figure 2 also shows the failure region with 6% of failure samples. The failure region means happening of unstable crack growth. The unstable crack growth happens in material structures without warning. This prognosis cannot be generated by the deterministic method. The deterministic S-FEM approach predicts that the model is in a safe region at any angle. Figure 14.1 shows that the failure of the sample appears from an angle of $2\phi/\pi = 0.51$ to 1. This result illustrates the cruciality of the probabilistic analysis in the framework of fatigue problems.

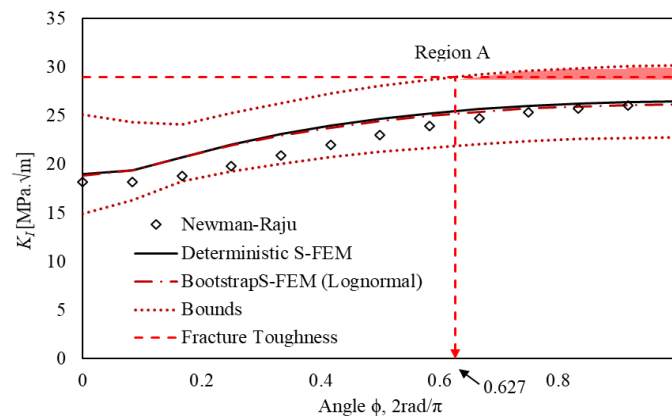


Figure 14.1: SIFs along the crack front for tension model C with upper and lower bounds

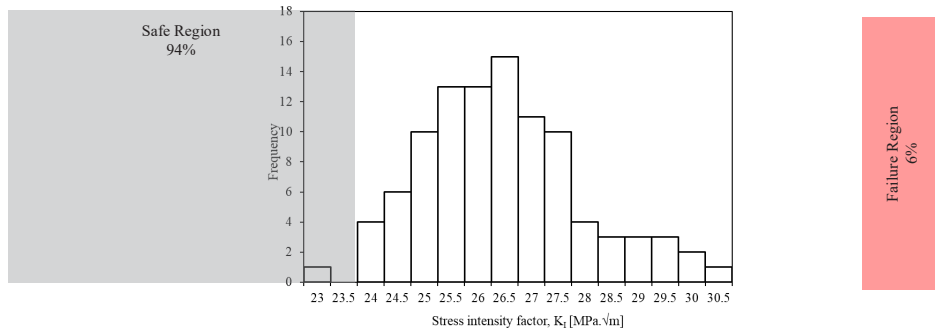


Figure 14.2: Histogram of SIFs for safe and failure region in tension model

CONCLUSION

The probabilistic analysis is useful to avoid the material structure from failure. The probability of failure was 0.06 or 6%, which surpassed the fracture toughness. This is due to the growth of fracture in the failure mode. From the 100 generation samples of SIFs, the prediction of safe samples was 0.94 or 94%, which were further examined in safe mode to prevent unstable crack propagation. Thus, an application of a bootstrap analysis in design is advisable in practical cases.

REFERENCES

- Her, S.-C., & Lin, S.-T. (2014). Non-destructive evaluation of depth of surface cracks using ultrasonic frequency analysis. *Sensors* (Basel, Switzerland), 14(9), 17146-17158.
- Wilson, D., Zheng, Z., & Dunne, F. P. E. (2018). A microstructure-sensitive driving force for crack growth. *Journal of the Mechanics and Physics of Solids*, 121, 147-174.
- Campagnolo, A., Berto, F., Razavi, S. M. J., & Ayatollahi, M. R. (2017). Some recent criteria for brittle fracture prediction under in-plane shear loading. *Procedia Structural Integrity*, 3, 110-118.
- Iwashita, T., & Azuma, K. (2019). Prediction of brittle fracture in notched specimens under cyclic loading. *Journal of Constructional Steel Research*, 162, 105721.
- Shi, K., Cai, L., Chen, L., & Bao, C. (2014). A theoretical model of semi-elliptic surface crack growth. *Chinese Journal of Aeronautics*, 27(3), 730-734.
- Zhiping, Q., Zesheng, Z., & Lei, W. (2017). *Numerical Analysis Methods of Structural Fatigue and Fracture Problems*: IntechOpen.

ACKNOWLEDGEMENT

The author would like to acknowledge the Ministry of Higher Education under Fundamental Research Grant Scheme FRGS/1/2019/TK03/UMP/02/21 (University Reference: RDU1901151) and Universiti Malaysia Pahang Al-Sultan Abdullah for financial support. Also, the authors would like to thank UMPSA for allowing the research to be conducted using High- Performance Computer (HPC).

P15

$J - \Delta\sigma$ Quantification of Out-of-plane Constraint Loss in Mode-I Fracture

N. Yusoff and F. Yusof

*School of Mechanical Engineering, Engineering Campus, Universiti Sains Malaysia, 14300 Nibong
Tebal, Penang, Malaysia*

Tel: +6045996388 Email: menorwahida@usm.my

ABSTRACT

The $J - \Delta\sigma$ approach has been shown to be able to quantify crack-tip constraint loss in high and low constraint geometries using a deformation parameter, $J/z\sigma_o$. The present study is motivated to use the parameter to assess the nature of out-of-plane constraint loss under tension loadings in a deeply-cracked compact tension (CT) specimen. The assessment focuses on the through-thickness variations of the crack-tip stress fields, which have been analysed through a full-field analysis of CT finite element models with reference to limiting two-dimensional solutions to allow the pattern of constraint loss to be recognized. The results show that the out-of-plane constraint in CT specimens is lost from a plane strain condition at the midplane to a plane stress field near the free-surface. The parameter $J/z\sigma_o$ has been shown to be able to unify the out-of-plane constraint loss in CT specimens into a single curve regardless of load and specimen thickness.

Keywords: Fracture mechanics, Out-of-plane, Crack-tip constraint, Constraint parameter, Compact tension.

INTRODUCTION

In-plane and out-of-plane constraint effects are fundamental to fracture mechanics issue on the effect of thickness on toughness. The economic pressures in structural integrity assessment have demanded ways to make use of the enhanced toughness of materials under reduction of thickness. This leads to the development of parametric approaches that focus on quantifying in-plane and out-of-plane constraint effects. In essence, the characterisation of the crack tip stress fields is directed towards quantifying the toughness of cracked materials as influenced by constraint.

A modified HRR field ($J - T_z$) was proposed to describe three-dimensional elastic-plastic crack-tip fields (Guo, 1993). Subsequent discussions on the $J - T_z$ approach (Neimitz and Galkiewicz, 2006, Hebel et al., 2007, Shlyannikov et al., 2011, Yusof and Leong, 2019) argued that the $J - T_z$ approach is limited by the dominance of the J-integral to quantify the crack tip fields. More recently, Yusof (2019) have correlated the loss of in-plane and out-of-plane constraints through a secondary term, $\Delta\sigma$, which measures the change in the maximum principal stresses at the crack tip. The change of the maximum principal stress within a micro-separation distance ahead of the crack tip is quantifiable through a deformation and lateral dimensional axis $J/z\sigma_o$. The development of the secondary constraint parameter $\Delta\sigma$ of Yusof (2019) refer essentially to crack tip stress fields ahead of the crack at $\theta = 0^\circ$ in non-hardening boundary layer formulation, bending and tension loads. In this paper, the effect of the crack tip constraint loss is further examined under tension loading in non-hardening compact tension specimens using the techniques developed by Yusof (2019).

In order to perform elastic-plastic analysis beyond small scale yielding and into full plasticity, a full-field finite element model of standard plane strain fracture toughness specimen of compact tension (CT) has been developed. To examine the effect of specimen thickness, three values of $B/(W - a) =$

1, 0.5, and 0.12 were considered, which respectively corresponded to thick, medium thickness, and thin specimens. All models were built from element type C3D20RH as implemented in Abaqus 6.12, as illustrated in Figure 15.1. The material response was idealised as elastic perfectly-plastic based on uniaxial idealisation.

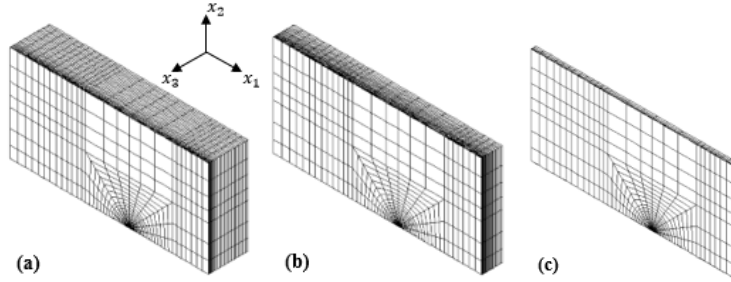


Figure 15.1: Three-dimensional full-field models of CT specimens of (a) $B/(W-a) = 1$, (b) $B/(W-a) = 0.5$, and (c) $B/(W-a) = 0.12$.

RESULTS AND DISCUSSION

The through-thickness variations of the three-dimensional crack-tip stress fields are examined with reference to the limiting two-dimensional plane strain and plane stress solutions to allow the pattern of out-of-plane constraint loss to be recognised. Based on the analysis of asymptotic stress fields (not reported here), it can be deduced that the behaviour of the out-of-plane constraint in CT specimens in full-plasticity is associated with the departure of the three-dimensional stress states from a fully-constrained plane strain condition at the midplane to a plane stress field near the free-surface through family of fields that differ hydrostatically but similar in deviatoric sense. To characterise the constraint loss, the $J - \Delta\sigma$ approach is implemented by expressing the hoop stress, $\sigma_{\theta\theta}$, and the mean stress, σ_m , at a fixed distance directly ahead of the crack front across the specimen thickness ($\theta = 0^\circ$) as a function of a normalized deformation parameter $J_{loc}/z\sigma_o$. Figure 15.2 shows the constraint loss of $\sigma_{\theta\theta}$ and σ_m as a function of $J_{loc}/z\sigma_o$ at four through-thickness sections ($z/B = 0.032, 0.1, 0.2, 0.5$) of CT specimens of three different thicknesses at radial distance $r = 2J/\sigma_o$ directly ahead of the crack front.

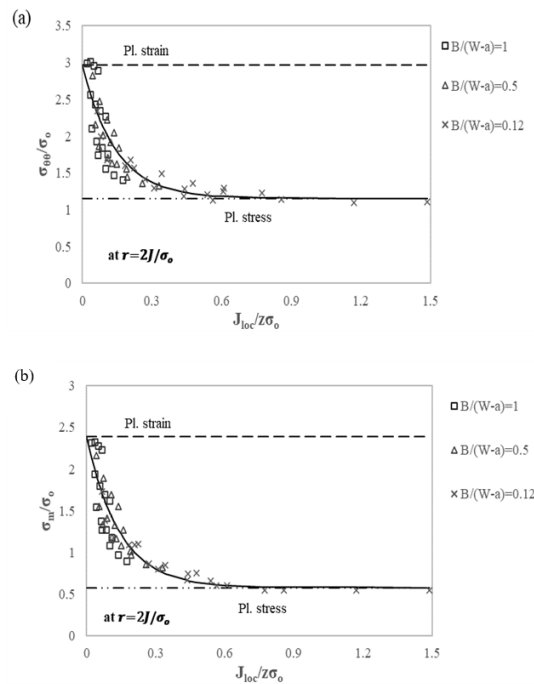


Figure 15.2: (a) $\sigma_{\theta\theta}$ and (b) σ_m as a function of $J/z\sigma_o$ at $\theta = 0^\circ$, and $r = 2J/\sigma_o$

The plots show that at a fixed distance ahead of a crack front, the out-of-plane constraint loss at different sections along the crack front of CT specimens of different thicknesses exhibits a unique behaviour that can be unified into an exponential decaying function as indicated by the solid lines. The exponential curve starts with a fully constrained plane strain value at the midplane of the thick specimen at low deformation levels. As the level of deformation increases, the constraint is lost along the crack front and the curve eventually approaches a steady state associated with a plane stress field at $J_{loc}/z\sigma_o \geq 0.5$, which is mostly exhibited by the thin specimen. The unified loci of the constraint loss as indicated by the solid line was analytically expressed by Yusof (2019) as:

$$\frac{\sigma_{ij}}{\sigma_o} = \left[\left(\frac{\sigma_{ij}}{\sigma_o} \right)_{T,Q}^{pl.strain} - \left(\frac{\sigma_{ij}}{\sigma_o} \right)^{pl.stress} \right] \exp \left(-\frac{Y(r,n)J_{loc}}{z\sigma_o} \right) + \left(\frac{\sigma_{ij}}{\sigma_o} \right)^{pl.stress} \quad (\theta = 0^\circ, i = j) \quad (1)$$

CONCLUSION

It can be concluded that the out-of-plane constraint loss in a CT specimen subjected to increasing deformation levels can be appropriately quantified using the $J - \Delta\sigma$ approach. It is shown that regardless of the specimen thickness, the out-of-plane constraint at different through-thickness sections can be approximated by the unified exponential locus of $J_{loc}/z\sigma_o$. Evidently, the distance-dependence of stress in three-dimensional crack-tip field is resolved when the stress is plotted as a function of $J_{loc}/z\sigma_o$. This demonstrates the ability of $J - \Delta\sigma$ approach to quantify the in-plane and out-of-plane constraint losses in three-dimensional cracked bodies as an extension to the two-parameter $J - T/Q$ theory that is restricted to plane strain conditions.

REFERENCES

- Guo, W. 1993. Elastoplastic three dimensional crack border field—I. Singular structure of the field. *Engineering Fracture Mechanics* 46: 93-104.
- Hebel, J., Hohe, J., Friedmann, V. & Siegele, D. 2007. Experimental and numerical analysis of in-plane and out-of-plane crack tip constraint characterisation by secondary fracture parameters. *International Journal of Fracture* 146: 173-188.
- Neimitz, A. & Galkiewicz, J. 2006. Fracture toughness of structural components: influence of constraint. *International Journal of Pressure Vessels and Piping* 83: 42-54.
- Shlyannikov, V. N., Boychenko, N. V. & Tartygasheva, A. M. 2011. In-plane and out-of-plane crack-tip constraint effects under biaxial nonlinear deformation. *Engineering Fracture Mechanics* 78: 1771-1783.
- Yusof, F. 2019. Three-dimensional assessments of crack tip constraint. *Theoretical and Applied Fracture Mechanics* 101: 1-16.
- Yusof, F. & Leong, K. H. 2019. Elastic-plastic J-Tz dominance in bending and tension loadings. *International Journal of Structural Integrity* 10: 644-659.

ACKNOWLEDGEMENT

The authors would like to acknowledge the financial support received from Universiti Sains Malaysia (USM) under the Short-Term Grant: 304/PMEKANIK/6315355.

Von-Mises Stress Analysis on the Faceplates of Laser-Welded Web-Core Steel Sandwich Plates

J. Xiaoxia^{1*}, G. Ningnian¹, Saffirna M. S.², Y. Gang¹, and W. Yaowei¹

¹*School of Mechanical Engineering, Ningxia University, 750021 Yinchuan, China*

²*Faculty of Mechanical and Automotive Engineering Technology, Universiti Malaysia Pahang Al-Sultan Abdullah, 26600 Pekan, Pahang, Malaysia*

³*College of Engineering, Universiti Teknologi Malaysia, 81310 Johor Bahru, Johor, Malaysia*
Email: jiangxx@nxu.edu.cn, Tel: +60389118411

ABSTRACT

The paper investigates the von-Mises stress distribution of web-core sandwich plate faceplates under yield loading. To avoid underestimating the stress with the homogenized solution, the investigation is carried out using experimental and finite element methods. Von-Mises stress is symmetrical about the loading centre along the surface of the top or bottom face-plate. The distribution of von-Mises stress, on the other hand, is not like the serrated shape. It is further demonstrated that the von-Mises stress distributions of the top and bottom faceplates are complex phenomena. The reasons for this include the geographical deflection of the faceplates, crack-like notches of the T-joint, diameter of the loading patch, span length between two supports, and geometric features of the structure. As a result, it is critical to investigate the controlling mechanism of the failure mode of laser-welded web-core sandwich plates.

Keywords: Von-Mises stress, Web-core sandwich plate, T-joint, Geometrical parameters

INTRODUCTION

The growing demand for faster, lighter, and safer marine and land transportation has heightened the need to research impactful sandwich structures. Laser welding was successfully used to manufacture the web-core steel sandwich plates due to the elevated pre-fabrication accuracy of the components and the ability to connect internal stiffeners with the face sheets from the outside (Jiang et al. 2017). The purpose of this paper is to investigate the yield stress distribution of the faceplate of a laser-welded web-core steel sandwich plate. The yield stress is being described using von-Mises stress theory. The following is the flow of this paper: First, the laser-welded steel web-core sandwich plates were subjected to overhanging three-point bending testing, followed by FE model verification (Jiang et al. 2016). Taking these findings into account, the von-Mises stress distribution of the top and bottom faceplates of the web-core sandwich plate is shown. Finally, the influence factors and recommendations for future study will be discussed.

EXPERIMENTAL AND FINITE ELEMENT MODEL

Web-Core Plate Geometrical Characteristics and Dimensions

The sandwich plate with the web-core is comprised of two flat faceplates with different thicknesses, t_f and core plate with different thicknesses, t_w . L represents the length of the sandwich plate while B represents the width of the sandwich plate. H represents the height of the sandwich plate, h_c is the height of the core-plate, and s represents the distance between two adjacent core plates. The dimensions of the sandwich plates are shown in Table 16.1. The base metal in this investigation is CCS-B, a low alloy and high strength type hull steel. This base material has a yield strength of 298 MPa and a tensile

strength of 470 MPa. The welding power required to attach the faceplate and core is 12 kW, with a speed of 2000 mm/min. Carbon dioxide is the shielding gas.

Equipment

According to ASTM C393-00 Standard test method for flexural properties of sandwich structures and GB/T 1456-2005 Performance test methods for sandwich structures, a specific test setup appropriate for simply support was created to assess the bending and shear stiffness of web-core sandwich plates. The tests were carried out on a hydraulic machine with a four-column force of 3150 kN. The span length is 936 mm, and the loading speed is 5 mm/min.

Finite element modelling

The welds in the sandwich plate model (Abaqus/CAE 6.11 standard solver) are all in the centre of each core plate, and the root gap is zero. The parts utilised in this model are CPE8R. In general, rectangular elements have an aspect ratio of about 1:2. The yield load of the testing plate was determined to be 25 KN. The analysis in this paper was carried out using stress analysis of the sandwich plate web-core.

Table 16.1: Dimensions of sandwich plates with different geometrical parameters

Parameters	L/mm	B/mm	H/mm	h_c/mm	s/mm	t_f/mm	t_w/mm
Test _{1,2,3}	1200	500	60	52	78	4	2

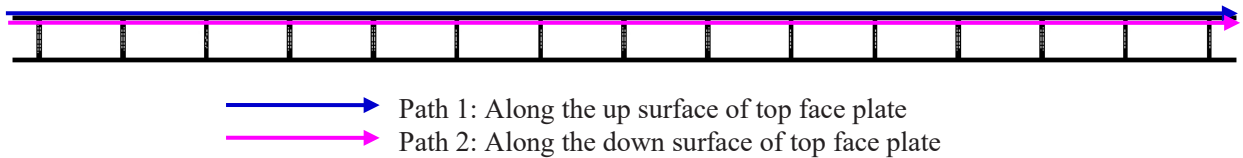


Figure 16.1 The path distribution for stress

RESULTS AND DISCUSSION

The region A on the upper surface of the top faceplate corresponds to the location of higher von-Mises stress, as shown in Figure 16.2(a). And the position where the loading is conducted is the von-Mises maximum stress point. Generally, for the solid plate under the bending loading, the smaller values of von-Mises stress, the farther away from the loading centre. Furthermore, this occurrence is unsuitable for sandwiches. Figure 16.2 shows an example (a).

Von-Mises stress is less in position B than it is in position A. The following are the four primary reasons: 1) every unit cell has location deformation because the sandwich plate is a type of frame structure plate, 2) the diameter of the loading patch and its loading position, 3) the geometrical parameters of the sandwich plate, such as the thickness of the face-plate, the height of the core-plate, and the spacing of the core-plates, and 4) the span between the two supporting points. This finding differs significantly from the HB-method mentioned by Romanoff (Romanoff, 2014).

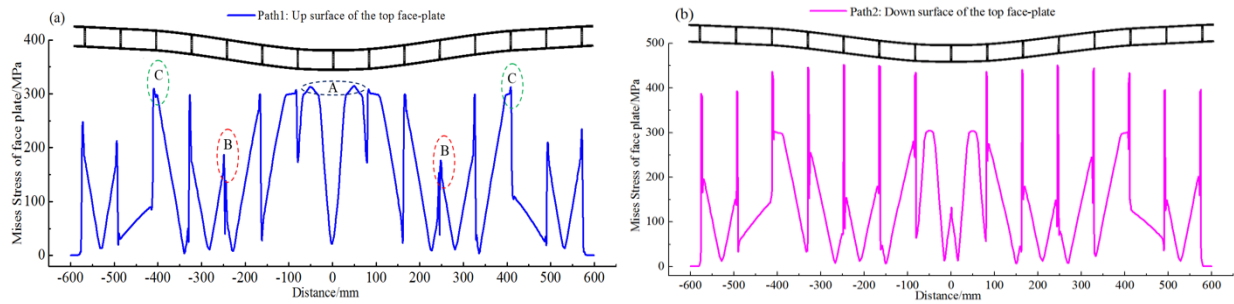


Figure 16.2: Stress distribution of sandwich plate

CONCLUSION

The following conclusions can be derived based on the experimental investigation:

1. Von-Mises stress is symmetrical about the loading centre along the surface of the top or bottom face-plate. The distribution of von-Mises stress, on the other hand, is not like the serrated shape. This finding no longer underestimates stress as the HB and FE approaches did without taking into account the laser-welded T-joint with crack-like notches.
2. It has no effect, such as the farther away from the loading centre, the smaller the von-Mises stress along the surface of the top or bottom face-plate. Four factors (e.g., location deformation, loading patch diameter and loading position, geometrical features, and the span length between two supporting points) are used to identify and explain some specific and major stress locations.

REFERENCES

- Romanoff, J., 2014. Periodic and homogenized bending response of faceplates of filled web-core sandwich beams. *Composite Structures*, 113, pp.83-88.
- Jiang, X., Zhu, L., Fei, S., Zhang, S., Hu, C., & Wang, G. 2016. Three-Point Bending Behaviour of Web-Core Steel Sandwich Panel. *International Conference on Advanced Material Science and Environmental Engineering (AMSEE 2016)*, Amsee, 149–152.
- Jiang, X. X., Ji, H., Rejab, M. R. M., Zhang, S., Ishak, M., & Zhu, L. 2017. Geometrical parameters influence on the stiffness of steel sandwich plates with web-core. *IOP Conference Series: Materials Science and Engineering*, 257(1). <https://doi.org/10.1088/1757-899X/257/1/012081>

ACKNOWLEDGEMENT

Authors acknowledge and thank Key Research and Development Plan of Ningxia Hui Autonomous Region, Ningxia, China (No.2019BDE03005) and Malaysia Ministry of Education, University Malaysia Pahang (PGRS1903160) for financial support.

P17

A New Flat Inspection Vehicle for Detecting Belt Damage

A. Li, B. Zhang, and H. Hao

School of Mechanical Engineering, NingXia University, Yinchuan, Ningxia, China

Tel: +8618209589641 Email: lah_20211017@163.com

ABSTRACT

Track inspection vehicle is the key equipment to detect belt damage in the process of belt conveyor operation. It has great significance to evaluate whether the belt can continue to be used and whether it will affect the completion of transportation work. A new flat inspection vehicle is proposed to detect belt damage. It was modelled and assembled on the SolidWorks 3D modelling platform, which reduces the production of physical prototypes. Now, the required sensors and vision system are equipped in the track inspection vehicle. It is driven by a dual hub motor, which is the first of its kind in the field. The length of the vehicle is 466 mm, the width is 488 mm, and the height is 227 mm, which shows the design concept of flat. The sheet metal processing technology is used in most parts of the vehicle, such as the chassis and shell, and the use of an integrated dual hub motor shows the design concept of lightweight. Experiments show that the inspection vehicle can run at a maximum speed of 0.2~0.7 m/s, and can complete the detection task of various belt damage types, which is placed under the belt conveyor to work.

Keywords: Belt damage, Flat inspection vehicle, Dual hub motor, Sheet metal process

INTRODUCTION

For long-distance bulk transportation, the belt conveyors are essential. There are various problems in the working process on the belt conveyor. For example, the damage to the belt will produce a skid phenomenon or will affect the quality of the conveying material. Then cause a large amount of material accumulation, leading to a production line shutdown, ultimately resulting in significant loss of economic property. In this case, the monitoring equipment is needed to replace the workers to detect and report belt damage in time. The track patrol vehicle appeared earlier than other patrol robots with other working modes, and its technology is relatively mature. It usually carries industrial cameras and sensors with various functions such as smoke, temperature, humidity, gas, etc. In general, it has the function of collecting images and various signals.

In the last century about the 80 s, some developed countries began to develop automatic inspection devices. Up to now, the detection of belt damage in long-distance bulk transportation has been accomplished by single-side suspended rail vehicles. In 2007, Shuxiang Huang et al developed a two-arm suspension inspection vehicle. In 2016, Ranchi Xinchuang Technology Co., LTD, which is located in Shenzhen, China, developed an inspection vehicle for the urban pipeline corridor. The above examples are the vehicles that are placed on one side of the equipment for inspection. Contrary to the general case, we designed a vehicle placed under the belt conveyor for inspection.

The patrol vehicle runs on two I-steel tracks placed below the belt conveyor. The dual hub motor is used to drive, which is different from the traditional stepper motor. This drive method greatly reduces the space and the mass of the device. This way has a better detection angle to collect the image information of belt damage through a very close distance. A driven wheel with a shock absorption function is equipped in the inspection vehicle, which makes it have better stability in the inspection process. The vehicle is in contact with the guide rail through two diagonal cow-eye universal wheels, which have better steering accuracy and prevent the occurrence of the jammed phenomenon.

RESULTS AND DISCUSSION

Figure 17.1 shows that the flat inspection vehicle which includes the following five systems: chassis, shell, drive system, auxiliary system, carrying machine. These five systems are described in detail below:

- 1) Chassis: The frame and chassis are integrated, which makes the device more compact and avoids the problem of loose connection due to emergency stops and emergency exercises in the process of operation. In addition, a new mechanism is used in the new design to improve the performance of the vehicle.
- 2) Shell: Using sheet metal bending process, the same as the chassis. It is made of stainless steel, light, and crashproof.
- 3) Drive system: Dual hub motor is used to drive, which contributes to a lightweight and is different from a traditional stepper motor.
- 4) Auxiliary system: The shockproof driven wheel is installed, which has the effect of cushioning, avoiding the loosening of the device caused by the turbulence during driving. The battery box installed at the bottom also uses sheet metal processing technology, which makes the installation very convenient.
- 5) Carrying machine: The carrying of an industrial camera and sensor enables the inspection vehicle to better capture images of belt damage and other relevant signals.

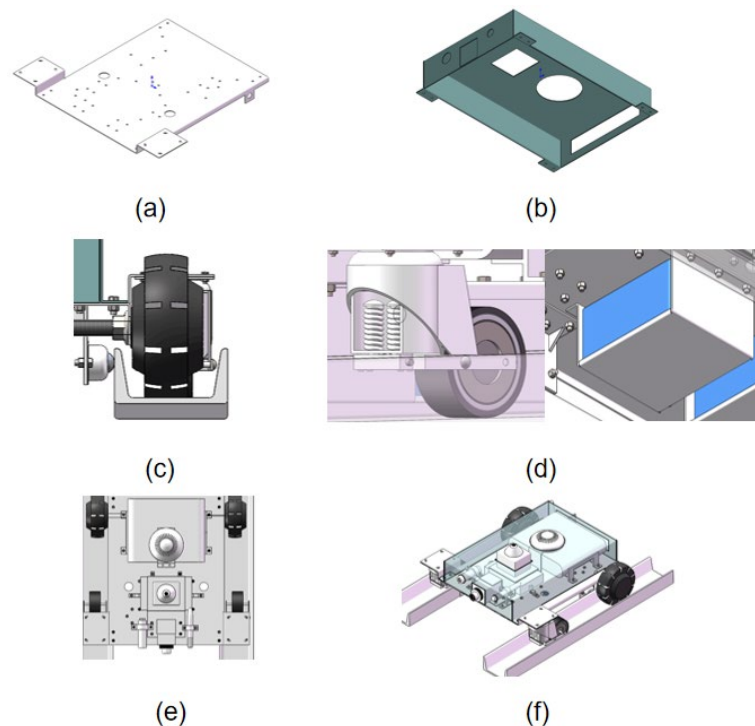


Figure 17.1: Parts of the flat inspection vehicle (a)chassis, (b) shell, (c) drive system, (d) auxiliary system, (e) carrying machine, (f) vehicle.

Figure 17.2 shows the physical model of the inspection vehicle processed from the SolidWorks 3D model. The model is not fully loaded with sensors, and currently only has an industrial camera and Industrial Computer. The inspection vehicle can be controlled by industrial computer to complete the image acquisition of belt damage.

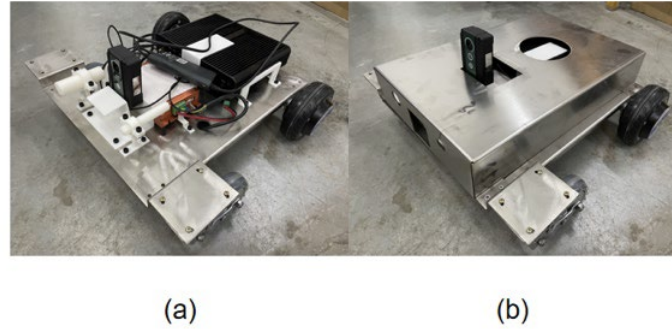


Figure 17.2: Inspection equipment installation (a) Industrial computer, (b) Industrial camera.

Figure 17.3 shows the images collected by the physical prototype in the belt damage detection experiment at the site of the belt conveyor. The collected images contain four types of belt damage, including cracks, scratches, voids, and pits. In terms of image resolution, the image resolution is higher than that of the side-mounted track patrol vehicle. This is because the flat inspection vehicle is placed at the bottom of the belt conveyor, which has a better image capture angle and distance.

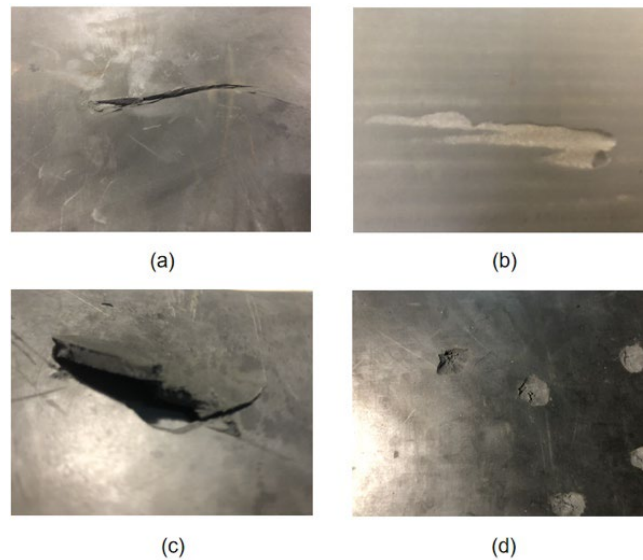


Figure 17.3: Collected images (a) crack, (b) scratch, (c) void, (d) pit

CONCLUSION

In this article, the track inspection vehicle for monitoring belt damage on the belt conveyor is designed. The study proposes a new Flat, dual hub motor driving rail inspection vehicle. Different from previous studies, the inspection vehicle is placed at the bottom of the belt conveyor to monitor belt damage. First, the vehicle was 3D modelled in SolidWorks software. Then the 3D model was built into a real object, and the physical model was used to conduct experiments on the worksite. The experiment shows that the inspection vehicle can monitor the belt damage according to the speed of 0.2~0.7 m/s. Four types of belt damage are discernible in the monitoring images. The results show that the new flat inspection vehicle can better complete the monitoring task of belt damage on the belt conveyor.

REFERENCES

- Wu, K., Zuo, Z. & Dou, S. 2018. Research and development status of China's rail inspection robot. *COMPUTER ENGINEERING & SOFTWARE*, 39(11): 80-83.
- Zhang, M., Zhou, M., Zhang, Y., Yu, Y. & Li, H. 2021. Mining Conveyor Belt Damage Detection Method based on deep learning. *Industry and Mine Automation*, 47 (6): 51-56.
- Zhou, Y., Xu, Y., Huang, Y. & Tang, C. 2021. Conveyor belt damage detection method based on improved YOLOv4. *Industry and Mine Automation*, 47(11): 61-65.

ACKNOWLEDGEMENT

I would like to express my gratitude and thanks to the University of Malaysia Pahang and Ningxia University for funding this research. And I would like to express my appreciation for my teachers, who give me a lot of guidance and help. These teachers are Professor Bo Zhang, Ms. Aili Qu, and Mr. Yaowei Yong.

P18

The Effectiveness of the Equal Load Carrying Capacity Joints Design Method Based on the Ideal Elastic-plastic Hypothesis

J. Guo^{1*}, Z. Dong² and H. Fang²

¹ School of Mechanical Engineering, Ningxia University, Yinchuan 750021, China

² State Key Laboratory of Advanced Welding and Joining, Harbin Institute of Technology, Harbin 150001, China

Tel: +8618946021636 Email: iamgjl@nxu.edu.cn

ABSTRACT

In order to further popularise the equal load carrying capacity (ELCC) design method based on the ideal elastic-plastic hypothesis, its effectiveness was verified by experiments, and the design results by different methods were compared and discussed. HSLA steel Q550D was used as the base metal, and ER70S-6 (diameter 1.2 mm) as the undermatched weld material. Tensile ELCC butt joints broke at the base metal with a load carrying capacities close to that of the base metal, bending ELCC butt joints failed at the base metal with a load carrying capacities no less than that of the base metal, which means the requirements of ELCC design had been meet. Moreover, the weld width is much smaller than that by other methods. The results demonstrate that the ELCC design based on the ideal elastic-plastic hypothesis is reasonable and feasible, can be generalized to other loads or joints.

Keywords: equal load carrying capacity, ideal elastic-plastic hypothesis, experiments, weld width.

INTRODUCTION

Developing high strength steel (HSS) is an effective strategy to reduce the weight of the structure. Equal matched and overmatched HSS joints suffer from cold cracks (Yang et al. 2020). Thus, the use of undermatched HSS joints is increasing. However, the strength of undermatched weld is insufficient, and current design methods do not recommend increasing the weld size, the only way to ensure safety is increasing the thickness of the base metal, which goes against the original intention of lightweight and severely restricts the application of HSS in welded structures. The equal load carrying capacity (ELCC) design of undermatched joints, can prevent the weld from failing ahead of the base metal and increase the load carrying capacity of the joints to that of the base metal, through enlarging the cross-sectional of the weld reasonably and optimizing the shape of the toe (Guo et al. 2019).

According to Zhao et al (2008, 2010), the ELCC theory for butt joints under static tensile can be established based on elasticity mechanics, the ELCC realisation conditions can be given in form of theoretical stress concentration coefficient, and the relationship between theoretical stress concentration coefficient and the geometry parameters of the joints can be established by numerical method. Both the finite element analysis and experimental results show that the designed joints all break at the base metal in overall yield fracture mode, which meets the requirements of ELCC design. According to Wang (2015), the ELCC theory for butt joints under static bending can be established based on elastic-plastic mechanics by parsing method, the designed joints all fail at the base metal and can bear a larger load than the base metal. Even for butt joints (the simplest joint), the methods above are still too cumbersome to be easily generalized to other loads or joints. What's more, the weld widths of the designed joints are too large, which not only increase the welding workload, but also may bring other adverse effects.

Based on the ideal elastic-plastic hypothesis, with the parameter of plastic limit loads, ELCC design is much easier (Guo et al. 2020), meaning the design method can be easily generalized to other loads or joints. However, its effectiveness has not been verified by experiments. Therefore, this paper will carry

out relevant experimental studies to verify the effectiveness of Guo's method. Moreover, the geometric parameters of ELCC butt joints designed by different methods will be compared to inspect whether Guo's method can decrease the weld width. Both tensile and bending loads are considered in the study.

RESULTS AND DISCUSSION

As shown in Figure 18.1, the tensile ELCC joints all break at the base metal with an elongation of about 16%, the tensile load carrying capacity of the joints is close to that of the base metal, which means the requirements of ELCC design have been meet. If the same optimizing shape is used at the toe, the weld width by Guo's method is much smaller than that by Zhao's method, as shown in Table 18.1. That is because the difference in ELCC realisation conditions. According to ELCC realisation condition of Zhao's method, the weld does not completely yield (only the weld root yields) when the base metal yields, while according to ELCC realisation condition of Guo's method, the weld completely yields when the base metal yields. As it turns out, the latter is sufficient, extra weld width does not make any sense.



Figure 18.1: The specimen of tensile ELCC butt joints

Table 18.1: Geometric parameters of ELCC butt joints (the groove angle is 90°)

Match Ratio μ	Reinforcement Height h/mm	Weld Width by Guo's method $2w/\text{mm}$	Weld Width by Zhao's method $2w/\text{mm}$
0.5	t	$4t$	$7.33t$
0.6	$2t/3$	$3.33t$	$6.13t$
0.7	$3t/7$	$2.86t$	$5.24t$
0.8	$t/4$	$2.50t$	$4.58t$

As shown in Figure 18.2 and Table 18.2, the bending load carrying capacity (in form of maximum bending force, F) of the bending ELCC joints is no less than that of the base metal, the requirements of ELCC design have been meet too. If the same optimising shape is used at the toe, the weld width by Guo's method is much smaller than that by Wang's method, because a great deal of base metal is treated as weld metal by Wang to enable the complex parsing processes to proceed.

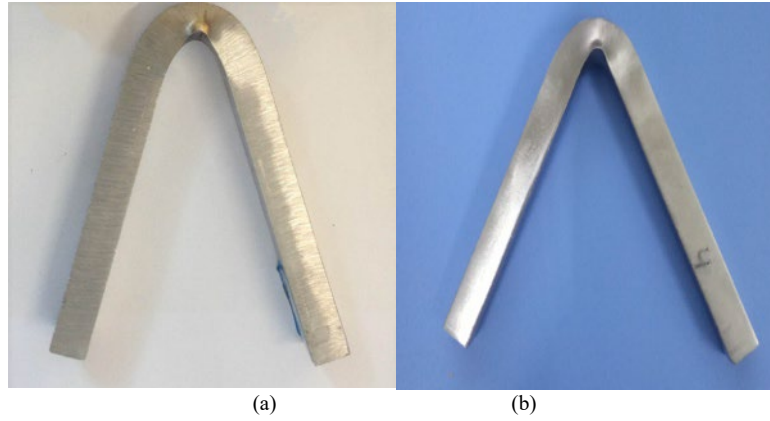


Figure 18.2: The specimen of bending tests: (a) the base metal, (b) bending ELCC butt joints

Table 18.2: Bending test results of butt joints

Sample Type	Sample No.	F (kN)	Average of F (kN)
Base Metal	1	20.35	20.57
	2	21.12	
	3	20.24	
ELCC Joints	1	22.13	21.52
	2	22.00	
	3	20.44	

CONCLUSION

ELCC butt joints were designed based on the ideal elastic-plastic hypothesis and then tested. Both tensile and bending ELCC butt joints failed at the base metal, and their load carrying capacities were close to or no less than that of the base metal, which means the requirements of ELCC design had been met. Moreover, the weld width is much smaller than that by other methods. The result of research shows that the ELCC design based on the ideal elastic-plastic hypothesis is reasonable and feasible, should be generalised to other loads or joints.

REFERENCES

- Yang J., Liu G. & Zheng W. 2020. Study on hydrogen diffusion behaviour during welding of heavy plate. *materials* 13(17).
- Guo J., Dong Z. Liu X. Wang P. & Fang H. 2019. Design of undermatching T joints based on the thought of equal load carrying capacity. *Journal of Mechanical Engineering*, 55(06):1-10.
- Zhao Z., Yang J., Fang H. & Wu F. 2008. Design of undermatching butt joint based on equal load-carrying ability. *Transactions of the China Welding Institution* 129(10): 93-96.
- Zhao Z., Fang H., Yang J. & Hu J. 2010. Design of undermatching butt Joints for equal load-carrying capacity and tensile and fatigue behaviours of joints. *Journal of Mechanical Engineering* 46(10): 75-80.
- Wang J. 2015. Design and welding with trailing shape modifying of under-matching welded joints based on principle of the equal bending load-carrying capacity. *Harbin: Harbin Institute of Technology*.
- Guo J., Dong Z. & Fang H. 2020. Equal Load Carrying Capacity Design of Butt Joints Based on Plastic Limit Loads. *IOP Conference Series Materials Science and Engineering*, 774:012008.

ACKNOWLEDGEMENT

This research received funding from the National Natural Science Foundation of China (no. 51875131).

P19

Research on Failure and Repair of Hydraulic Support Column Seal

S. Mu, K. An, and Y. M. Qiao

Ningxia University, Yinchuan 750000, China
Tel: 13895185972 Email: Musong1972@163.com

ABSTRACT

The failure causes of a group of hydraulic columns to be repaired using drum-type ring composite seal were analysed. By fitting the collected data after repair, the selection basis of sealing size after repair was found. ABAQUS analysis confirmed that the seal selected by fitting size could achieve the sealing effect. and the final size is within the range of enterprise technical requirements.

Keywords: hydraulic seal, size fitting, simulation analysis, failure analysis

INTRODUCTION

The stable work of hydraulic support is the lifeline of coal mine safety, and the hydraulic support often needs to be replaced by sealing and cylinder maintenance due to leakage. After several technical iterations, the sealing structure used in the enterprise is now composed of composite structures (Figure 19.1). Moreover, many scholars have made a more detailed explanation of the relevant aspects of such sealing structure. Jia et al. (2021) obtained the order of damage degree of column dynamic seal parts by studying the double telescopic column hydraulic support. Shi et al. (2021) analysed the influence of relevant size parameters of composite bud seal on the sealing performance of column by studying the structure of composite bud seal. Li et al. (2017) analysed the drum seal with known size, and the simulation results show that the friction coefficient has a great influence on the actual effect of the composite seal.

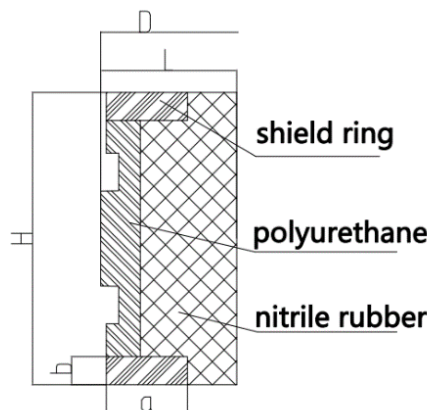


Figure 19.1: Composite seal composition

In this paper, the theoretical analysis of a group of hydraulic supports to be repaired is mainly carried out to explain some factors of column damage. The relevant data fitting and finite element analysis were carried out on how to select the seal of the repaired hydraulic support column. Finally, it is proved that the sealing size selection proposed in this paper is still effective for the column size not repaired by the enterprise.

RESULTS AND DISCUSSION

Failure analysis of composite seal

Through the analysis of the sealing structure of the repaired hydraulic support column (Figure 19.2), it is found that: 1. The white retaining ring is used to withstand the eccentric force caused by the column tilt and slow down the wear rate of the main sealing components. 2. For composite seals, the main wear component is polyurethane, while NBR has heard deformation to provide preload. 3. Because the retaining ring itself is non-closed form, there is very much dirt on the contact surface of the element. 4. Cylinders and pistons also produce partial wear and tear through prolonged use



Figure 19.2: Example of seal failure

Size fitting of drum seal structure

The main components of hydraulic support seal are composed of super elastomers. According to the data collected in the production process, the key factors affecting the seal are found and fitted into a plane (Figure 19.3). And showing strong correlation characteristics, the size selection is given by the fitting formula.

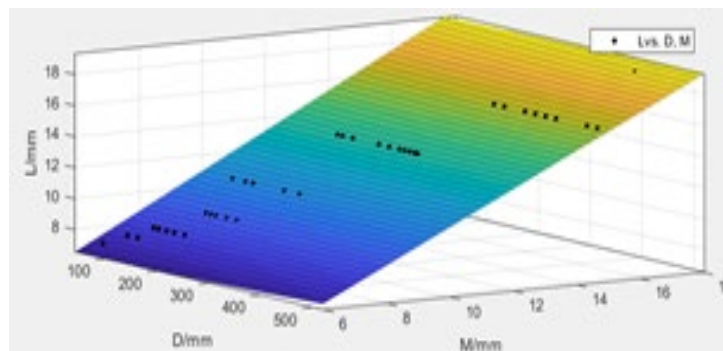


Figure 19.3: The summarised fitting results and the measured results

Establishment and Analysis of Sealing Model

The relevant dimensions of a batch of repaired columns are predicted, and the relevant parameters determining the sealing size are obtained. Bring the relevant parameters into the fitting plane. The relevant size of the seal was obtained, and the three-dimensional model can be established by combining it with the fixed size (the thickness of the model is 1 mm, indicating that the thickness of the model can be ignored). The deformation of piston groove and the installation eccentricity of seal itself are not considered. The finite element model after applying load and boundary conditions (Figure 19.4).

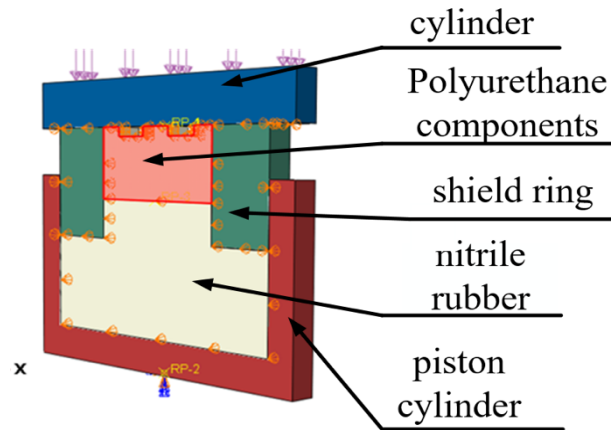


Figure 19.4: Finite element model of seal

Through the finite element analysis, it is determined that the contact stress of the main seals is greater than the normal working pressure (31.5MPa) in a certain area (Figure 19.5). According to the wear mechanism of rubber seal (Bai et al. 2022 and Wei 2018), the sealing structure can preliminarily ensure the sealing effect.

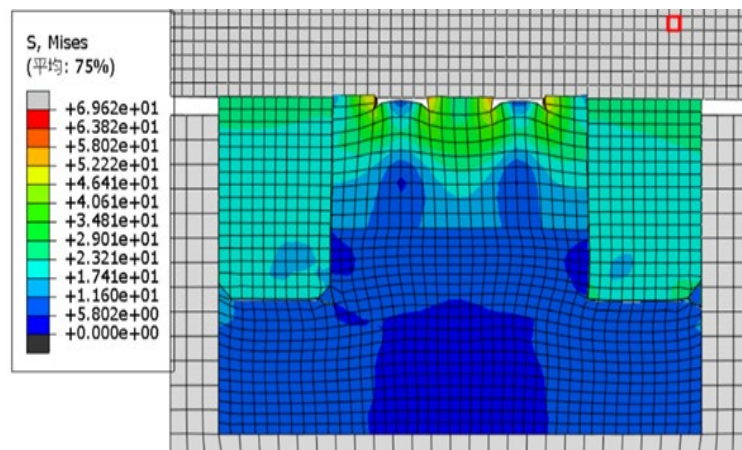


Figure 19.5: Simulation results of sealing structure

CONCLUSION

1. This paper briefly analyses four common causes of failure seals
2. Through the data collected in the enterprise production, the correlation function of sealing structure size selection is fitted, and the modelling and finite element simulation are carried out.
3. Through the analysis of the simulation results, it is proved that the sealing size obtained by fitting is effective.

REFERENCES

- Bai W., Hong L., Zhang C. & Qian Y. 2022 Dual Time-varying Correlation Degradation Modeling and Reliability Evaluation of Airborne Reciprocating Seals. *Chinese Hydraulics & Pneumatics*. (2): 31-38.
- Wei K. 2020. Research on Sealing Characteristics of Hydraulic Cylinder for Pure Water Hydraulic Support (*Master's thesis, Xi'an University of Science and Technology*).

- Jia H., Li G., Chi Z., Wang J. & Cao L. 2021. Research on Sealing Device Based on Double Telescopic Column with Large Cylinder Diameter. *Coal Mine Machinery* (03),110-114.
- Li H., Cao C. Zhou Q. & Zeng Y. 2017. Influence of Friction Coefficient on the Sealing Performance of Combined Drum ring. *Lubrication Engineering* 42(9): 118-121+133.
- Shi P., Leng J., Zao W. & Jing S. 2021. Study on structural parameters and sealing performance of hydraulic bracket column combined bulk seal, *Lubrication Engineering* 46(1): 80-85.

ACKNOWLEDGEMENT

This study was completed with the help of Ningxia University and Tiandi Ningxia Support Equipment Co., Ltd.

P20

Surface Enhanced Raman Scattering Effect of AgNPs/SiO₂ Prepared by Rapid Thermal Annealing Method

Z. Jia¹, Y. Wang², and Y. Xu¹

¹*School of Mechanical Engineering, Ningxia University, Yinchuan 750021, China*

²*School of Physics and Electronic-Electrical Engineering, Ningxia University, Yinchuan 750021, China*

Tel: +8618795307663 Email: yqwang_nxu@163.com, yrxu@nxu.edu.cn

ABSTRACT

Rapid thermal annealing is an advantageous method for preparation and application of plasmonic nanostructures, which offers low cost and controllable size and geometry. In this paper, surface-enhanced Raman scattering of silver nanoparticles on a quartz substrate (AgNPs/SiO₂) prepared by annealing at 900 °C is investigated. The distribution of AgNPs is characterised by thermal field emission scanning electron microscopy. It is found that the continuous silver film evolves into uniformly dispersed particles of two sizes, under the holding time of 5 mins. The size is 136±29 nm and 54±12.6 nm, respectively. The calculated absorption line of AgNPs shows a peak at 373 nm. The calculated maximum electric field enhancement factor is up to 7, which achieves surface-enhanced Raman scattering detection of crystal violet solution with content at 10⁻⁸ mol/L. It is revealed that plasmonic substrates fabricated by rapid thermal annealing are beneficial for the application of nanostructures into solar cells and trace detection.

Keywords: Localised surface plasmon (SP), Rapid thermal annealing, silver nanoparticles, Electromagnetic field, Surface-enhanced Raman scattering.

INTRODUCTION

Surface-enhanced Raman scattering originates from the localised plasmon resonances (LSPs) of metal nanostructures. Under the irradiation of light, free electrons in nanostructure show collective oscillation on the surface of metal nanostructures, called LSPs, which increases the local electromagnetic field and enhancing some optical processes, such as Raman scattering, named surface-enhanced Raman scattering (SERS) (Feng et al. 2019, Kneipp et al. 1997). Due to the high photo intensity, SERS shows potential applications in micro/nano scale characterisation, trace detection, and high-resolution imaging.

SERS substrate, as metal nanostructures, can be prepared via chemical synthesis, energy beam etching, seed-induced growth, self-assembly, rapid thermal annealing and templating methods (Jahn et al. 2016). Rapid thermal annealing is an advantageous method, which offers low cost, controllable size and geometry and stable and repeatable structures. It attracts lots of research attentions. S. P. Tiwari et al investigate the morphology change of gold film deposited on glass before and after annealing at 500 °C as well as its effect on Gd₂O₃: Er³⁺/Yb³⁺ film (Tiwari et al. 2020). Ji et al study the morphology and SERS effects of magnetron sputtering gold films annealed at several temperatures (Ji et al. 2019).

Silver nanoparticles (AgNPs) prepared by rapid thermal annealing is attractive in applications of solar panel (Xu et al. 2021, Mandal and Sharma, 2016) owing to its excellent plasmon properties as well as inexpensive preparation cost and convenient process. In this paper, AgNPs are prepared by rapid thermal annealing method with annealing temperature of 900 °C and annealing time of 5 min. The morphology of AgNPs is characterised by using thermal field emission scanning electron microscope. The enhanced electric field distribution is studied by electromagnetic simulations. A surface-enhanced

Raman experiment of gentian violet absorbed on AgNPs is carried out to investigate their plasmon enhancement.

RESULTS AND DISCUSSION

According to SEM image of AgNPs shown in Figure 20.1, silver nanoparticles distribute uniformly with two sizes, 136 ± 29 nm and 54 ± 12.6 nm, respectively, following Volmer-Weber film growth regime.

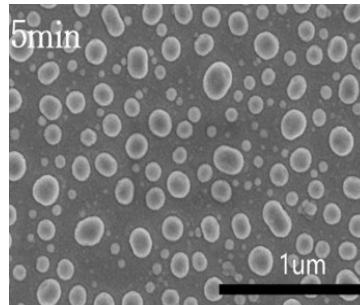


Figure 20.1: SEM image of prepared AgNPs

The calculated absorption spectrum of AgNPs (Figure 20.2) shows a peak at 373 nm, validating strong plasmonic properties in AgNPs. As shown in Figure 20.3, the maximum electromagnetic field enhancement factor can reach 7.08 around surfaces of NPs, which provides opportunities for amplifying Raman scattering.

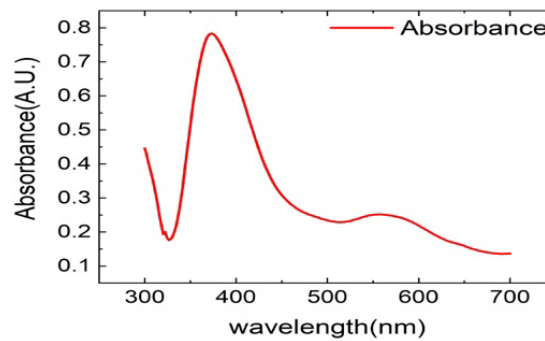


Figure 20.2: Calculated absorption spectrum of prepared AgNPs

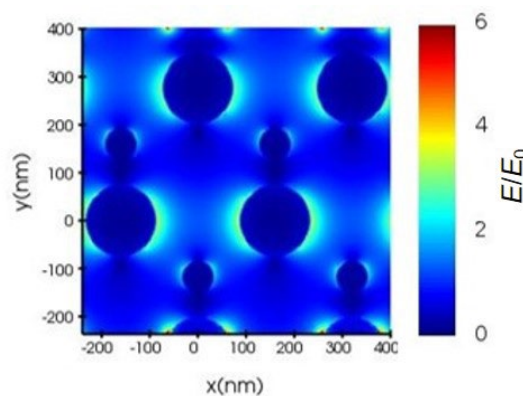


Figure 20.3: Enhanced electric field distribution around AgNPs

Surface modification treatment is made to suppress the fluorescence of AgNPs before SERS measurement. Crystal violet (CV) is selected as probed molecule. It is found that CV molecules can be detected when absorbed on our prepared AgNPs with content as low as 10^{-8} mol/L (Figure 4). In SERS spectrum of CV molecules with content at 10^{-8} mol/L some characteristic peaks can be clearly distinguished compared with that at 10^{-7} - 10^{-5} mol/L, including Raman peaks at 1619 cm^{-1} , 1587 cm^{-1} , 1535 cm^{-1} , 1371 cm^{-1} , 1175 cm^{-1} , 915 cm^{-1} , 806 cm^{-1} , 724 cm^{-1} , 442 cm^{-1} , 421 cm^{-1} , and 216 cm^{-1} . It is verified that Ag nanoparticles prepared via rapid thermal annealing method provide strong plasmon property used for trace detection by SERS.

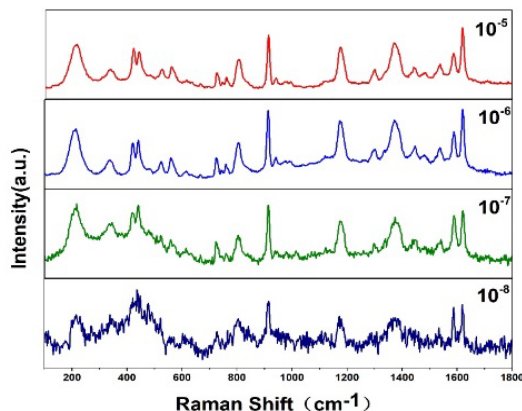


Figure 20.4: SERS spectra of crystal violet on AgNPs with content from 10^{-8} mol/L to 10^{-5} mol/L

CONCLUSION

In this paper, surface-enhanced Raman scattering effect of silver nanoparticles on a quartz substrate (AgNPs/SiO₂) prepared by annealing at 900°C is experimentally and numerically studied. Thermal field emission scanning electron microscope is used to analyse particle size distribution. It is found that the continuous silver film evolves into uniformly distributed particles of two sizes, $136\pm 29\text{ nm}$ and $54\pm 12.6\text{ nm}$, respectively, for holding time of 5 min. The calculated absorption line of AgNPs has a peak at 373 nm . The electromagnetic simulations based on experimental datum reveal that the local electric field around particles can be enhanced by as high as 7 times. Our prepared plasmon substrate can be used to detect crystal violet molecules with content as low as 10^{-8} mol/L. The results demonstrate that plasmonic substrates prepared by rapid thermal annealing is beneficial for application of nanostructures into solar cells and trace detection.

REFERENCES

- Feng, S. L., Wang, J. Y., Chen, S., Meng, L. Y., Shen, S. X. & Yang, Z. L. 2019. Surface plasmon resonance "hot spots" and near-field enhanced spectroscopy at interfaces. *Acta Physica Sinica* 68: 147801-147816.
- Jahn, M., Patze, S., Hidi, I. J., Knipper, R., Radu, A. I., MUHLIG, A., Yuksel, S., Peksa, V., Weber, K., Mayerhofer, T., Cialla-may, D. & Popp, J. 2016. Plasmonic nanostructures for surface enhanced spectroscopic methods. *Analyst* 141:756-93.
- Ji, J. 2019. Preparing surface-enhanced Raman active substrate using annealed Au thin films. *Industrial Technology Innovation* 5:50-54.
- Kneipp, K., Wang, Y., Kneipp, H., Perelman, L. T., Itzkan, I., Dasarl, R. R. & Feld, M. S. 1997. Single molecule detection using surface-enhanced raman scattering (SERS). *Physical Review Letters* 78: 1667-1670.
- Mandal, P. & Sharma, S. 2016. Progress in plasmonic solar cell efficiency improvement: A status review. *Renewable & Sustainable Energy Reviews* 65: 537-552.

- Tiwari, S. P., Kumar, A., Kumar, K., Singh, M. R., Bharti, G. P., Khare, A., Swart, H. C. & Verma, S. K. 2020. LSPR-mediated improved upconversion emission on randomly distributed gold nanoparticles array. *New Journal of Chemistry* 44: 19672-19682.
- Xu, R., Du, L., Adekoya, D., Zhang G., Zhang, S., Sun, S. & Lei, Y. 2021. Well-defined nanostructures for electrochemical energy conversion and storage. *Advanced Energy Materials* 11: 2001537.

ACKNOWLEDGEMENT

The authors would like to express their gratitude and thanks to Natural Science Foundation of Ningxia in China for funding this research (2020AAC03072, 2021AAC03059, 2021BEB04020).

Design and Simulation Analysis of Bolt Group Connection of BS-type Flange Cast Steel Right-angle Sea Valve

Z. Wang, J. Yu, Y. Ma and A. Qu

School of mechanical engineering, NingXia University, Yinchuan, China

Tel: +86 13932153006 Email: spacestarlink@126.com

ABSTRACT

BS-type flange cast steel right-angle sea valve is an important valve used to prevent media back flow in marine pipeline system. The valve and pipeline system are connected by bolt group, which bears axial load. In order to ensure the reliability and tightness of the connection, the bolt is designed by design calculation and finite element method, and the design process follows the Chinese national standard. The result shows that the bolt group connection obtained through design calculation meets the design requirements, and the finite element analysis results shows that the bolt force is the largest in the middle part of the bolt. This design method provides the design idea and research foundation for the valve connection design.

Keywords: Right-angle sea valve; Bolt group design; Finite element analysis.

INTRODUCTION

BS type flanged cast steel right Angle valve is used in the vessel pipeline system with seawater and fresh water as the medium. The valve is circular disc and the medium backflow is blocked by closing the valve. Valves and pipes are connected with flange plate, in which the bolt group connection plays a role in ensuring the safety and reliability of pipelines.

At present, there are many Chinese scholars doing research on bolt design under different working conditions. Dai Chaoqun designed the bolts on the cylinder head of diesel engine by calculating the axial force of the bolts when it reaches the lower yield point and combining the experience of using bolts in the cylinder block. Li Liu designed the anchor bolts of the storage bin by calculating the working load of the bolts under the tilting moment. Li Chang checked the tensile strength of dangerous interface of the bolt by calculating preload, and reasonably designed the bolt connection on the bullet train. Shao Wenbin designed bolt connection in automobile mounting system by comparing allowable preload and calculated maximum preload. These studies use the method of theoretical calculation to verify the reliability of bolt design by calculating the tensile strength of bolt dangerous interface. They provide the idea of theoretical calculation for bolt design.

In addition, finite element method becomes a necessary supplement to theoretical calculation. Cheng Fengwen used ComosWork to conduct finite element simulation and optimize bolt design for internal pressure vessel bolt connection. Huang Xianzhen used Abaqus to analyse the reliability of bolts applying preload, which provided a new idea for bolt design. Zhang Chi carried out finite element analysis on bolts through LS-DYNA and found the tensile and yield limits of bolts when they failed to disconnect. He provided an improved method for helicopter fuel tank to resist crash-breaking bolts. These studies verify the rationality of bolt design by means of finite element analysis. This kind of research method is closer to the real working condition and provides a more convincing method for bolt design. At the same time, this paper also provides the idea of using finite element analysis for bolt design.

The design of threaded connection in the “Mechanical Design Manual” mainly uses the principle of statics to derive the bolt force and in product design uses more empirical methods. The above theoretical design and finite element method research show that most bolt connection designs do not combine the two for bolt design. Based on the results of theoretical design and finite element analysis, this paper provides a solution for the bolt connection design of valves.

RESULTS AND DISCUSSION

We took the valve with nominal diameter of 250 mm as the research object. Through the search of the Chinese industrial standard (*CB/T 3087-1994*) of the valve, the bolt group was designed. It needs twelve M16 bolts to connect the pipeline and the valve. The arrangement of twelve bolts on the flange plate is circular.

On the valve body, the bolt group connecting the valve body and the flange plate is used to resist the axial force and ensure the sealing of the connection between the valve body and the pipeline. The bolt connection should run through the holes of the two flange plates. The semi-threaded bolt was chosen and the nut should be tightened because the bolts need a long length and the axial force is large.

The thread is coarse thread. Thickness of flat washer is 3 mm according to the Chinese national standard. The grade of nut and bolt is 5 and 4.8, respectively and the material of them is Q235. In addition, the required thickness between the two flange plates is 69 mm, so the bolt length is 70 mm. By calculation, the maximum preload is 31268 N.

After the bolt group was designed, we used two ways to verify the rationality of the design. One is theoretical calculation and the other is finite element analysis. The theoretical calculation was to calculate the tensile strength of the dangerous section of the bolt. Because the bolt is plastic material, according to the fourth strength theory and the relative formula, tensile strength of dangerous section of bolt is 270 MPa.

$$\sigma_{ca} = \frac{1.3F_0}{\frac{\pi}{4}d_1^2} \quad (1)$$

The next step is finite element analysis. At the beginning, add the required material to the engineer data.

Table 21.1: Material information

Name	Young's modulus (N/m ²)	Poisson's ratio	Density (kg/m ³)	Yield strength (N/m ²)
Fe 360A	2.10E+11	0.274	7.83E+03	2.35E+08
230-450	2.11E+11	0.311	7.83E+03	2.48E+08

Then we put the model into ANSYS Workbench to mesh and set boundary conditions. First, set the fixed support on the bonnet surface. Second, set the preload for each bolt. And the load(0.6 MPa) which is under the pressure condition of the strength experiment is applied to each surface inside the valve body.

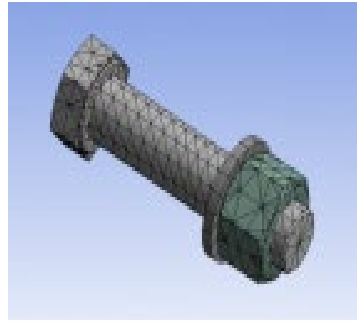


Figure 21.1: Condition of the mesh on one bolt

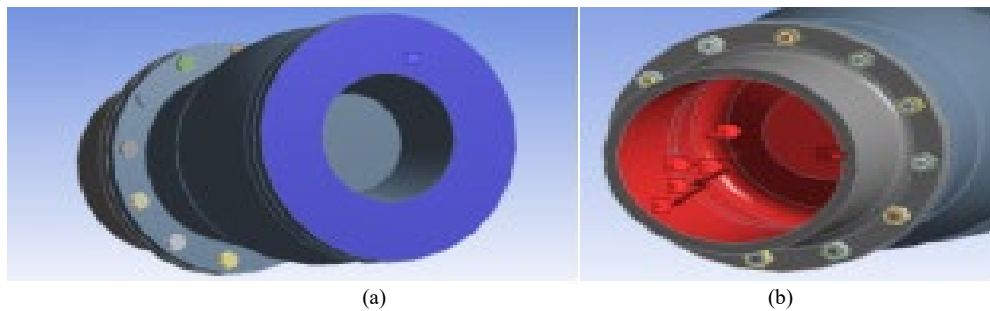


Figure 21.2: Boundary conditions:(a) Fixed support, (b) Load constraint

Finally ,we got the result. The maximum equivalent stress of bolt connection appeared in the position close to the middle of bolt and the number of equivalent stresses is between 265.97 MPa and 299.21 MPa.

CONCLUSION

The rationality of the bolt group design is verified by theoretical calculation and finite element calculation. In this design, the grade of the bolt is 4.8. The results of theoretical calculation are basically consistent with those of finite element analysis. Yield strength of bolt is 320 MPa, there is no yield deformation under the two methods. The reliability of the design can be proved. The design of this bolt group is reasonable and can be put into use.

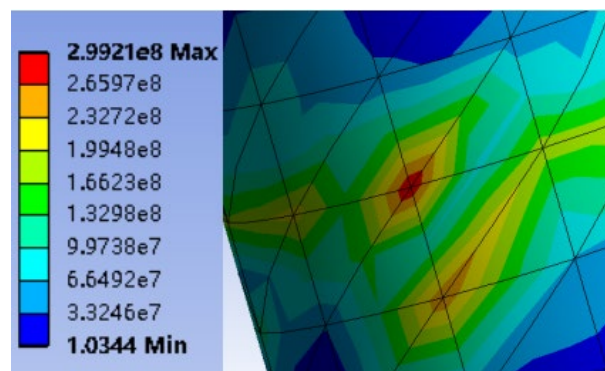


Figure 21.3: Von-Mises Stress distribution result(on one bolt)

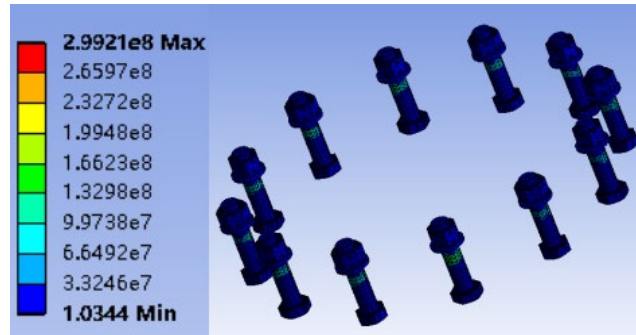


Figure 21.4: Von-Mises Stress distribution result (bolt group)

REFERENCES

- Dai, C.Q., Yao Q., Du A.Y., Jiang L.L. & Zheng X.Q. 2019. Designing of Diesel Engine Cylinder Head Bolt. *Auto Sci-tech* (04),52-56.
- Li L. & Qi C.G. 2020. Design of Anchor Bolt-Set on Belt Conveyor Loop Take Up. *Colliery Mechanical & Electrical Technology* (03),42-44.
- Li C., Zhou A.D. & Lv Z.L. 2019. Design and Research on Bolt Connection of Motor Train Unit. *Technology and Market* (07),13-15.
- Shao W.B. 2021. The Design of Bolted Joints in Powertrain Mounting System. *Automobile Applied Technology* (15),67-.
- Cheng W.F. 2008. Simulation and optimisation design by computer for connected bolt. *Modern Manufacturing Engineering* (11),44-46+123.
- Zhang C. 2019. Design and analysis of anti-falling bolt for helicopter fuel tank. *Modern Business Trade Industry* (23),201-202.

ACKNOWLEDGEMENT

Ningxia University, Demonstration course of “Ideological and Political Theories teaching in All Courses” for funding this research.

P22

Effect of 3.5% NaCl Solution on the Mechanical Properties of 7075-aluminium Alloy with RRA Regression Re-ageing

Z. Pei¹, W. Guan^{1,2}, K. Linyuan^{1,2}, W. Yan¹, Z. Zekun¹, and S. Xin¹

¹ School of Mechanical Engineering, Ningxia University, Yinchuan 750021, China

² Ningxia Key Laboratory of Computer Aided Engineering Technology for Intelligent Equipment, Ningxia University, Yinchuan 750021, China

Tel: +8613895189949 Email: belonging@163.com

ABSTRACT

The 7075-aluminium alloy is a widely used high-strength aerospace aluminium alloy. To investigate the corrosion performance of the 7075-aluminium alloy in 3.5% NaCl solution under regression re-ageing conditions. Uniaxial normal temperature tensile, digital image correlation (DIC) and scanning electron microscopy (SEM) observations were used. The corrosion behaviour of the 7075-aluminium alloy in 3.5% NaCl solution after regression re-ageing (RRA) treatment was investigated by reference to adequate regression treatment conditions, with pre-ageing at 120 °C for 24 h, regression at 190 °C for 20 min and re-ageing at 120 °C for 24 h. The results show that the corrosion time of 7075-aluminium alloy in 3.5% NaCl solution ranged from 10 to 70 hours. The peak stress decreased from 681Mpa to 647Mpa, the tensile strength decreased by 4.99% and the elongation decreased by 20.8%. DIC analysis showed that the maximum strain before fracture decreased from 0.28 to 0.21 and when corrosion proceeded to 70h, the 7075-aluminium alloy showed the significant brittle fracture. Scanning electron microscopy showed that the corrosion behaviour of the 3.5% NaCl solution occurred mainly around the precipitated phase of the 7075-aluminium alloy. As the corrosion time increased, the corroded cracks gradually increased in size and expanded.

Keywords: Regression re-ageing, mechanical properties, DIC, microscopic corrosion, 7075-aluminium alloy

INTRODUCTION

The 7075-aluminium alloy is a representative 7XXX series aluminium alloy. In practical applications, it will inevitably be exposed to corrosive environments such as salt spray, moisture and localised condensation (Pm et al. 2021). The accumulation of corrosion can seriously affect the mechanical properties and service life of the equipment. Therefore, the corrosion behaviour of aluminium alloys in 3.5% NaCl solutions under the action of typical environmental factors and the corresponding mechanisms were studied (Pu et al. 2021). It was beneficial to carry out life management of equipment and components. This will further provide reference and theoretical guidance for the prevention of failure behaviour of aluminium alloys and the adoption of reasonable protection methods.

Higher-strength and better corrosion resistance of aluminium alloys are generally contradictory. For example, an increase in Zn content improves the strength of an aluminium alloy, but at the same time decreased its corrosion resistance. As an important means of optimising the properties of 7075 series aluminium alloys, heat treatment plays an important role in improving the corrosion resistance of the alloy. T6 peak age treated aluminium alloys have high strength but are more sensitive to corrosion. T7 over-aged aluminium alloys have good corrosion resistance but reduced strength. To achieve high strength and good corrosion resistance at the same time, attempts have been made to develop new heat treatment processes. An example is the RRA treatment (Nandana et al. 2018), which maintains strength comparable to the T6 condition but with improved corrosion resistance. The regressive re-ageing (RRA)

treatment is a relatively complex three-stage ageing process that was divided into three stages: pre-ageing, regression and re-ageing (Ozer et al. 2017). The characteristics of these three stages are used to maintain the high mechanical properties of the alloy while at the same time having good corrosion resistance.

The effects of RRA on the mechanical properties and corrosion behaviour of 7075 after corrosion in 3.5% NaCl solutions are not yet well understood. Effective corrosion control methods are also not yet well understood. Therefore, it is important to carry out research on the mechanical property changes and corrosion behaviour of 7075-aluminium alloys after corrosion by regression re-ageing.

RESULTS AND DISCUSSION

Figure 22.1 shows a specimen in uniaxial tension. The true stress-strain curve that can be obtained from the tensile test is shown in Figure 22.2. It demonstrates that the peak stress, yield strength and elongation of the material decrease as the corrosion time increases. The specific trends and quantitative effects of these need to be investigated further in the analysis department. The strain history for uniaxial tension of the 7075-aluminium alloy can be obtained by analysis of the DIC technique as shown in Figure 22.3. The figure allows for the quantification of the fracture mechanics characteristics of the studied material, including important information such as fracture strain, instability strain and fracture strain distribution. The technical steps of the RRA heat treatment of the specimens are shown in Figure 22.4.

Specifically, we need to continue to further investigate the strain history of the 7075-aluminium alloy after corrosion. In the next step in this work, microscopic corrosion changes also need to be analysed by SEM.

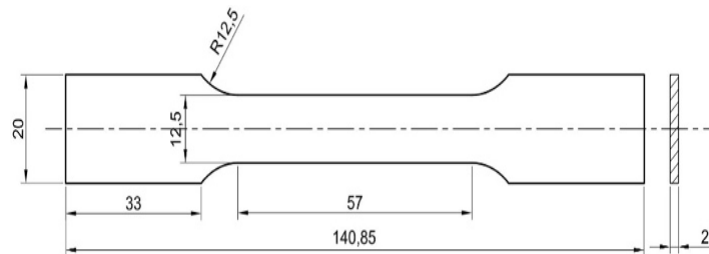


Figure 22.1: Schematic diagram of the outline and size of uniaxial thermal tensile test specimens (unit: mm)

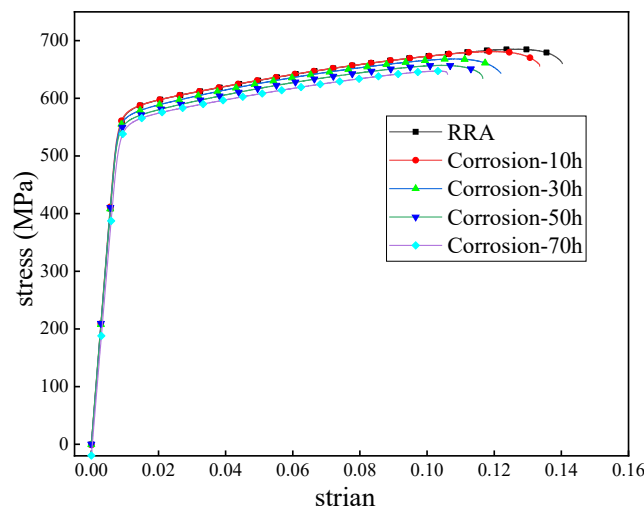


Figure 22.2: True stress-strain for different corrosion times of 3.5% NaCl solutions

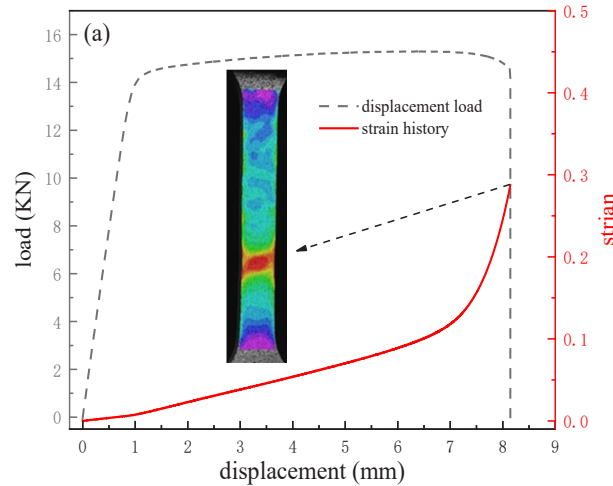


Figure 22.3: Load and strain history of the 7075-aluminium alloy after RRA heat treatment

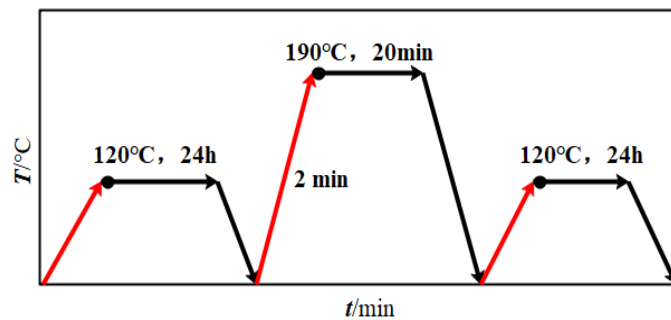


Figure 22.4: Ageing schemes of RRA treatment

CONCLUSION

The results show that the corrosion time of 7075-aluminium alloy in 3.5% NaCl solution ranged from 10 to 70 hours. The peak stress decreased from 681Mpa to 647Mpa, the tensile strength decreased by 4.99% and the elongation decreased by 20.8%. DIC analysis showed that the maximum strain before fracture decreased from 0.28 to 0.21 and when corrosion proceeded to 70h, the 7075-aluminium alloy showed the significant brittle fracture. Scanning electron microscopy showed that the corrosion behaviour of the 3.5% NaCl solution occurred mainly around the precipitated phase of the 7075-aluminium alloy. As the corrosion time increased, the corroded cracks gradually increased in size and expanded.

REFERENCES

- Pm, A. , Rkn, A. , Rk, B. , Pp, A. , Rkbn, A. , & Skg, A. 2021. Effect of post processing heat treatment on friction stir welded/processed aluminium based alloys and composites - sciencedirect. *CIRP Journal of Manufacturing Science and Technology* 35, 96-105.
- Pu, J. , Zhang, Y. , Zhang, X. , Yuan, X. , & Jin, Z. 2021. Mapping the fretting corrosion behaviours of 6082 aluminium alloy in 3.5% nacl solution. *Wear* 203975: 482-483,
- Ozer G, Karaaslan A. 2017. Properties of AA7075 aluminium alloy in aging and retrogression and reaging process. *Transactions of Nonferrous Metals Society of China* 27(11): 2357-2362.

Nandana M S, Bhat K U, Manjunatha C M. 2018. Effect of retrogression heat treatment time on microstructure and mechanical properties of AA7010. *Journal of Materials Engineering and Performance* 27(4): 1628-1634.

ACKNOWLEDGEMENT

The authors gratefully acknowledge research support from:(1)Project(52165020) supported by the National Natural Science Foundation of China; (2)Youth Top Talent Project of Ningxia (2020011); (3)Project (2018YFB20004000) supported by the Key Research and development Program of China.

Numerical Study of Random Corrosion Characteristics of Metal Surfaces Based on the Cellular Automata Method

Z. Zekun¹, W. Guan^{1,2*}, K. Linyuan^{1,2}, Y. Qingrui^{1,3}, Z. Pei¹, and X. Yiquan^{1,2}

¹ School of Mechanical Engineering, Ningxia University, Yinchuan 750021, China

² Ningxia Key Laboratory of Computer Aided Engineering Technology for Intelligent Equipment, Ningxia University, Yinchuan 750021, China

³ Sinopec Great Wall Energy & Chemical (Ningxia) Co.

Tel: +8613895189949 Email: belonging@163.com

ABSTRACT

Coal chemical enterprises in the production process due to corrosion of metal equipment shorten the service life of the normal production and thus affect the safety risks. Before the advent of CA numerical simulation, the main research focus was on the electrochemical and chemical aspects of corrosion, but the implementation is more complex and cannot be observed in real-time the emergence and development of the pitting process. Therefore, the establishment of a meta-cellular automata model of the corrosion evolution process can reflect the influence of complex physicochemical systems by analysing the influence of different parameters leading to the occurrence of corrosion. In this paper, we simulate the dissolution reaction on the solid surface by building an electrochemical reaction model; introducing the passivation probability to correct the chemical reaction rate; And applying this model to simulate the growth and change of pitting corrosion of metal materials and their anti-corrosion layer under damaged conditions, and established a model pitting pit growth model and diffusion under the effect of corrosion pit growth model at the same time using quantitative analysis means to analyse the impact of reaction passivation probability, chemical reaction rate, diffusion coefficient on the degree of corrosion. With this corrosion model, the corrosion morphology change characteristics similar to the real metal can be simulated.

Keywords: Metal pitting simulation, Electrochemical reaction, Cellular automata method, Numerical simulation

INTRODUCTION

Corrosion evolution modelling will simultaneously involve multi-phase problems, multi-dimensional problems, and the coupling between different physicochemical phenomena such as particle diffusion, electrochemical reactions, etc., in which the change of the elements in the corrosion process and the generation of corrosion products involves the influence of complex physical and chemical factors, some of which cannot be accurately described and have a random nature.

Before the emergence of CA numerical simulation, the main focus of research is the electrochemical and chemical aspects of corrosion. In the actual industrial production is often a variety of corrosion types mixed occurrence, interaction, and artificially accelerated corrosion experiments are more complex to implement, not only can not the real-time observation of the process of the emergence and development of corrosion pits but also cannot be in-depth to figure out the mechanism of corrosion occurrence. The numerical simulation study of corrosion using relevant mathematical models can, to a certain extent, make up for the shortcomings of traditional methods.

For the quantitative characterisation of corrosion damage there are already more classical models whose accuracy and reliability have been verified, but in the early studies of pitting growth models, some pitting growth models based on stochastic processes were developed (D.D.

MACDONALD,1992)(Pérez et al, 2017) because only a single pitting was studied and the growth of pitting was considered as a certain time-dependent process and the stochastic nature of pitting was ignored (C. Jiang et al.2018) (Xu Z et al. 2021).

In this paper, the influence of the pitting corrosion evolution process on the surface morphology of the specimen is modelled using a cellular automaton with stainless steel as the research object. The material transfer, metal dissolution and passivation processes in the metal corrosion process are simulated to investigate the effects of parameters such as electrolyte concentration, dissolution probability, and passivation probability on the evolution process of corrosion damage on the surface and depth direction of individual pits. To provide theoretical guidance and experimental reference for the safe service and design guidelines of metal equipment in coal chemical enterprises.

RESULTS AND DISCUSSION

Since the corrosion process is a complex process involving the influence of physical and chemical factors, some of which cannot be accurately described and are stochastic in nature, it is difficult to describe its evolution by solving the partial differential equation alone.

The essence of the electrochemical corrosion process on the metal surface is that it constitutes an electrochemical corrosion cell, in which an oxidation reaction occurs at the anode to produce metal ions and electrons; a reduction reaction occurs at the cathode to consume the electrons produced at the anode. The corrosion process can be defined as two processes of dissolution and passivation, respectively, according to the process can be defined CA model reaction rules as shown in Figure 23.1.

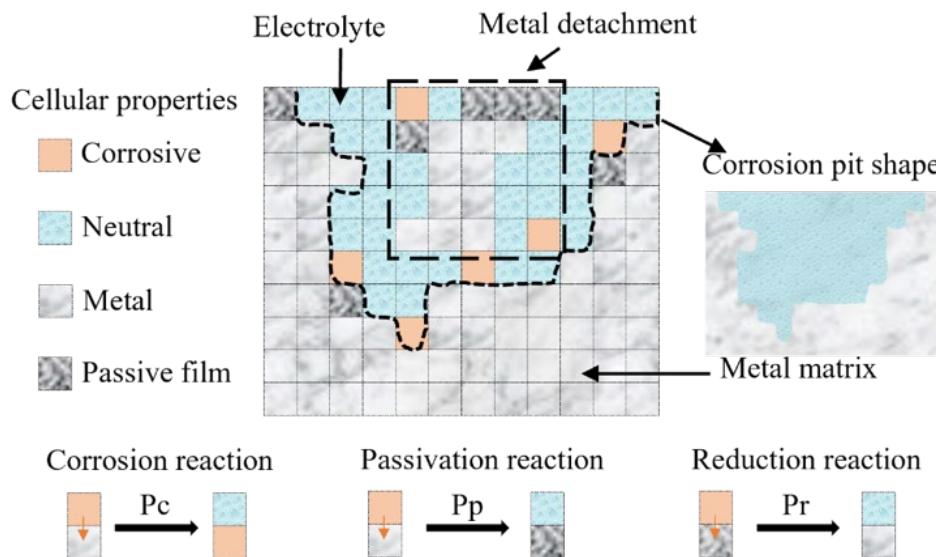


Figure 23.1: Evolutionary rules for simulation of corrosion damage by Cellular automata

Due to the autocatalytic effect of the occluded corrosion cell leads to a non-uniform medium in the hole, so the corrosion rate in the corrosion pit is characterised by spatial non-uniformity.

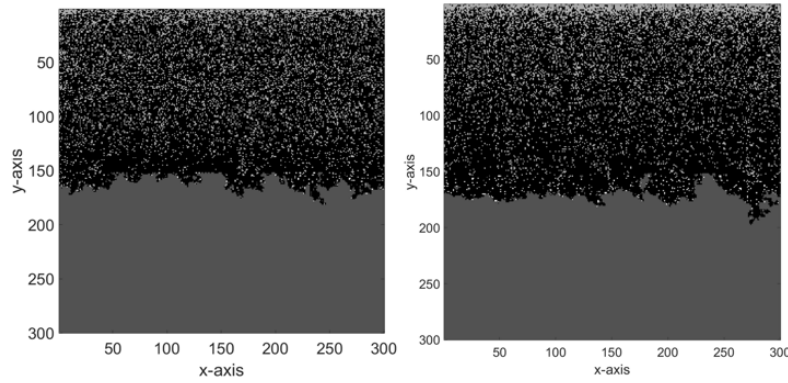


Figure 23.2: Comparison of uniform corrosion morphology of metal substrate surface in different concentrations of electrolyte

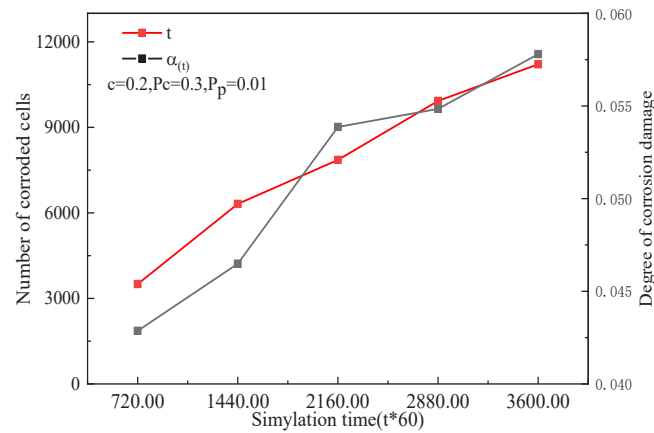


Figure 23.3: Comparison of the number of corroded cells and the amount of change in the degree of damage

CONCLUSION

The growth and changes of pitting corrosion of metallic materials and their anticorrosive layers under damage conditions were simulated using cellular automata, and a model of corrosion pit growth and diffusion under the influence of the corrosion pit growth model was established, while the effects of reaction passivation probability, chemical reaction rate, and diffusion coefficient on the degree of corrosion were analysed by means of quantitative analysis. Using this corrosion model, the corrosion morphological change characteristics similar to those of real metals can be simulated.

REFERENCES

- D.D. MACDONALD, The point-defect model for the passive state, *J. Electrochem. Soc.* 139 (1992) 3434–3449. doi:10.1149/1.2069096.
- Pérez-Brokate, Cristian Felipe, Di Caprio D, Féron, Damien, et al. Pitting corrosion modeling by means of a stochastic cellular automata-based model[J]. *Corrosion Engineering Science & Technology*, 2017:1-6.
- C. Jiang, C. Wu, X. Jiang, Experimental study on fatigue performance of corroded high-strength steel wires used in bridges, *Constr. Build. Mater.* 187 (2018) 681–690.
- Xu Z, Wei X, Lu J, et al. Simulation of corrosion behaviour of Fe–Cr–Ni alloy in binary NaCl–CaCl₂ molten salt using a cellular automata method[J]. *Solar Energy Materials and Solar Cells*, 2021, 231:111301.

ACKNOWLEDGEMENT

The authors gratefully acknowledge research support from (1) Project (52165020) supported by the National Natural Science Foundation of China; (2) Youth Top Talent Project of Ningxia (2020011); (3) Project (2018YFB20004000) supported by the Key Research and Development Program of Ningxia Province, China.

Research on Localised Corrosion and Mechanical Properties of High-strength Aluminium Alloys

W. Yan¹, W. Guan^{1,2*}, K. Linyuan^{1,2}, Z. Pei¹, W. Wenhui¹, Z. Xuejun^{1,2},
and L. Zhiwen³

¹*School of Mechanical Engineering, Ningxia University, Yinchuan 750021, China*

²*Ningxia Key Laboratory of Computer Aided Engineering Technology for Intelligent Equipment, Ningxia University, Yinchuan 750021, China*

³*School of Mechanical Engineering, University of South China, Hengyang 421001, China*
Tel: +8613895189949 Email: belonging@163.com

ABSTRACT

The local corrosion mechanism and its effect on the tensile property of 2024 aluminium alloy are investigated by immersion test, uniaxial tensile test, open circuit potential test and scanning electron microscopy. The results show that 2024 aluminium alloy exhibits pitting and intergranular corrosion after 12 h of immersion in EXCO solution. Within 1 h of immersion, the surface oxide film ruptured, accelerating the dissolution of the anodic aluminium substrate. Within 12 h of immersion, the cathode hydrogen precipitation reaction caused by a significant increase in the pH of the solution, and the corrosion rate gradually stabilised. The stress concentration due to corrosion pits resulted in a dramatic reduction in tensile strength and elongation of the specimens corroded for 12 h. The increase in immersion time intensifies the corrosion damage to the material. The tensile fracture after corrosion is ductile fracture. This study can provide a theoretical basis for the design of corrosion protection of high-strength aluminium alloys.

Keywords: corrosion, mechanical properties, aluminium alloy, fracture morphology

INTRODUCTION

2024 aluminium alloy is widely used in aerospace applications because of its high strength, low density, and good plasticity, and is also the main material for structures such as aircraft frames, skins, wing beams, and thin-walled fittings (Li et al. 2021). Affected by the service environment, aluminium alloy structures are often subject to corrosion damage. Corrosion is a destructive damage caused by a chemical or electrochemical reaction between a metal and its environment, which is an irreversible damage process that develops and accumulates with increasing service time and seriously affects the service life of the material. If corrosion is neglected, it can lead to an aircraft accident. Thus, the issue of aircraft corrosion is very important and critical. At present, not only the excellent mechanical properties of aviation aluminium alloy are required, but also the good corrosion resistance. Therefore, it is important to study the mechanism of local corrosion and its effect on the mechanical properties of the structure.

Pitting has been studied extensively. In terms of the pitting mechanism, it is now generally accepted that pitting of aluminium alloys is an electrochemical behaviour (Harlow et al. 2012). Most pitting is due to the destruction of the passivation film on the metal surface, resulting in accelerated localised metal dissolution. Pitting damage can be effectively quantified. According to Qiu & Tang (2018), the degree of corrosion damage of aluminium alloy materials can be characterised by the surface corrosion rate. There is a power function relationship between the maximum depth of pitting pits and the exposure time for AA1050 aluminium alloy, while a linear relationship between the diameter of pits and the exposure time (Surnam et al. 2017). In addition, the mechanical properties of materials after corrosion damage are also widely concerned. Corrosion damage can change the tensile properties of the material.

Hydrogen embrittlement due to corrosion and stress concentrations at the corrosion pits cause a significant reduction in the tensile strength, yield strength, and elongation of G250 mild steel (Li et al. 2018).

In summary, the existing studies mainly focus on the pitting mechanism. However, the connection between local corrosion mechanisms and mechanical properties has not been fully evaluated and understood. Therefore, the corrosion behaviour and tensile properties of 2024 aluminium alloy in EXCO (exfoliation corrosion) solution under different corrosion time were investigated by immersion test, uniaxial tensile test, electrochemical test and SEM analysis. It can provide the theoretical basis for the design of corrosion protection of aviation aluminium alloy.

RESULTS AND DISCUSSION

Figure 24.1 shows the stress-strain curves obtained from uniaxial tensile tests under different corrosion times, and corrosion changed the tensile properties. It can be seen that the tensile stress in the plastic section decreases with increasing corrosion time, and the tensile elongation also gradually decreases, indicating that the plasticity of the material continues to weaken.

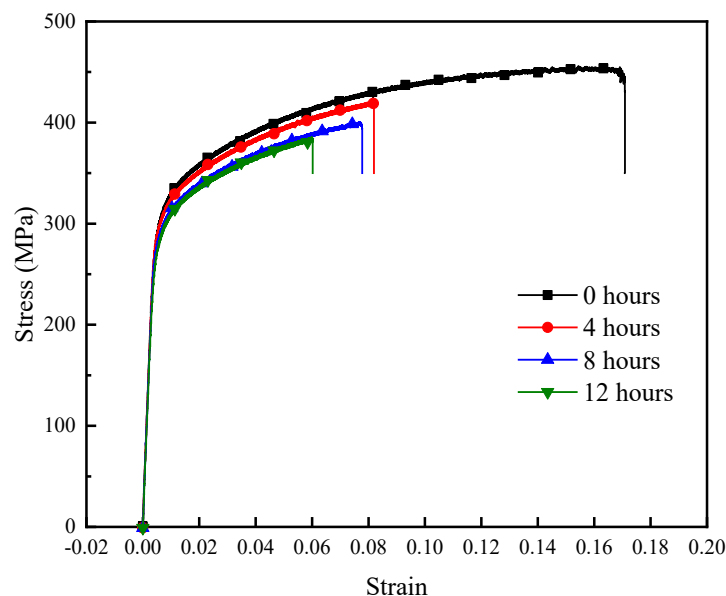


Figure 24.1: Stress-strain curve of 2024 aluminium alloy under different corrosion time

Figure 24.2 is the open circuit potential curve. It can be seen that the open circuit potential value drops rapidly from -0.667V to -0.679V after 1 h of immersion, which indicates that the protective oxide film on the surface breaks down and pitting damage occurs during this period.

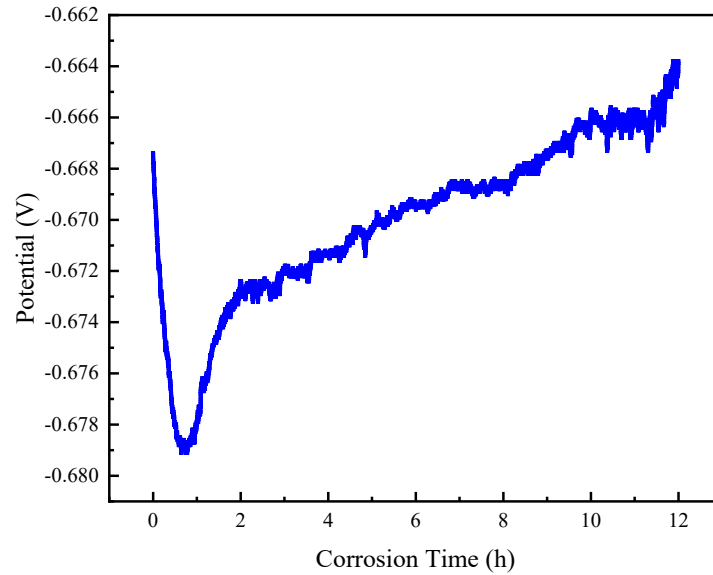


Figure 24.2: Open circuit potential curve

Figure 24.3 is the fracture morphology of corrosion specimens. It can be seen that there are loose corrosion layers of the corrosion specimen surface. The longer the corrosion time, the greater the thickness of the corrosion layer. It shows that the increase of corrosion time intensifies the corrosion damage to the material.

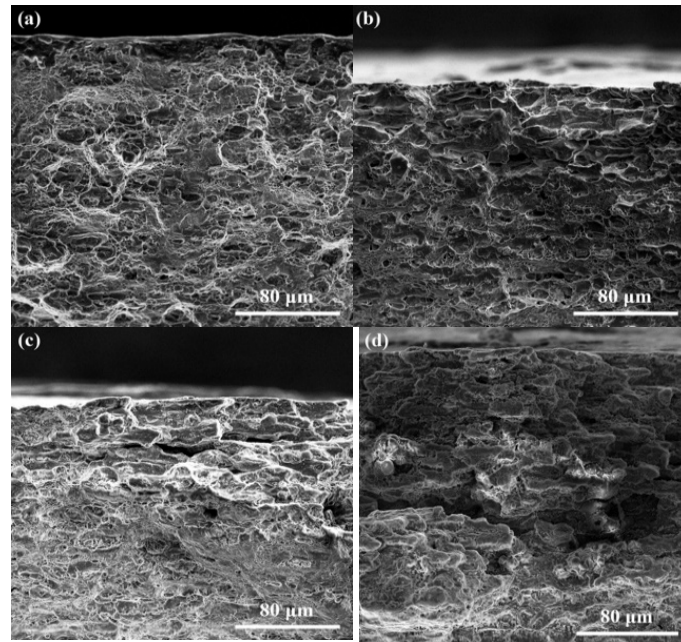


Figure 24.3: Fracture morphology of corrosion specimens: (a)0 h;(b)4 h;(c)8 h;(d)12 h

CONCLUSION

2024 aluminium alloy shows pitting and intergranular corrosion of 12 h after immersion in EXCO solution. The stress concentration due to corrosion pits resulted in a rapid decrease in tensile strength and elongation. The increase of immersion time intensifies the corrosion damage to the material. The tensile fracture after corrosion is ductile fracture.

REFERENCES

- Harlow, D. G. 2012. Constituent particle clustering and pitting corrosion. *Metallurgical and Materials Transactions* 43(8):2832-2838.
- Li, B., Dong, L.H., Wang, H.D., Zhou, Y.X. & Gao, C. 2021. Research progress on corrosion fatigue of aerospace aluminium alloy. *Surface Technology* 50(07):106-118.
- Li, L., Mahmoodian, M. & Li, C. Q. 2018. Effect of corrosion and hydrogen embrittlement on microstructure and mechanical properties of mild steel. *Construction and Building Materials* 170:78-90.
- Qiu, T., Tang, X. W., Song, J. H. & Song, H. P. 2018. Quantitative relation between corrosion damage and mechanical parameters of 2024-T4 aluminium alloy. *Nonferrous Metals Engineering* 8(04):19-24.
- Surnam, B. Y. R. & Seechurn, Y. 2017. Pitting corrosion of aluminium in the tropical climate of Mauritius. *Corrosion Engineering, Science and Technology* 52(6):411-417.

ACKNOWLEDGEMENT

The authors gratefully acknowledge research support from: (1) Project(52165020) supported by the National Natural Science Foundation of China; (2) Project(52005244) supported by the National Natural Science Foundation of China; (3) Youth Top Talent Project of Ningxia (2020011); (4) Project (2018YFB20004000) supported by the Key Research and development Program of Ningxia Province, China.

P25

Failure Analysis of Ear Socket of Gangue Retaining Bracket for Coal Mine

S. Mu, Y. Qiao , K. An, and J. Wu

School of Mechanical Engineering, Ningxia University, 755110 Yinchuan, China

Tel: 13895185972 Email: Musong1972@163.com

ABSTRACT

In this paper, the common failure forms and causes of hydraulic support used in coal mines are analysed and sorted out, and the mechanical analysis is carried out on the failure problem of the ear seat fracture at the connection between the gangue retaining plate and the top beam side protection plate in the fully mechanised mining face of the 8304-roof cutting and unloading gob-side entry retaining in a coal mine. After calculation, it is found that the actual safety factor is less than the available safety factor, which does not meet the safety standards of the industry. The finite element analysis software was used to simulate the force of the ear seat, and the stress concentration position was found. The structural improvement measures were proposed to improve the stress concentration of the ear seat, which provided effective guidance for the design and maintenance of the hydraulic support.

Keywords: Gangue retaining bracket, ear socket fracture, stress analysis

INTRODUCTION

The gangue retaining support is an important hydraulic support in gob-side entry retaining mining. It is placed on the edge of the goaf and the transportation roadway to prevent the gangue generated by the caving of the covered strata in the goaf from entering the transportation roadway and the hydraulic support, which directly affects the success of gob-side entry retaining. Therefore, it is of great significance to analyse the failure of the gangue retaining support. In this paper, the damage of hydraulic support in a coal mine is investigated and counted, and some common failure forms of hydraulic support are summarized. The causes are analysed, and the corresponding solutions are put forward.



Figure 25.1: Failure connector

In the use and maintenance process, people found that the hydraulic support column failure, structural failure, top beam failure, base failure, side plate failure. Wei Zhang (2017) and Bao-cai Ren (1995) have analysed the failure modes, and they believe that the above failures are mostly caused by unreasonable material selection, poor welding process, unreasonable structural design of structural parts, and unreasonable design of weld fracture. There is little analysis on the failure of hydraulic support connector, and only the failure of column socket and column cap is analysed. It is believed that the failure reason is that: (1) The casting column socket has poor impact toughness, and cracks and fractures occur under the impact load. (2) The unreasonable welding design makes the welding seam strength low, and the weld tearing occurs when carrying out the load. The failure analysis of the ear socket of the key connector of the gangue baffle has not been reported. Therefore, it is of great significance to the failure analysis of the ear socket.

RESULTS AND DISCUSSION

Through the mechanical theoretical calculation of the baffle ear seat, it is found that the actual safety factor is only 1.11, which is less than the safety requirements of the industry, and the fracture is possible.

In order to understand the stress condition of the connecting ear socket and find out the stress concentration position, the corresponding finite element simulation model is established, and it is found that the stress concentration position is located at the root of the ear socket, which is consistent with the actual fracture position.

In order to improve the problem of stress concentration and small safety factor, the structure and size are optimised. After calculation, the improved safety factor reaches 1.71, which meets the requirements of industrial safety. It can be seen from the stress nephograms in Figure 25.2 and Figure 25.3 that the concentrated stress at the root of the ear socket was reduced from 948 MPa to about 200 MPa, and the stress concentration at the root of the ear socket was improved, which improved the strength of the ear socket. The strength of the redesigned structural components met the operating conditions.

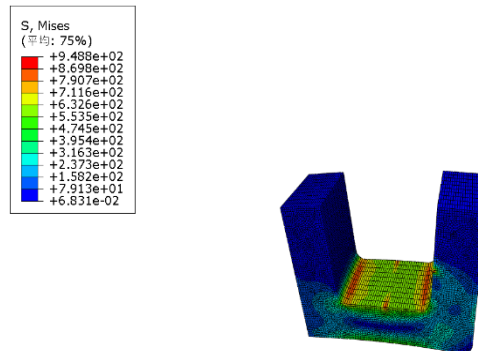


Figure 25.2 Before optimisation

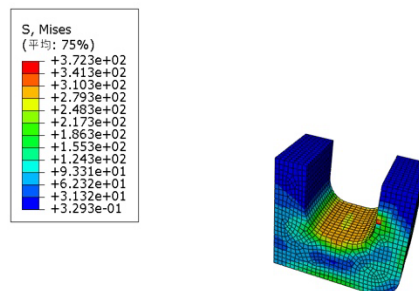


Figure 25.3 After optimisation

CONCLUSION

In this paper, through the analysis of the fracture failure of the ear seat of the gangue retaining support, it is concluded that the failure reason is due to the unreasonable size and structure design. After the improvement of the structure, the stress concentration of the ear seat is improved, and the stress concentration caused by the excessive force of the ear seat is avoided, so that the stability and strength of the ear seat are improved.

REFERENCES

- Bao-cai Ren, Zhi-gang Li .1995. Failure analysis and measures of hydraulic support structure. Zhongzhou Coal 06 .36-38
- Wei Zhang. (2017). Common failure modes and countermeasures of hydraulic supports for coal mines. Coal mining (06), 22-25.

ACKNOWLEDGEMENT

The research was completed with the support of Ningxia University and Tian di Ningxia Support Equipment Co., LT

Tension-tension Fatigue Behaviour Analysis of Glass Fiber Reinforced Polymer Composite

M. Loman¹, M. H. Zohari¹, and F. Lamin²

¹*Faculty of Mechanical & Automotive Engineering Technology, Universiti Malaysia Pahang Al-Sultan Abdullah, 26600 Pekan, Pahang, Malaysia.*

²*Vehicle Safety and Biomechanics Research Centre, Malaysian Institute of Road Safety Research, Jalan TKS 1, Taman Kajang Sentral, 43000 Kajang, Selangor, Malaysia.*

Tel: +6094246240 Email: miminorazeen@umpsa.edu.my

ABSTRACT

Fabrication of a composite structure with an acceptable consistency is a challenge, mainly due to fatigue and delamination. In this study, the fatigue behaviour of glass fiber reinforced polymer (GFRP) workpieces has been investigated under the tension-tension loading fatigue test. The GFRP workpieces were fabricated using the hand layup process and their consistency was evaluated, accordingly. They were cut from a GFRP panel which was then tested under a uniaxial static test. After the ultimate strength of the material has been determined, the fabricated GFRP workpieces undergo a fatigue test. The GFRP workpieces have been tested at several stress levels of 40%, 50%, 60%, 70%, and 80% of the ultimate strength until failure. As a result, it was revealed that the fatigue life curve of the fabricated GFRP was consistent with the previous study, thus validating the presented fabrication process.

Keywords: composite, fabrication, fatigue, glass fiber reinforced polymer, hand layup process.

INTRODUCTION

Hand layup is a process of fabricating composites using reinforcement, resin, and hardener through a manual process. It must follow the steps of putting or placing the fiber mat which is then followed by hardener and resin. The sequence is repeated for all the layers of reinforcements of the composite. Fabrication cost is a primary concern to produce larger structures while maintaining the demanding design specifications (Kim et al. 2014). It is also well aware by all that mechanical properties and performances of the composite structures are highly affected by composite manufacturing processes (Ramulu et al. 2004). Glass fiber reinforced polymer (GFRP) composite has become trending material in various applications including automotive, aerospace, turbine blade construction, and others (Stanciu et al. 2021). Considering its which that is less expensive than other types of fiber-reinforced polymer, GFRP is preferred in most applications (Jiang et al. 2021).

The popularity of composites components in modern engineering structures has tremendously increased. Therefore, it is vital to understand the behaviour of the composite materials (Movahedi-Rad et al. 2019). Its high strength is comparable to metal properties and most important is its versatility is having lightweight and good corrosion resistance. However, even if a structure made of composites is strong and durable, it will eventually be degraded by the cyclic loads applied. This phenomenon typically occurred in automotive components or civil structures. The durability and integrity of the structures will reduce by failure mechanisms, especially fatigue (Xing & Yu 2021). In specific, fiber-reinforced polymers show elastoplastic deformation upon loaded, which then changes to pure plastic deformation.

It happens when the load is increased to its ultimate strength, or the sample had almost come to its fatigue limit (Singh et al. 2021). Extensive experimental and theoretical studies need to be conducted on the fatigue behaviour of GFRP structures to understand it (Wang et al. 2020). On top of that, validation of the consistency of the fabricated composite structure is essential. This study aims to validate a hand layup GRFP fabrication process by evaluating the fatigue behaviour of tension-tension-loaded GFRP workpieces. By replicating fatigue tests at several loadings, the result was compared with previous studies to explore any significant variation.

RESULTS AND DISCUSSION

Before proceeding to the fatigue test, the GFRP workpieces have been tested using a tensile test. the objective of the static test was to determine the ultimate strength of the fabricated GFRP. Table 26.1 shows tensile properties of the fabricated GRFP. These properties are required as a pre-determined constant for the fatigue test. During the hand layup process, the glass fiber mat was layered alternately with the epoxy resin to form a composite panel as in Figure 26.1. The stacking sequences of the glass fiber mat were [0/90].

Table 26.1: Tensile properties of fabricated GFRP

<i>Tensile properties</i>	<i>Value</i>
Tensile strength (MPa)	110
Maximum load (N)	4431



Figure 26.1: GFRP panel during the fabrication process

Figure 26.2 shows the variation of the maximum fatigue load against fatigue life. The fatigue life is presented in the number of cycles to failure, N on a logarithmic scale. From the experimental result, it can be observed that the maximum number of cycles sustained by the fabricated GRFP structure varies from $1\text{E}+2$ to $1\text{E}+5$, for the maximum applied stress of 90 MPa to 40 MPa. The results were scattered in a linear trendline with a correlation coefficient R^2 of 0.8038.

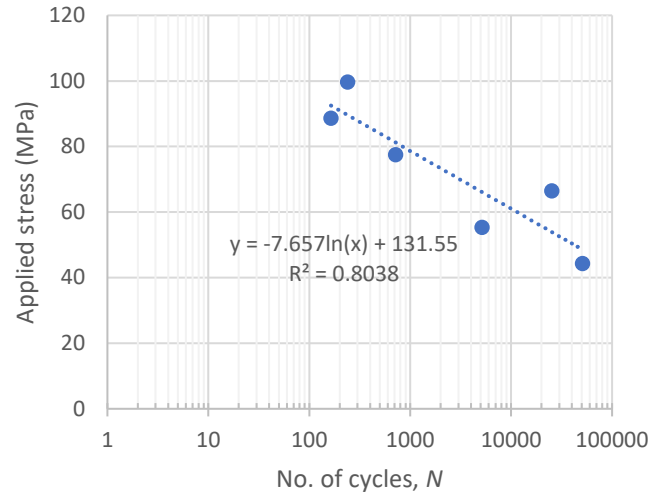


Figure 26.2 Stress life curve for fabricated GFRP.

CONCLUSION

The stress life curve of the fabricated GFRP is comparable to the previous studies. This finding validates the hand layup fabrication process. Thus, further analysis using the fabricated GRFP is reliable.

REFERENCES

- Kim, S.Y., Shim, C.S, Sturtevant, C., Kim, D.W and Song, H.C. 2014. Mechanical properties and production quality of hand-layup and vacuum infusion processed hybrid composite materials
- Ramulu, M., Stickler, P.B., McDevitt, N.S., Datar, I.P., Kim, D. and Jenkins, M.G., 2004. Influence of Processing Methods on the Tensile and Flexure Properties of High Temperature Composites. *Composite Science & Technology*, 64:1763-1772.
- Stanciu, M.D., Drăghicescu, H.T. & Rosca, I.C. 2021. Mechanical Properties of GFRPs Exposed to Tensile, Compression and Tensile–Tensile Cyclic Tests. *Polymers*, 13:898.
- Movahedi-Rad, A.V., Thomas Keller, T. & Vassilopoulos, A.P. 2019. Stress ratio effect on tension-tension fatigue behaviour of angle-ply GFRP laminates. *International Journal of Fatigue* 123:144–156.
- Xing, P. & Yu, H. 2021. Tension-tension fatigue behaviour and its modelling for life estimation of rivet-bonding CFRP/CFRP single lap joints. *Composite Structures* 275:114367.
- Jiang, Z., Wan, S., Fang, Z., & Song, A. 2021. Static and fatigue behaviours of a bolted GFRP/steel double lap joint. *Thin-Walled Structures* 158:107170.
- Singh, K.K., Ansari, M.T.A., & Azam, M.S. Fatigue life and damage evolution in woven GFRP angle ply laminates. *International Journal of Fatigue* 142:105964.
- Wang, L., Zhang, Y.W., Ho, J.C.M. & Lai, M.H. Fatigue behaviour of composite sandwich beams strengthened with GFRP stiffeners. *Engineering Structures* 214:110596.
- Ferdous, W., Manalo, A., Joshua Peauril, J., Salih, C., Reddy, K.R., Yu, P., Schubel, P. & Heyer, T. 2020. Testing and modelling the fatigue behaviour of GFRP composites – Effect of stress level, stress concentration and frequency. *Engineering Science and Technology, an International Journal* 23[5]:1223-1232.

ACKNOWLEDGEMENT

This research received financial support from the Ministry of Higher Education Malaysia under the Fundamental Research Grant Scheme FRGS/1/2019/TK03/UMP/02/7 @ RDU1901116.

Mechanical Analysis and Constitutive Modelling of Non-linear Behaviour of Ag-based Conductive Ink

S. Zulfiqar¹, A. A. Saad¹, Z. Ahmad², and Z. Bachok¹

¹*School of Mechanical Engineering, Universiti Sains Malaysia (USM)*

²*School of Materials & Mineral Resources Engineering, Universiti Sains Malaysia (USM)*

Tel: +60195186084 Email: azizsaad@usm.my

ABSTRACT

Stretchable electronic devices mainly comprised of a substrate and conductive ink. The electrical conductivity of such devices depends on the choice of conductive ink utilised in the circuit. This study focusses on the mechanical and material characterisation of silver-based conductive ink. The conductive ink is composed of a polymeric binder, cross-linking agent, organic solvent, catalysts and silver powder. Universal tensile testing is used to analyse the stress-strain behaviour of conductive ink. Moreover, the nanoindentation testing is done to observe the elastic modulus and hardness of the ink. The experimental results are then employed on the different elastic-plastic constitutive models to describe the non-linear behaviour of conductive ink.

Keywords: Silver Conductive Ink, Nanoindentation, Elastic-Plastic Modelling, Tensile Test, Material Characterisation.

INTRODUCTION

In last few decades, the stretchable and flexible electronic devices have become more popular in various mechanical and electrical applications such as mechanical sensors (Cheng et al., 2015), actuators, microfluidic circuits (Akther et al., 2020), robotics (Casanova-Moreno et al., 2017), electronic devices (S. Zulfiqar et al., 2020; S Zulfiqar et al., 2021), etc. These devices are composed of stretchable substrate and conductive ink. Conductive ink is the most important and primary part of any electronic system that can be design or fabricate into certain geometry to get the desire stretchability.

Several research have been carried out on the development of different conductive inks in which different types of conductive fillers and elastomers are utilised (Sana Zulfiqar et al., 2021). These conductive fillers include carbon-based conductors, metal-based (Au, Ag, Cu, Al, etc.) and organic conductive polymers. In this study, Ag is opted as a conductive filler in the metallic filled polymer composite ink because it possesses high electrical conductivity ($6.3 \times 10^7 \Omega^{-1} \text{ m}^{-1}$) and remains conductive after atmospheric exposure than other metallic fillers like copper and aluminium whose electrical conductivity decreases after reacting with oxygen.

The Ag-based conductive ink is obtained by mixing PDMS-OH epoxy resin with Ag powder in the presence of organic solvent toluene. The reaction is then followed by cross-linking agent and catalysts (acetic acid and dibutyltin dilaurate) at room temperature. The curing of ink is done at room temperature for 24 hours. ASTM D412 type C standard is used to make the three samples of the ink for further analysis. The structural reliability and mechanical properties of conductive ink are obtained via UTM and nanoindentation testing. Although, nanoindentation has become very popular technique of finding mechanical properties of different materials (Rouf et al., 2020). The depth sensing indentation is widely used to extract the hardness and elastic properties of specimen with known geometry of indenters and material properties.

The aim of this study is to investigate the mechanical and material characterisations of fabricated Ag-based conductive ink using UTM machine and nanoindentation. The elastic-plastic constitutive model is also applied on the experimental data to examine the non-linear behaviour of the ink under plane stress condition obtained from both the testing.

RESULTS AND DISCUSSION

The mechanical performance of conductive ink was checked by tensile testing and nanoindentation technique. The tensile test was carried out on the three dog-bone shaped samples of the ink in UTM with a crosshead speed of 10 mm/min and 10 kN load cell at room temperature. The samples were gripped at their gauge length of 25 mm and the average experimental values produced the engineering stress-strain curve of the ink as shown in Figure 27.1 (Aziz et al., 2020; Sana Zulfiqar et al., 2021).

Oliver and Pharr's method is used in nanoindentation to determine the hardness and elastic modulus based on unloading processes. Berkovich indenter is used in nanoindentation testing with an angle of $\theta = 70.3^\circ$. The basic assumption is that the contact periphery sinks in in a manner that can be described by models for indentation of a flat elastic half-space by rigid punches of simple geometry. Assuming that pile-up is negligible. The hardness and elastic modulus can be calculated as:

$$H = \frac{P_{max}}{A} \quad (1)$$

$$S = \beta \frac{2}{\sqrt{\pi}} E_{eff} \sqrt{A} \quad (2)$$

$$\frac{1}{E_{eff}} = \frac{1 - \nu^2}{E} + \frac{1 - \nu_1^2}{E_1} \quad (3)$$

Finally, the engineering stress-strain data from UTM is then converted to yield stress and plastic strain for constitutive modelling. The material parameters of the ink are obtained by implementing the multi-linear plastic models, such as isotropic and kinematic hardening models, on the stress and plastic strain data. Moreover, the same elastic-plastic model is applied on the nanoindentation data to check the non-linear behaviour of the ink.

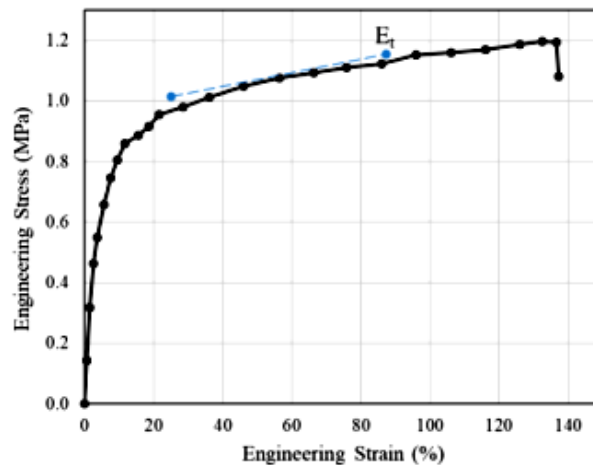


Figure 27.1: Stress-strain curve of conductive ink using UTM

CONCLUSION

The mechanical properties of conductive ink are obtained through tensile testing and nanoindentation. The stress-strain data from UTM is then utilised for material modelling in order to determine the material parameters that show the non-linearity of the ink. The multi-linear models are also applied on the elastic modulus of the ink obtained through nanoindentation. The results from both the testing reveal that the formulated conductive ink exhibits good stretchability and strength under defined conditions.

REFERENCES

- Akther, F., Yakob, S. B., Nguyen, N. T., & Ta, H. T. (2020). Surface Modification Techniques for Endothelial Cell Seeding in PDMS Microfluidic Devices. *Biosensors*, 10(11).
- Aziz, N. A., Saad, A. A., Ahmad, Z., Zulfiqar, S., Ani, F. C., & Samsudin, Z. (2020). Chapter 8 - Stress analysis of stretchable conductive polymer for electronics circuit application. In A. S. H. Makhoulf & M. Aliofkhazraei (Eds.), *Handbook of Materials Failure Analysis* (pp. 205–224). Butterworth-Heinemann.
- Casanova-Moreno, J., To, J., Yang, C. W. T., Turner, R. F. B., Bizzotto, D., & Cheung, K. C. (2017). Fabricating devices with improved adhesion between PDMS and gold-patterned glass. *Sensors and Actuators, B: Chemical*, 246, 904–909.
- Cheng, C., Yang, A., Cheng, H., & Lai, M. (2015). Development of Piezoelectric Gas Micro Pumps with the PDMS Check Valve Design (Vol. 9, Issue 7, pp. 1252–1257).
- Rouf, S., Altaf, S., Malik, S., & Najjar, K. A. (2020). Comparative Analysis Carried Out on Modern Indentation Techniques for the Measurement of Mechanical Properties: A Review. *Post-Transition Metals*.
- Zulfiqar, S., Saad, A. A., Chek, M. W., Sharif, M. F. M., Samsudin, Z., & Ali, M. Y. T. (2020). Structural and Random Vibration Analysis of LEDs Conductive Polymer Interconnections. *IOP Conference Series: Materials Science and Engineering*, 815(1).
- Zulfiqar, S., Saad, A. A., Sharif, M. F. M., Samsudin, Z., Ali, M. Y. T., Ani, F. C., Ahmad, Z., & Abdullah, M. K. (2021). Alternative manufacturing process of 3-dimensional interconnect device using thermoforming process. *Microelectronics Reliability*, 127, 114373.
- Zulfiqar, Sana, Saad, A. A., Ahmad, Z., Yusof, F., & Bachok, Z. (2021). Structural Analysis and Material Characterisation of Silver Conductive Ink for Stretchable Electronics. *International Journal of Integrated Engineering*, 13(7), 128–135.
- Islam, H. & Soares, C.G. 2018. Estimation of hydrodynamic derivatives of a container ship using PPM simulation in OpenFOAM. *Ocean Engineering* 164:414-425.

ACKNOWLEDGEMENT

This research is carried out under the research project supported via Research University Grant 1001.PMEKANIK.8014067 and Short-Term Grant 304.PMEKANIK.6315494 supported by Universiti Sains Malaysia (USM). The authors also like to acknowledge the Polymer and Rubber labs of school of materials and mineral engineering of USM for giving experimental services to pursue this study

P28

Fatigue Crack Growth Behaviour of AlSi10Mg Material Produced by Direct Metal Laser Sintering Using S-version Finite Element Method

C. F. Yung¹, S. D. Urai^{1,3}, M. S. Shaari¹, A. Takahashi² and M. A. M. Romlay¹

¹*Faculty of Mechanical and Automotive Engineering, Universiti Malaysia Pahang Al-Sultan
Abdullah,*

26600 Pekan, Pahang, Malaysia

²*Faculty of Science and Technology, Department of Mechanical Engineering, Tokyo University of
Science, 2641 Yamazaki, Noda-shi, Chiba-ken 278-8510, Japan*

³*Institut Latihan Perindustrian Kota Kinabalu, Jalan 3 KKIP Selatan Sepanggar,
88460 Kota Kinabalu, Sabah, Malaysia
Email: shamil@umpsa.edu.my*

ABSTRACT

The researchers are becoming more interested in additive manufacturing (AM). This technology has numerous advantages, particularly in the aerospace, medical, and automotive industries. Due to the limited integrity of AM materials, there is a need to research this manufacturing technology. The primary concern of engineering applications is material failure. The fatigue crack growth rate of the AM material AlSi10Mg is determined using the S-version FEM. The global-local technique was used in the S-version FEM, which consists of two distinct meshes, global and local. S-version FEM concentrated exclusively on the superimposed local crack. The Stress Intensity Factor (SIF) of the crack is calculated using the Virtual Crack Closure Method (VCCM). The SIF value is critical because it has an effect on the crack growth. The Newman & Raju equation was used to validate the S-version FEM result. It demonstrates poor agreement due to computational and meshing limitations.

Keywords: Additive Manufacturing (AM), Fatigue Crack Analysis, S-version finite element method (S-version FEM), Virtual Crack Closure Method (VCCM), Aspect ratio.

INTRODUCTION

Numerous manufacturing industries nowadays have embraced Additive Manufacturing (AM) technology in recent years. AM is a manufacturing process of adding materials instead of subtracting materials such as in conventional method, for instance, the computer numerical control (CNC) (Azri et al. 2018). When 3D printer technology is used for manufacturing, a variety of defects are found at the end of the product. These structural characteristics are dependent on the process parameters and scanning methods used to sintering Laser Metal which have a substantial effect on the material's mechanical properties, particularly its fatigue life.

Fatigue failures in engineering structures have been improved over the past few decades. This phenomenon namely deterioration occurred due to cyclic stress of fatigue which was detected before it reaches threshold. The threshold occurs at stress which is below the monotonic yield strength and researcher have to conduct many experiments to predict the relationship of fatigue with human beings safety (Akramin et al. 2020). The conventional method for determining and analysing the fatigue crack growth behaviour by performing an experiment (Haghgouyan et al. 2021). However, this method is expensive due to the cost of specimen preparation and experiment. Therefore, researchers have been switched to numerical computational which is cheaper and cost-effective.

The finite element method (FEM) is very commonly used to analyses the engineering structure. However, the FEM analysis is insufficient for analysing engineering problems due to the problem of complexity of structure and computation time. Hence, the S-version FEM is developed to enhance the computation result and overcome the obstacles faced in FEM. S-version FEM was developed by Fish 1992 (Fish 1992), and the author highlights the “S” is superimposed meshes. The stress intensity factor (SIF) can be determined by calculating the energy release rate at the crack tip by applying the virtual crack closure method (VCCM). In this research, S-version FEM is used to perform the finite element analysis on the surface crack in a prismatic bar made of AlSi10Mg material.

RESULTS AND DISCUSSION

Figure 28.1 shows a single crack growth on the rectangular bar corresponding to beach marks. The initial surface crack of this simulation is set at 2 mm for crack depth and 5 mm for crack length. These crack propagations are occurred with the uniform loading and having the same force in each step of propagation.

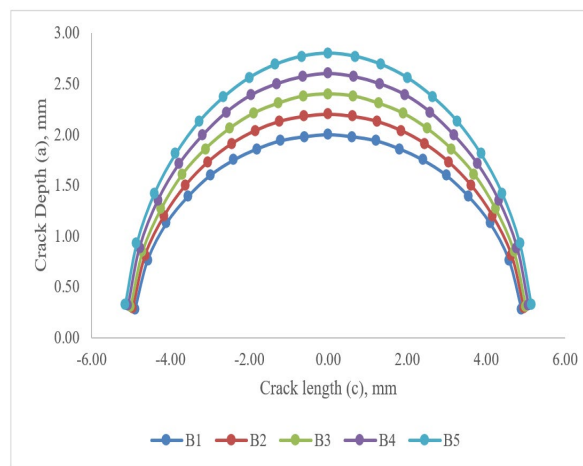


Figure 28.1: Prediction of fatigue crack growth for semi-elliptical surface crack

Figure 28.2 shows the SIF behaviour of $0.4 a/c$ according to beach marks along with the crack front. From the result, the semi-elliptical surface crack propagates uniformly due to uniform loading. The crack growth smoothly between 30° and 130° . The highest value of SIF is at 90° , which is the crack has sufficient of stress; hence it eases the propagation of crack.

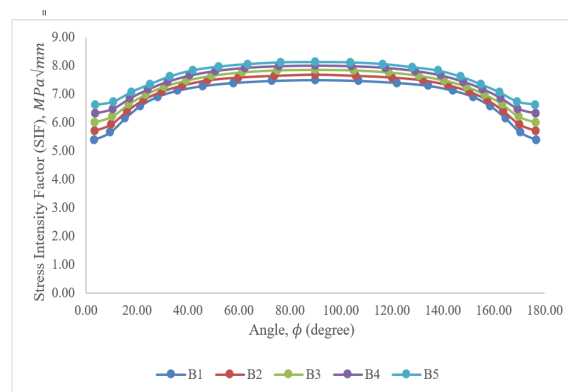


Figure 28.2: SIF behaviour according to beach mark and angle

Figure 28.3 shows the crack depth of the 1.0 a/c is the highest among the other a/c. For 0.4, 0.6, 0.8 and 1.0 a/c, the SIF trend tends to converge at 90° after the seventh beach marks. The trend for 0.2 a/c shows as in the graph is most probably due to the meshing of the element.

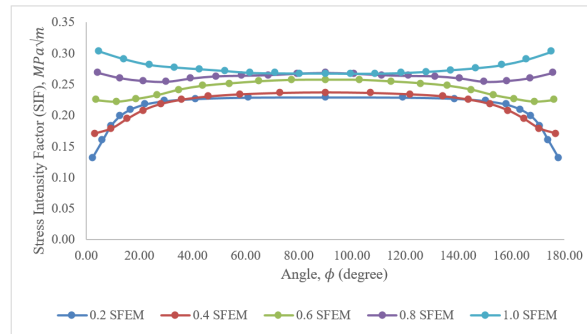


Figure 28.3: SIF corresponding with different aspect ratio

Figure 28.4 shows the result of validation between the normalised SIF obtained from S-version FEM and analytical solution. Based on the graph of 1.0 a/c shows better agreement between the S-version FEM and the analytical solution. This is due to the computational limitation to perform the simulation and the condition on the local crack mesh might affect the result. The RSME for aspect ratio of 0.2, 0.4, 0.6, 0.8 and 1.0 is 0.24, 0.17, 0.15, 0.13 and 0.16.

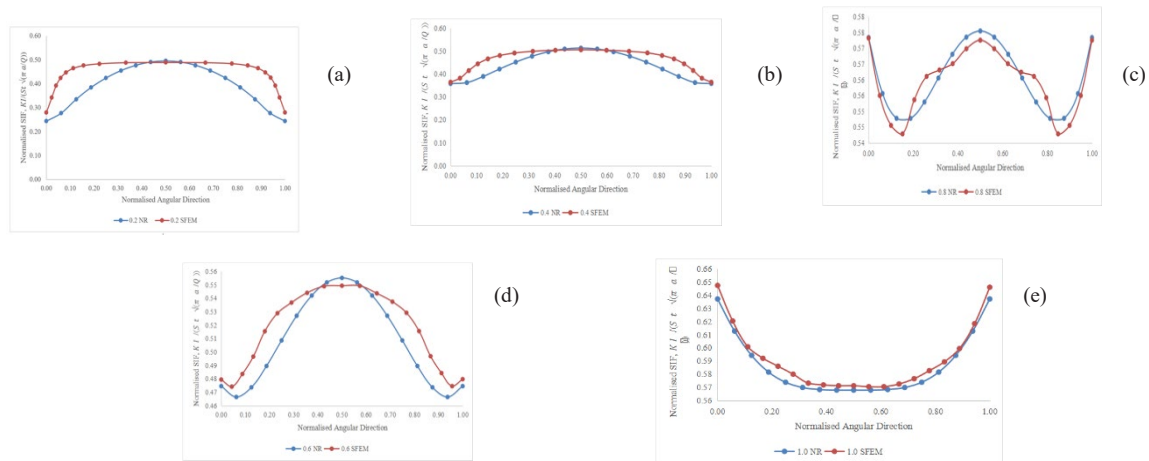


Figure 28.4: Comparison of SIF for surface crack between S-version FEM and Newman Raju 1979 corresponding different aspect ratio (a/c), (a) 0.2, (b) 0.4, (c) 0.6, (d) 0.8, (e) 1.0

CONCLUSION

In a conclusion, the prediction of fatigue crack growth for AlSi10Mg material using S-version FEM was successfully achieved.

Akramin, M. R. M., Marizi, M. S., Husnain, M. N. M. & Shamil Shaari, M. 2020. Analysis of surface crack using various crack growth models. *Journal of Physics: Conference Series* 1529(4).

Azri, M., Shaari, S., Ariffin, A.K. & Abdullah, S. 2018. Microstructure, mechanical properties and fatigue behaviour of AlSi10Mg: An additive manufacturing material. *International Journal of Engineering & Technology* 7:186–90.

Fish, J. 1992. The S-version of the finite element method. *International Journal for Numerical Methods in Engineering* 33:1081–1105.

Behrouz, H., Young, B., Picak, S., Baxevanis, T., Karaman, I. & Lagoudas, D. C. 2021. A unified description of mechanical and actuation fatigue crack growth in shape memory alloys. *Acta Materialia* 217:117155.

ACKNOWLEDGEMENT

The authors would like to acknowledge National Science and Engineering Research Council (NSERC), Ministry of Higher Education Malaysia and Universiti Malaysia Pahang Al-Sultan Abdullah for research grants and funding.

Effects on Hybridization of Interlayer Composites and Self-Reinforced Polypropylene

M. I. Ibrahim^{1,2}, M. R. M. Rejab², M. Quanjin², and M. F. Rani^{1,2}

¹*Faculty of Engineering and Technology, DRB-HICOM University of Automotive Malaysia, 26607 Pekan, Pahang, Malaysia*

²*Faculty of Mechanical & Automotive Engineering Technology, Universiti Malaysia Pahang Al-Sultan Abdullah, 26600 Pekan, Malaysia*

Tel: +60139767045 Email: ruzaimi@umpsa.edu.my

ABSTRACT

Recently, the high strength, lightweight composites has emerged as the leading candidate to substitute conventional materials in various applications. Fibre-reinforced plastic (FRP) such as carbon, glass and aramid particularly have been recognized extensively. The less commonly found hybrid composites are regarded to have a significant potential to be more flexible and ability to combine the advantages of different composites. In addition, thermoplastic composite known as self-reinforced polypropylene (SRPP) hold a unique characteristic and shall be included in the material options. In this study, five composite designs consist of various woven fibres and SRPP sheets had been prepared via hand lay-up technique. Static mechanical properties of fabricated test samples were determined through a tensile and flexural test according to the ASTM standards. The results obtained from this study revealed that the tensile strength and modulus were at their highest levels for carbon fibre reinforced plastic (CFRP), while the flexure performance displayed an increase as the hybridization of carbon/aramid fibre reinforced plastic (CAFRP) shown positive effect. However, the placement of SRPP sheets in the interlayer hybrid composite showed a reduction of tensile and flexural strength due to the bonding weakness. The current study found that SRPP-based composites produce high toughness but low stiffness characteristics, whereas other FRP composite designs indicate the features of high strength and stiffness but low elongation.

Keywords: Hybrid Composites, Carbon Fibre, Aramid Fibre, Self-reinforced polypropylene.

INTRODUCTION

Fibre-reinforced plastic (FRP) composites are known as high-strength and lightweight materials, thus contributing its wide application in defence technology and aerospace sector for many years. Latest progress in the development of FRP composites have expand its application to the other industries, such as automobiles, rail transport, construction and ship industry (Church, 2018). For example, FRP composites such as glass and carbon mainly produce significant weight savings in the transportation department without compromising specific strength and stiffness. In addition, its structural properties and durability generate a large research interest concerning energy saving and crashworthiness (Rajesh et al., 2017, Nickels, 2018).

In recent years, there has been an increasing interest in hybrid composites due to improved flexibility compared to typical FRP composites. Hybrid composites can combine a different fibre character, therefore enhance the benefits in terms of overall stiffness, load resistance, durability and material cost. For instance, hybrid composite structures have been attempted by mixing woven carbon and glass fibres in matrix of epoxy, with view of cost reduction (Hashim et al., 2017, de Oliveira Filho et al., 2019). Subsequently, there are numerous potential arrangements of combinations and variables which can lead to more exploration in this area.

In other study, thermoplastic composite known as self-reinforced polypropylene (SRPP) is another low-density material that hold a high tensile strength and high strain-to-failure characteristics. Moreover, it has the benefits of good impact strength and high fracture resistance, thus the SRPP is regarded as another potential replacements in the composite research (Tang et al., 2019).

The main aim of the present study is to investigate the effects on hybridization of interlayer composites and SRPP. Materials utilised in this study are carbon, glass and aramid fibres, in addition of SRPP as reinforcements for the purpose of determining the effects on mechanical performance. Five types of composite designs were fabricated using conventional hand lay-up technique. Finally, the specimens undergone the tensile and three-point flexural tests according to ASTM standards to define the mechanical characteristics of each design.

RESULTS AND DISCUSSION

The fractured samples of the tensile testing are displayed in Figure 29.1. Meanwhile, Figure 29.2 represents the tensile stress-strain pattern as a comparison of the average results for all the composite designs. Furthermore, the typical flexural stress-strain curve and strength for each composite designs are illustrated clearly in Figure 29.3 and Figure 29.4.

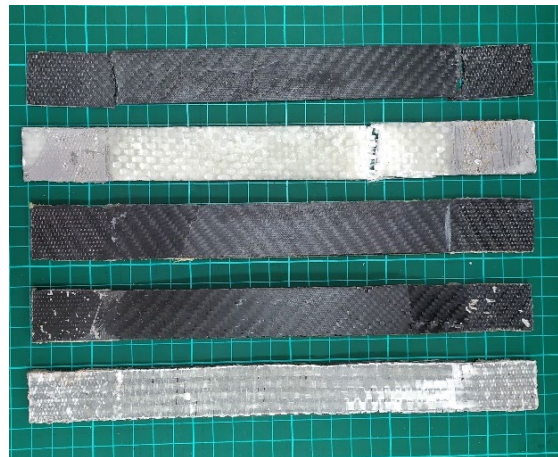


Figure 29.1: The fractured composite specimens of tensile testing. From top, CFRP, GFRP, CAFRP, CFRP/SRPP and GFRP/SRPP.

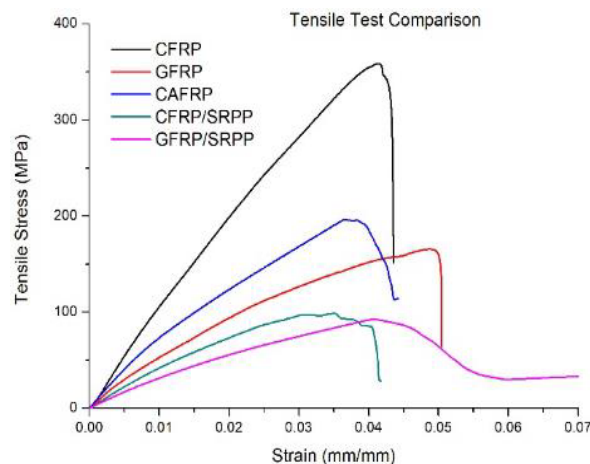


Figure 29.2: Tensile stress-strain curve results for various composite designs.

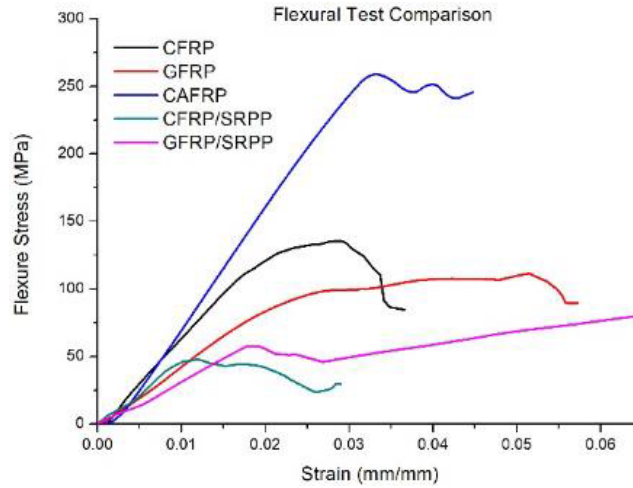


Figure 29.3: Flexural stress-strain curve results.

CONCLUSION

In conclusion, the present study succeeded to attain the tensile and flexural performance of five composite designs based on different interlayer arrangements. It is apparent from the results that interlayer hybridization significantly affects the mechanical performance of the composite. As predicted, CFRP marked the best results in tensile strength and modulus. Interestingly, the hybrid CAFRP achieved the best outcomes compared to the other types of arrangement in flexural test mode, which shows the stiffness potential of the design. However, the inclusion of SRPP sheets in the composite layers displayed a reduction of both tensile and flexural strength. The weak interlayer bonding of SRPP was detected as the main factor, even though the composites produced a higher strain performance. The following conclusions can be drawn from the present study, SRPP-based composites demonstrate high toughness but low stiffness characteristics, whereas other FRP composite designs realise high strength and stiffness but low elongation.

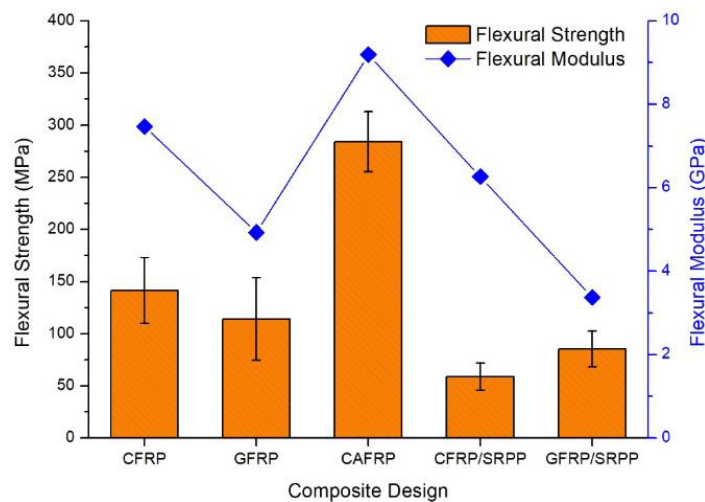


Figure 29.4: Comparison of flexural strength and modulus of the composite materials.

REFERENCES

- Church, D. 2018. A revolution in low-cost carbon fiber production. *Reinforced Plastics*, 62: 35-37.
- De Oliveira Filho, G. C., De Sousa Mota, R. C., Da Conceicao, A. C. R., Leao, M. A. & De Araujo Filho, O. O. 2019. Effects of hybridization on the mechanical properties of composites reinforced by piassava fibers tissue. *Composites Part B: Engineering*, 162: 73-79.
- Hashim, N., Majid, D. L., Uda, N., Zahari, R. & Yidris, N. 2017. Vacuum infusion method for woven carbon/Kevlar reinforced hybrid composite. *IOP Conference Series: Materials Science and Engineering*, 270.
- Nickels, L. 2018. Composites driving the auto industry. *Reinforced Plastics*, 62: 38-39.
- Rajesh, V., Rao, P. M. V. & Sateesh, N. 2017. Investigation of Carbon Composites Subjected to Different Environmental Conditions. *Materials Today: Proceedings*, 4: 3416-3421.
- Tang, J., Swolfs, Y., Aslani, A., Mencattelli, L., Bullegas, G., Pinho, S. T., Lomov, S. V. & Gorbatiikh, L. 2019. Engineering tensile behaviour of hybrid carbon fiber/self-reinforced polypropylene composites by bio-inspired fiber discontinuities. *Composites Part B: Engineering*, 178.

ACKNOWLEDGEMENT

The authors are grateful to Ministry of Higher Education for financial supports from FRGS grant and the Universiti Malaysia Pahang Al-Sultan Abdullah (UMPSA) under PGRS1703108.

P30

Mechanical Characterisation of 3/2 Fibre Metal Laminate Materials

M. F. Rani^{1,2}, M. R. M. Rejab², M. I. Ibrahim^{1,2}, and N. K. Romli²

¹*Faculty of Engineering and Technology, DRB-HICOM University of Automotive Malaysia, 26607 Pekan, Pahang, Malaysia*

²*Faculty of Mechanical & Automotive Engineering Technology, Universiti Malaysia Pahang Al-Sultan Abdullah, 26600 Pekan, Malaysia*

Tel: +60139766981 Email: ruzaimi@umpsa.edu.my

ABSTRACT

Development of lightweight materials for vehicle bodies, particularly in the automotive industry, is regarded as one of the best alternative methods for reducing fuel consumption and hazardous emissions. Weight reduction can enhance a vehicle's fuel efficiency without compromising safety strength standards. Fibre metal laminates (FML) are hybrid composite constructions comprised of thin sheets of metal alloys and plies of fibre-reinforced polymeric materials that offer outstanding mechanical qualities, including light weight, high fatigue growth resistance, and high strength and stiffness. Multi-material automobile bodies will permit optimal material selection in structural components, resulting in improved performance and reduced costs. The aims of this research is to fabricate and examine the failure behaviour of a 3/2-layer fibre metal laminate subjected to a quasi-static indentation test (QSI). Carbon fibre reinforced polymer (CFRP), glass fibre reinforced polymer (GFRP), and self-reinforced polypropylene (SRPP) were stacked with aluminium 2024-T3 to form the fibre metal laminate (SRPP). The QSI speed impact test analysis was conducted at 1 mm/min, 5 mm/min, 10 mm/min, 50 mm/min, and 100 mm/min with varying parameters. To forecast the behaviour and performance of the FML composite, the experimental performance of the specimen was compared. Variable crosshead speeds have altered the affected region of the FML, resulting in debonding of the laminate due to ongoing loading, as indicated by the test results.

Keywords: Fibre Metal Laminates, Low Velocity Impact Test, Carbon Fiber Reinforced Polymer (CFRP), Glass Fiber Reinforced Polymer (GFRP), and Self-Reinforced Polypropylene (SRPP)

INTRODUCTION

The approach of reducing the weight of the vehicle's body is one of the most significant steps taken by automakers to enhance vehicle performance, reduce fuel consumption, and provide environmental benefits. Aluminium alloy, magnesium, and several types of lightweight automobile bodies are created by combining composite material with aluminium alloys or other high-strength steels which this combination has the potential to reduce vehicle fuel consumption and emissions (Heibeck et al., 2021). Lightweight from many materials can be included into the car's body, with the aim being to employ the proper multi-material adapt at the right spot without compromising the car's intended function. By merging two or more distinct materials using the sandwiching method between aluminium and composite material specified in research, it is possible to regard it as a hybrid reinforced multi-layer material form with fibre composites as its fundamental structural components. Several alternative combinations of lamination materials can be employed in FML, taking into account parameters such as fibre type, layer thickness, number of layers, and orientation complexity. A 10% decrease in vehicle mass can enhance fuel economy by 6% to 8% and has an even greater influence on battery efficiency (Ganesarajan et al., 2022).

In the fabrication of fibre metal laminate, aluminium is frequently used for metal ply over other metals because to its corrosion-resistant mechanical properties and lightweight material. Material behaviour through FML is improved through their combination of the mechanical properties in their constituents to good ductile property, fatigue strength and impact resistance (Hynes et al., 2022). Typically for composite side, Glass reinforced aluminium laminate or GLARE is one of reinforcing material joining in FML which based on glass fibre, aramid material as aramid reinforced aluminium laminate or ARALL and another typical reinforced material applied is carbon named as carbon reinforced aluminium laminate or CARALL (Rajan et al., 2018). The matrix of the laminate damaged caused by impact and the connection between matrix-fibre to become degradation. The epoxy resin is brittle and has low resistance to the crack propagation (Romli et al., 2017).

The present research aim to analyse the impact behaviour of several fibre metal laminates under two conditional tests through low velocity impact test and quasi-static indentation test. By using different constant speed, failure mechanisms of the FML with different composite layer materials of CFRP, GFRP, SRPP and the hybrid combination of material laminate being determined. Finally, the characterise the fracture pattern of impact behaviour between the different layer will be discovered by using microstructure analysis

RESULTS AND DISCUSSION

Four types of composite material were used and test under quasi-static test. These materials were bonded to the aluminium alloys with epoxy as adhesive between the layers. As for SRPP, heating polypropylene film will act as the adhesive layer and interlayer between sheet metal and composite fibre. The preparation of the specimen was using hand layup method, as shown in the Figure 30.1, with the square dimension of 150 mm × 150 mm. The surfaces of the aluminium alloy were treated by high air pressure sandblasting technique using Al_2O_3 sand which to improve the surface texture toughness bonding between the composite fibres (CFRP, GFRP) and aluminium alloy sheet metal. The type of the aluminium alloy used in this research is aluminium 2024-T3. Figure 30.2 demonstrate the graph of load against deflection for 3/2 aluminium/ CFRP and Figure 30.3 show the fracture cross section on the GFRP sample.

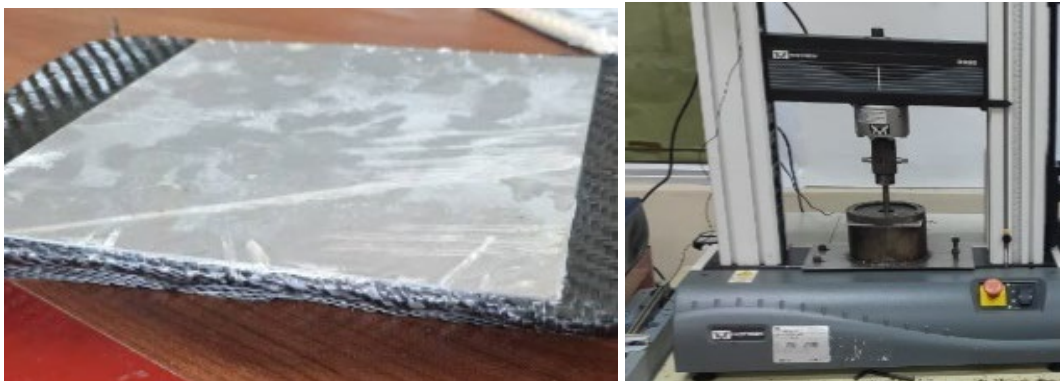


Figure 30.1: Sample of 3/2 FML CFRP (left) and Compression Test Machine Type Instron 3369 Universal Testing Machine (right)

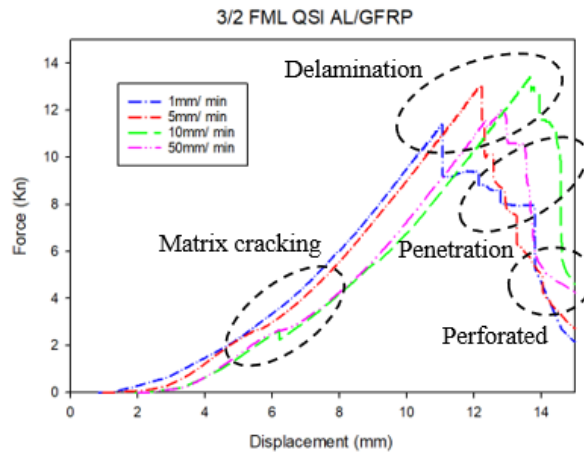


Figure 30.2: Graph of load against central deflection for 3/2 Aluminium/CFRP

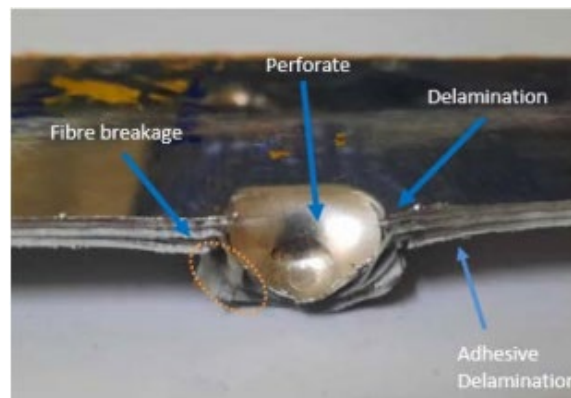


Figure 30.3: Fracture Cross section of Glass Fibre Metal laminate

CONCLUSION

In nutshell, this research bridges the assessment of potential lightweight automotive materials with the evaluation of two testing methodologies used in assessing their low-velocity and quasi-static impact tests. Preceding study on 2/1 layer of FML indicate the combination of AL/SRPP show the high resistance toward the delamination behaviour compared to other specimens. The recent study of 3/2 layer configuration show the impact response of AL/SRPP lead to have high energy of load (N) against central deflection (mm) impact response compared to other material.

REFERENCES

- Ganesrajan, D., Simon, L., Tamrakar, S., Kiziltas, A., Mielewski, D., Behabtu, N. & Lenges C. 2022. Hybrid composites with engineered polysaccharides for automotive lightweight. *Composites Part C: Open Access*, 7.
- Heibeck, M., Rudolph, M., Modler, N., Reuter, M. & Filippatos, A. 2021. Characterizing material liberation of multi-material lightweight structures from shredding experiments and finite element simulations. *Minerals Engineering*, 172.
- Hynes, N. R. J., Vignesh, N. J., Jappes, J. T. W., Velu, P. S., Barile, C., Ali, M. A., Farooq, M. U. & Pruncu, C. I. 2022. Effect of stacking sequence of fibre metal laminates with carbon fibre reinforced composites on mechanical attributes: Numerical simulations and experimental validation. *Composites Science and Technology*, 221.

- Rajan, B. M. C., Kumar, A., Sornakumar, T. & Kumaar, A. S. 2018. Impact Response and Damage Characteristics of Carbon Fibre Reinforced Aluminium Laminates (CARAL) under Low Velocity Impact Tests. *Materials Today: Proceedings*, 5, 20070-20077.
- Romli, N. K., Rejab, M. R. M., Bachtiar, D., Siregar, J., Rani, M. F., Harun, W. S. W., Salleh, S. M. & Merzuki, M. N. M. 2017. The behaviour of Aluminium Carbon/Epoxy fibre metal laminate under quasi-static loading. *IOP Conference Series: Materials Science and Engineering*, 257.

ACKNOWLEDGEMENT

The authors are grateful to the Universiti Malaysia Pahang Al-Sultan Abdullah (UMPSA) for financial supports.

P31

Fatigue Characterisation for Effective Strain Damage Model on Various Road Load Condition

L. Abdullah, S. S. K. Singh, S. Abdullah, A. H. Azman, and A.K. Ariffin

*Department of Mechanical and Manufacturing Engineering, Faculty of Engineering and Built Environment, Universiti Kebangsaan Malaysia, UKM Bangi 43600, Selangor, Malaysia
Email: salvinder@ukm.edu.my*

ABSTRACT

The aim of this paper is to characterise the fatigue under various random strain loads for extreme value distribution that considers the effects of cycle sequence condition. Fatigue under various random strain loads of a leaf spring is important as it is designed for maximum tension and compression load. Finite element analysis is carried out to determine the critical region on the leaf spring. In addition, the strain loads were experimentally captured under three different road load conditions, i.e. rural, campus and highway, in order to obtain the strain load behaviour under dynamic loadings. The fatigue life predicted by the strain-life models and the ESD model was within 1:2 and 2:1 boundary conditions of conventional fatigue life. In addition, the strain life is correlated using strain-life models because it reflects effects of cycle sequence. Hence, the effective strain damage model that considers the cycle sequence effect can provide an accurate fatigue life prediction compared towards the conventional strain life model under various road loads.

Keywords: Fatigue life, Finite element, Leaf Spring, Strain load, Cycle sequence effect.

INTRODUCTION

Automotive components or structures are typically subjected to numerous deterministic and random loads. Fatigue failure due to repetitive loading starts with crack dissemination until the ensuing fracture. Such a breach generally occurs in automotive components made of steel. Fatigue failure in automotive suspension systems has been characterised using macroscopic investigation on leaf springs subjected to static loading and fatigue tests (Nofri, 2020). Leaf spring is exposed to cyclic fatigue loading because of irregular road surface. Thus, this component supports a minimal weight of over 10^7 load cycles within its in-service life (Ceyhanli & Bozca, 2020).

Durability analysis is essential to ensure that structures or components can withstand under random amplitude loads. This evaluation is implemented according to fatigue damage theory that involves damage accumulation rule and an appropriate cycle counting technique (Zhao et al. 2018). This study aims to evaluate fatigue life prediction from strain-life models of random strain data for various road conditions and the extended use of carbon steel spring. Thus, effective strain damage (ESD) model is proposed to characterise the fatigue life prediction on random road loads for further fatigue assessment.

METHODOLOGY

The process flow of fatigue characterisation using finite element analysis (FEA) and durability models illustrates in Figure 31.1. FEA was used to determine the critical hotspot on the leaf spring. A vehicle load of 34.814 kN was applied using the FE model, and the total vehicle mass was divided by four to mimic the applied load for each leaf spring. Figure 31.2 shows the critical hotspot region with a maximum deformation at 22.75 mm. The maximum critical region based on the von-Mises stress shown

in Figure 31.3 is determined as the location where the strain gauge is attached in the U-bolt area with space constraints.

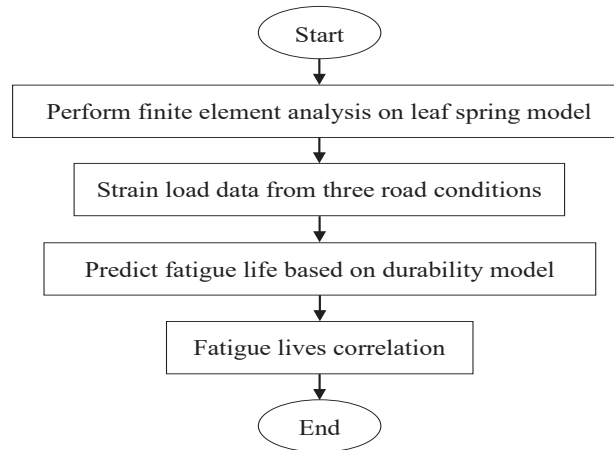


Figure 31.1: Process flow of fatigue characterisation

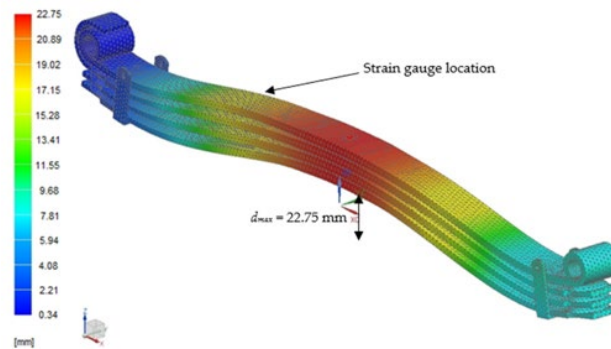


Figure 31.2: Critical hotspot region at maximum load

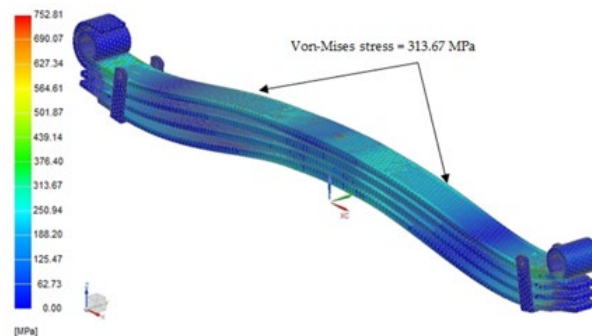


Figure 31.3: von-Mises stress mapping under maximum load

RESULTS AND DISCUSSION

Various strain data were captured using strain gauge on a leaf spring of a commercial bus. Data were extracted in three road conditions; a highway, a rural, and a campus road. Data were collected in the duration of 60 s at a 500 Hz sampling rate resulted in 30,000 discrete data points. Figure 31.4 shows the strain data for the highway, rural and campus roads. It shows a campus data exhibit several high

amplitude events compared with rural data due to braking activity at humps found along the campus road, as highlighted in Figure 31.4(c) (black circle).

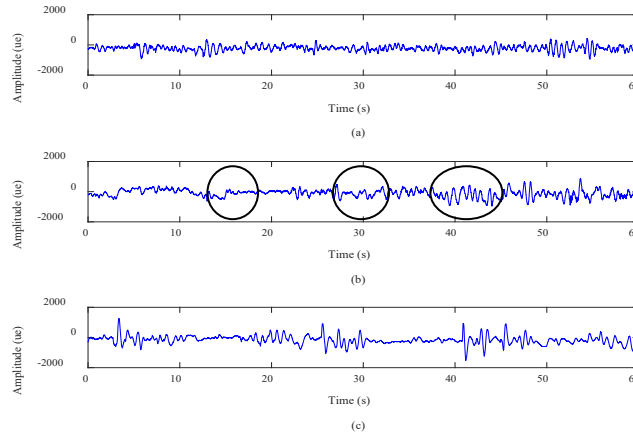


Figure 31.4: Time history of (a) highway, (b) rural, and (c) campus

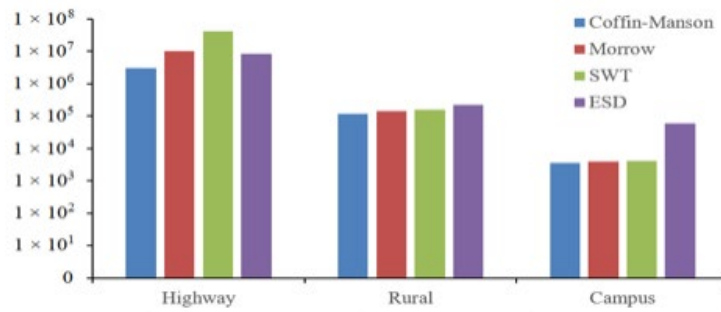
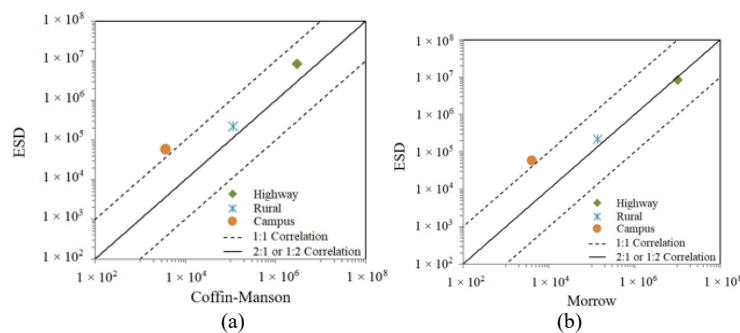


Figure 31.5: Fatigue life prediction of leaf spring

Figure 31.5 displays the correlation between the fatigue life predicted using ESD and other strain-life models. The correlation shows that the data are within the fatigue life boundary conditions of 1:2 and 2:1. The campus data lie outside the 1:2 or 2:1 boundary line, which is likely due to several high strain amplitude ranges. This indicates the survivability of the predicted fatigue lives and can be used in further fatigue assessment. The fatigue life predicted using the strain-life models of Coffin-Manson, Morrow and Smith-Watson-Topper (SWT), and ESD method is illustrated in Figure 31.6. The SWT model achieved the highest range in fatigue life as compared to the Coffin-Manson, Morrow and ESD models. The results present the highway data had a small strain amplitude which led to maximum fatigue life. Fatigue lives indicated that the ESD model uses a different approach in assessing the cycle sequence effect compared with the conventional strain-life models that only consider mean stress effects.



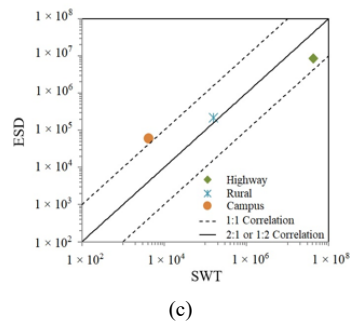


Figure 31.6: Fatigue lives correlation of (a) ESD vs. Coffin-Manson, (b) ESD vs. Morrow, and (c) ESD vs. SWT

CONCLUSION

Fatigue characterisation based on cycle sequence effect method under various strain loads of the carbon-steel heavy-vehicle leaf spring was discussed in this study. The FEA shows the deformation location occurred in the middle region of the leaf spring. The time history demonstrated that campus data contained highest amplitude ranges compared with rural and highway data. The highest fatigue life obtained from highway data was in the range of 4.19×10^7 to 3.02×10^6 cycle/block. Results indicated that strain-life models including the ESD model were within 1:2 and 2:1 conventional fatigue life and it can be acceptably used in predicting fatigue life. Hence, the ESD model can successfully estimate the fatigue life in assessing leaf spring durability under various road loads condition.

REFERENCES

- Ceyhanli, U.T., Bozca, M. 2020. Experimental and numerical analysis of the static strength and fatigue life reliability of parabolic leaf springs in heavy commercial trucks. *Advances in Mechanical Engineering* 12(7):1–17.
- Nofri, M. 2020. Leaf Spring Damages Analysis on Four Wheel Vehicle. *Chemical and Process Engineering Research* 63:27-38.
- Zhao, L.-H., Cai, H.-C., Wang, T., Zheng, S.-L. 2018. Durability assessment of automotive structures under random variable amplitude loading. *Advances in Mechanical Engineering* 10(4):1-8.

ACKNOWLEDGEMENT

The authors would like to express their gratitude to Universiti Kebangsaan Malaysia (Research funding: FRGS-1-2019-TK03-UKM-02-1 and DCP-2019-015 for supporting this research.

P32

Damage Assessment of Stiffened Steel Plates Subjected to Blast Loads Using Finite Element Analysis

N. S. A. Razak and A. Alias

*Faculty of Civil Engineering Technology, Universiti Malaysia Pahang Al-Sultan Abdullah, 26300, Gambang, Kuantan, Pahang
Email: aizat@ump.edu.my*

ABSTRACT

This paper study the damage of stiffened steel plates subjected to blast loads at close distances. Finite element method and analysis were used to simulate and investigate the damage of steel plates when subjected to the blast loads. The 3D finite element models were developed using solid elements and classical plasticity constitutive equations combine with ductile damage criterion were used to model the behaviour and damage of steel plates. The Cowper-Symonds equation was used to consider the strain rate effects. The blast loads were applied using Conwep function. The finite element models were verified and validated against disparate experimental results where a good agreement was obtained. The study was extended by including stiffeners to the steel plates to study the influence of stiffeners configurations on the damage of the steel plates when subjected to blast loads. This study confirms that the onset of failure of stiffened plates occurs at much lower blast pressure compared to unstiffened steel plates. Moreover, the damage or failure of the steel plates are influenced by the location and arrangement of the stiffeners with respect to the centre of explosions as well as the size of the stiffeners. The damage of the plates is dominated by tearing of the plate with ‘petals’ propagated away from the point impact along the edge of stiffeners.

Keywords: Stiffened Steel Plates, Blast Loads, Finite Element Analysis, Damage.

INTRODUCTION

The performance of steel plates with stiffeners improves when subjected to blast loads (Razak & Alias, 2021; Gan *et al.*, 2022). Stiffeners play an important role in energy absorption mechanism of stiffened steel plates (Kong *et al.*, 2021) where the numbers and the size of stiffeners could influence this capability (Gan 2022, Razak & Aizat 2021). However, the additional stiffeners has resulted the steel plates prone to failure at much lower blast impulse compared to unstiffened steel plates (Langdon, Yuen & Nurick, 2005). This finding was observed not only in experiment but also in their numerical model. Their numerical model, however, indicated the failures of the stiffened steel plates through the increase of localised temperature and excessive stretching of the elements hence, no failure criterion was used in the study. Mode I (large permanent plastic deformation) and Mode II (tearing of the plates) failures were observed in their study. Therefore, this study incorporates damage model to investigate and assess the influence of stiffener sizes and configuration to the damage of stiffened steel plates. The increase in size and closer arrangement of stiffener might increase the rigidity of the plate which could resulted in earlier onset of failure.

METHODOLOGY

This study used finite element analysis (FEA) using Abaqus to investigate and assesses the behaviour and damage of stiffened steel plates subjected to blast loads. Half-symmetry FE models were developed according to experimental work conducted by Yuen *et al.* (2018). The blast loads were varied from 29.1 to 36.4 g of TNT and applied using the Conwep blast load function which the standoff distance is 40

mm. The steel was modelled using classical plasticity with ductile damage model and the Cowper-Symonds model for damage and strain rate effect, respectively. The failure criterion was based on plastic strain where 0.25 was used according to reported literature for similar type of materials (Li *et al.*, 2018; Gan *et al.*, 2022). The sizes of the stiffeners were varied with four different sizes which represent four different width-to-height (b/d) ratios. Four difference stiffeners configurations were used as depicted in Figure 32.1.

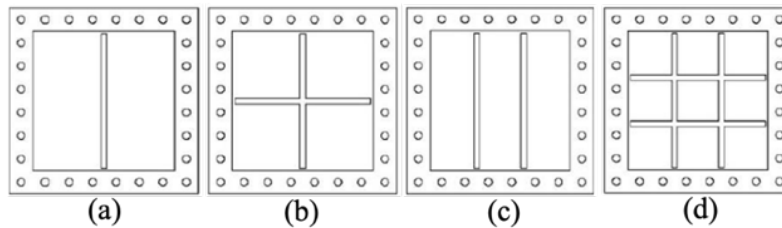


Figure 32.1: Stiffeners configurations a) Single Stiffener (SS), b) Cross Stiffener (CS), c) Double Stiffener (DS) and d) Double-Cross Stiffener (DC) with different b/d ratios.

RESULTS AND DISCUSSION

Table 32.1 tabulates the failure modes of different stiffeners configurations with constant b/d ratio of 0.5 subjected to three different explosive weight. The results suggest that the stiffened plates are prone to failure at much lower blast impulse except for CS-0.5 plates, which the mode of failure similar to Control plates. Thus, CS-0.5 could be suggested as the best stiffener configuration based on the results as depicted in Figure 32.2. The results from Table 32.1 concur with the finding of Langdon, Yuen and Nurick, (2005) regardless the number of stiffeners on the steel plates. The torn plate folded outward and created what is known as ‘petalling’ (Langdon, Yuen & Nurick, 2005).

Table 32.2 shows the influence of stiffeners width/height (b/d) ratios and the configurations of the stiffeners subjected to blast pressure from exploded TNT of 36.4 g. In general, increasing the b/d ratio does not always improve the mode of failure of stiffened plates. Figure 32.3 shows that the damage of the DC stiffened steel plates become more severe when the b/d is increased. In contrast, the mode of failure for the CS configuration improved from Mode II to Mode I. This improvement could be contributed by the configuration of the stiffeners with respect to the point of impact of the blast pressure on the stiffened plates as shown in Figure 32.4. Figure 32.4(a) suggests that the damage is due to insufficient stiffeners at the point of impact compared to Figure 32.4(b) where two stiffeners intersect at the point of impact. Although there is more than one stiffener presents in Figure 32.4(c) and 32.4(d), the location of the stiffeners is further from the point of impact.

Table 32.1: The influence of stiffeners to the modes of failure of stiffened steel plates

TNT (g)	Control*	SS- 0.5	CS- 0.5	DS- 0.5	DC- 0.5
29.1	MI	MII	MI	MI	MI
32.8	MI	MII	MI	MII	MII
36.4	MII	MII	MII	MII	MII

*Control: Unstiffened steel plates

MI: Mode I (Plastic deformation)

MII: Mode II (Plate tearing)

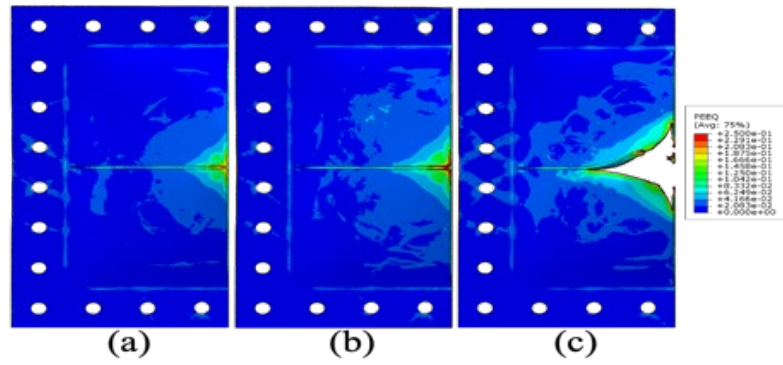


Figure 32.2: The damage of CS-0.5 steel plates subjected to a) 29.1 g, b) 32.8 g and c) 36.4 g of TNT

Table 32.2: The influence of stiffeners width/height ratio and configurations to the modes of failure

TNT (g)	b/d	SS	CS	DS	DC
36.4	0.5	MII	MII	MII	MII
	1.0	MII	MII	MII	MII
	1.5	MII	MI	MII	MII
	2.0	MII	MI	MII	MII

MI: Mode I (Plastic deformation)

MII: Mode II (Plate tearing)

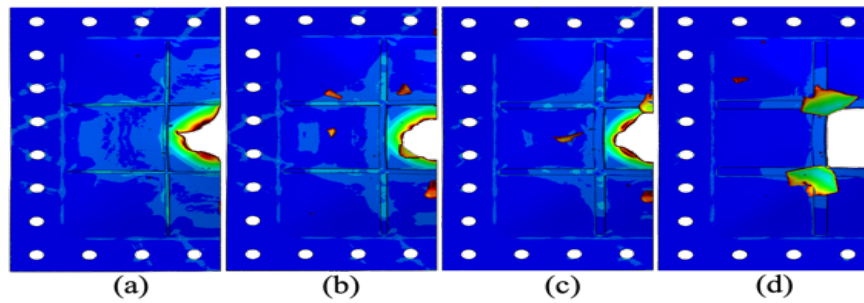


Figure 32.3: Mode 2 damage of double cross stiffened steel plates subjected to blast load of 36.4 g of TNT with different b/d ratios a) 0.5, b) 1.0, c) 1.5 and d) 2.0.

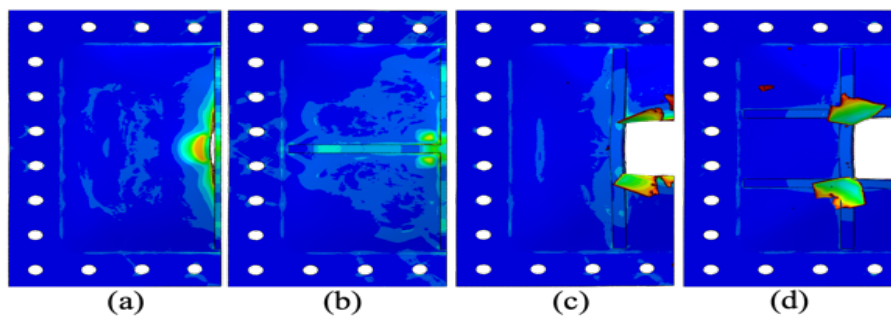


Figure 32.4: The influence of stiffeners configurations; a) single stiffener b) cross stiffeners, c) double stiffeners and d) double cross stiffener on the modes of failure of stiffened steel plates due to 36.4 g of TNT with respect to the point of impact ($b/d=2.0$).

CONCLUSION

This study confirms that the onset of failure of stiffened plates occurs at much lower blast pressure compared to unstiffened steel plates. Moreover, the damage or failure of the steel plates are influenced by the location and arrangement of the stiffeners with respect to the point of impact. In addition to that, the damage is also influenced by the size of the stiffeners. The damage of the plates is dominated by tearing of the plate with ‘petals’ propagated away from the point impact along the edge of stiffeners.

REFERENCES

- Gan, L. *et al.* (2022) ‘Influence of U-shaped stiffeners on the blast-resistance performance of steel plates’, *Journal of Constructional Steel Research*, 188(November 2021), p. 107046.
- Kong, X. *et al.* (2021) ‘Corrected method for scaling the dynamic response of stiffened plate subjected to blast load’, *Thin-Walled Structures*, 159(June 2019), p. 107214.
- Langdon, G. S., Yuen, S. C. K. and Nurick, G. N. (2005) ‘Experimental and numerical studies on the response of quadrangular stiffened plates. Part II: Localised blast loading’, *International Journal of Impact Engineering*, 31(1), pp. 85–111.
- Li, M. *et al.* (2018) ‘Experimental and numerical study on damage mechanism of CFDST bridge columns subjected to contact explosion’, *Engineering Structures*, 159(January), pp. 265–276.
- Razak, N. S. A. and Alias, A. (2021) ‘The Response of Steel Plates Subjected to Close-in Blast Loads’, *IOP Conference Series: Earth and Environmental Science*, 682(1), p. 012036.
- Yuen, S. C. K. *et al.* (2018) ‘The influence of orientation of blast loading on quadrangular plates’, *Thin-Walled Structures*, 131(April), pp. 827–837.

ACKNOWLEDGEMENT

The authors would like to express their gratitude and thanks to Universiti Malaysia Pahang Al-Sultan Abdullah for funding this research under the Post Graduate Research Scheme (PGRS200384).

Numerical Simulation of Temperature Field in Ultra-narrow Arc Welding of Thin-walled Steam Turbine Valve Body Material

Gouningnian¹, Shunshuo¹, Jiangxiaoxia¹, Yuanyongguang¹, Wangbangle¹ and Hanxiaotong²

¹*Ningxia University, Ningxia Yingchuan 750021*
Universiti Teknologi Malaysia, 81310 Johor Bahru, Johor
Tel: 18629647969 Email: gounn@nxu.edu.cn

ABSTRACT

This paper takes the welding experiment is carried out on the 60mm thickness of the steam turbine valve body material ZG13Cr9Mo2Co1NiVNB casting steel test block by using the technology of ultra-narrow gap welding with flux strips constraining arc. And the temperature field during welding is measured by using the thermocouple. Then, the finite element model of the welding temperature field was established by writing ANSYS software, and the uniform body heat source model and the element birth and death technology were used. The results show that the thermal cycle curve obtained by the simulation is basically consistent with the thermal cycle curve measured during the experiment, and the simulation result of the molten pool morphology is also consistent with the actual weld macroscopic morphology; when the current is 185A, the bottom of the sidewall is well fused. This model provides an effective basis for further optimisation of welding process parameters.

Keywords: ultra-narrow gap welding, temperature field, birth and death element, numerical simulation.

INTRODUCTION

With the rapid development of modern industrial equipment, various large-scale and thick-plate metal structures are used more and more widely (Chen et al. 2019). At present, in enterprises, manual arc welding and carbon dioxide gas shielded welding are adopted for the butt joint of large steel castings. When welding, the workload is large (Božić 2018).

The technology of ultra-narrow gap welding with flux strips constraining arc by Zhu (2012) with its lower heat input can effectively inhibit the growth of grains in the heat-affected zone, and obtain a strong, tough, and excellent weld. In order to reduce many experiments, numerical simulation is used to calculate the welding temperature field before the relevant welding process is formulated, which provides a reference for formulating a reasonable welding process.

RESULTS AND DISCUSSION

The position of the measuring point is 9mm away from the bottom surface of the current weld bead. The thermal cycle curve of the numerical simulation results and the experimental results are compared. It is found that the simulation result is similar to the temperature cycle curve measured by the experiment in the temperature rising part, as shown in Figure 33.1. Due to the welding equipment and the anti-deformation setting of the groove, the thermocouple wire cannot directly contact the welding material. So that the temperature change rate of the measured thermal cycle curve is smaller than that of the simulated thermal cycle curve. Figure 33.2 and Figure 33.3 are the simulated melt pool and thermal cycling curve distributions at 185 A.

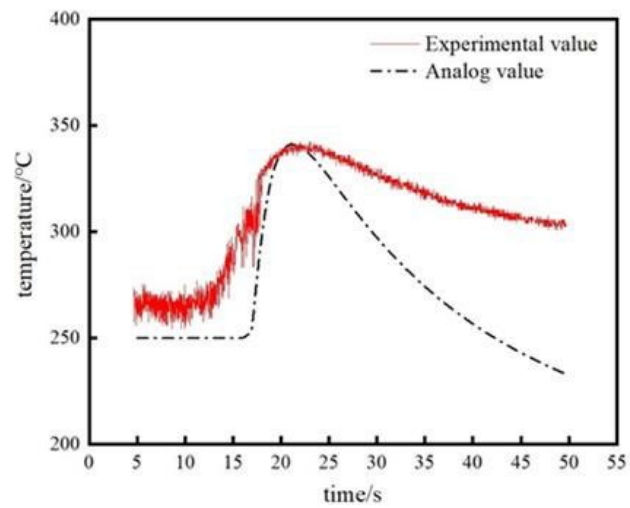


Figure 33.1: Comparison of temperature curves between measured values and simulated values

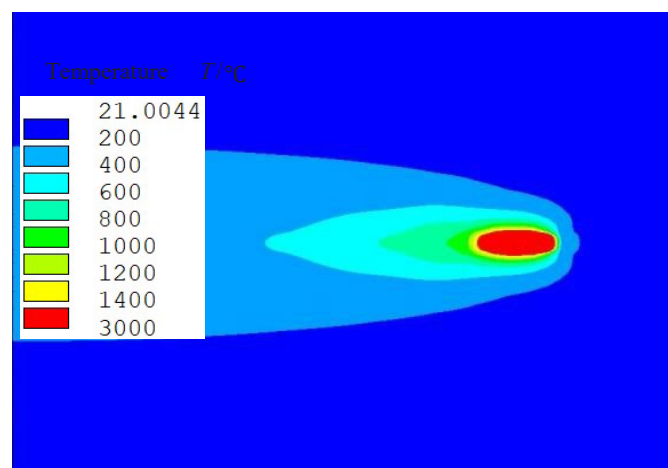


Figure 33.2: Temperature cloud diagram under 185A

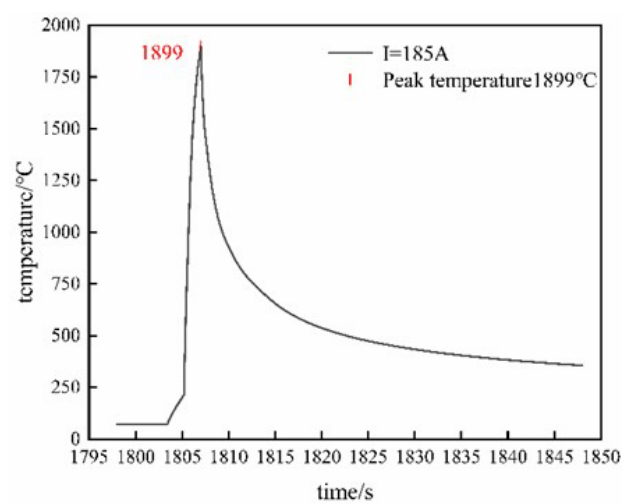


Figure 33.3 Temperature-time curve

CONCLUSION

Aiming at the welding problem of steam turbine valve body of Shared Casting Steel Co., Ltd., this paper adopts the method of experiment and simulation to study, and draws the following conclusions according to the research results:

1. The simulated thermal cycle curve is close to the thermal cycle curve of the welding measurement, and the morphology of the simulated molten pool is basically consistent with the macroscopic morphology of the weld.
2. The welding current increases, the peak temperature of the welding temperature field increases, and the weld width of the simulated molten pool becomes larger. When the welding current is 185A, incomplete fusion does not occur at the root of the sidewall.

REFERENCES

- Božić Željko, Schmauder Siegfried, Wolf Hinko. The effect of residual stresses on fatigue crack propagation in welded stiffened panels[J]. *Engineering Failure Analysis*, 2018, 84: 346–357.
- Chen Zhanglan, Xiong Yunfeng, Qiu Haijun, et al. Stress intensity factor-based prediction of solidification crack growth during welding of high strength steel[J]. *Journal of Materials Processing Technology*, 2018, 252: 270–278.
- Zhu Liang, Pan Yan Li, Li Yuan Bo. Ultranarrow Gap Welding with Constrained Arc by Flux Strips[J]. *Advanced Materials Research*, 2012, 472–475: 2814–2818.

ACKNOWLEDGEMENT

Appreciation is given to Key Research and Development Plan of Ningxia Hui Autonomous Region, Ningxia, China (No. 2019BDE03005) and Malaysia Ministry of Education, University Malaysia Pahang (PGRS1903160) for funding this research study.

P34

Failure Analysis of Radiant Tube in Ethylene Furnace

M. H. Mohammad

*PETRONAS Chemicals Derivatives Sdn Bhd, Administration Building, PETRONAS
Petroleum Industry Complex, KM106 Jalan Kuala Terengganu-Kuantan, 24300
Kertih, Terengganu, Malaysia
Tel: +60179002564 Email: mohdharris@petronas.com.my*

ABSTRACT

Radiant tube failure happened inside an ethylene furnace at a petrochemical plant after three (3) years in service. Upon detailed inspection, it was found that the tube experienced a circumferential crack in the parent material located at top section where it was covered by a roof refractory. The tubes were manufactured from ET45 micro centrifugal casting. A similar incident has also taken place at a parallel furnace previously. The failed tube was then cut, preserved, and sent out for a detailed metallurgical test. Findings concluded that the most likely cause of failure was due to cyclic thermal loading whereby temperature swing during the decoking cycle and furnace trip event. Furthermore, the movement of the tubes within the roof penetration arrangement may have been causing cyclic thermal and mechanical bending load in top part of the tube. There was no evidence of carburization, nitriding or creep damage in the material examined. This paper will discuss in detail the metallurgical test findings, causes of failure, and way forward to mitigate and avoid a similar incident from happening again in the future.

Keywords: ET45 micro, radiant tube, shock loading, precipitate, decoking

INTRODUCTION

Cracking furnaces are used in the production of olefin such as ethylene and other various products. The reaction typically happened at a very high temperature at around 850 °C. After three (3) years in operation, an operator detected a tube leak at one of the furnaces during regular visual inspection through the furnace sight-port. The furnace was then safely brought to shut down and all necessary inspection and repair were carried out accordingly.

Inspection using penetrant test (PT) identified one circumferential crack reported approximately 55mm long at parent material. The crack was located at top of furnace tube, 120mm below the top weld where ET45 micro furnace tube is joined with the grade 321 stainless steel inlet header tubes. (See Figure 34.1 & 34.2)

At the top of furnace, the tubes pass through the roof. The roof penetration consists of an interlocking refractory that surrounds the tube. This is backed by the steel firebox structure, which has an oval shaped steel ring that the tube passes through. The crack on tube was in the area of the tube that is covered by the refractory.

RESULTS AND DISCUSSION

ET45 micro material typically undergoes a significant increase in hardness upon initial exposure to high temperature due to formation of the fine secondary carbide network (See Figure 34.3). This initial increase in hardness is accompanied by a decrease in the material ductility. The material

around the crack in tube had an average bulk hardness of 348HV which is high for this material. Consistent with this, the microstructure in the vicinity of the crack had a very fine secondary carbide network.

Inspection and testing of material approximately 1m away for the failure locations reveals lower average hardness of 279 HV and coarser secondary carbide network. The crack was in a section of the tube that was within the roof refractory material during service. This means that this part of the tube was less directly exposed to the flue gas in the furnace and was therefore operating at a lower temperature than the tube further down. This lower temperature resulting in a lower amount of ageing over the time in service, is likely what led to the finer carbide network and increase hardness in the tube section that was inside the roof. It is also expected that there would be a corresponding reduction in ductility in this same area.

Based on visual examination, there was no evidence of glazing or smoothing from external surface and from internal surface, the condition was cleaned and there was no evidence of coking. The measured outer diameter agreed well with the nominal specified outer diameter. The measured wall thickness was also uniform, and all readings were slightly above nominal wall thickness. The fracture surface was completely covered in a thin oxide layer but its appearance was typical of a brittle fracture in a cast material.

From this it can be inferred that the tops of the tubes, where they were encased in the roof refractory were less ductile and more prone to brittle failure than the tube sections further down in the hotter part of the furnace. This is further evidenced by the fact that multiple similar failures have previously occurred in different tubes in the same roof region.

Another potential cause of such failure is shock loading of the tube due to the decoking process. The failure was detected 46 days after the last decoking run, which is within the timeframe given which is 3 to 370 days. It should be noted that the location of failure was at the inlet end of tube. This is the first part of tube that would be contacted by the relatively cold air/steam during decoking, meaning that this part of tube is potentially more prone to thermal shock during decoking than areas further down.

Multiple tube failures at roof penetration of the furnaces also suggests that the physical layout of the penetration could be influencing the failures. Inside the furnace tubes are relatively free to move due to their long length and hanging design. However, at the roof plate the movement of tubes side to side is restricted by the ring through which the tubes penetrate the roof plate. A bending load will be applied to the tube to restraining the tube sideward movement. The motion of tube may also cause it to be pressed against the roof refractory on one side, leaving a gap on the opposite side of tube where hot flue gas can flow. This could lead to uneven temperature distribution around the tube circumference and an associated thermal stress.

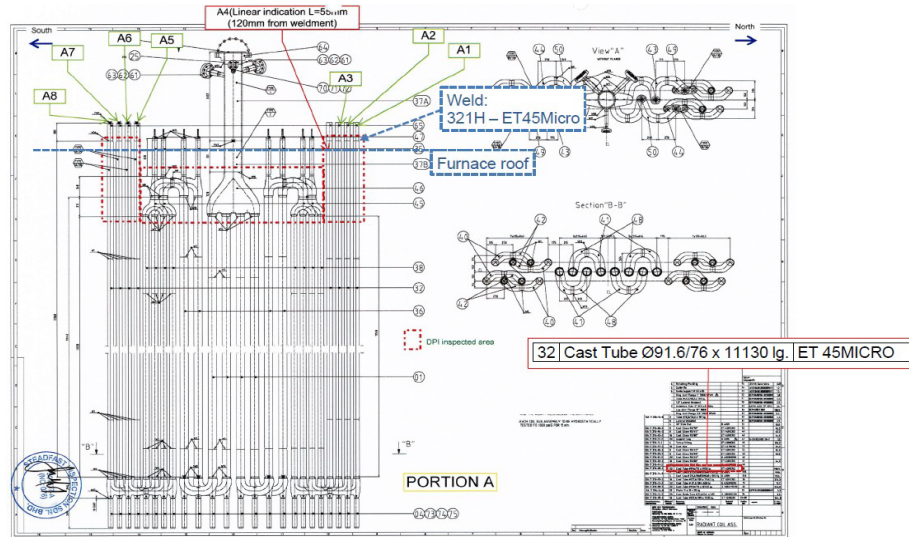


Figure 34.1: Furnace radiant tubes general arrangement showing location of failure

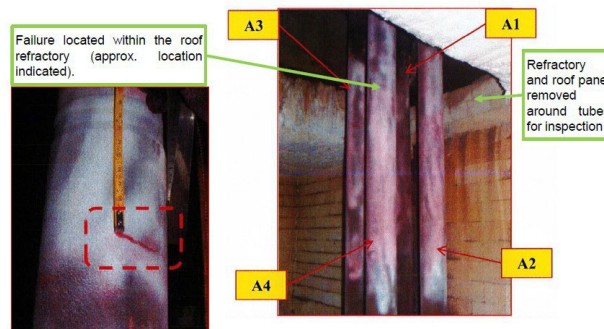


Figure 34.2: Location of crack approximately 120 mm below top weld

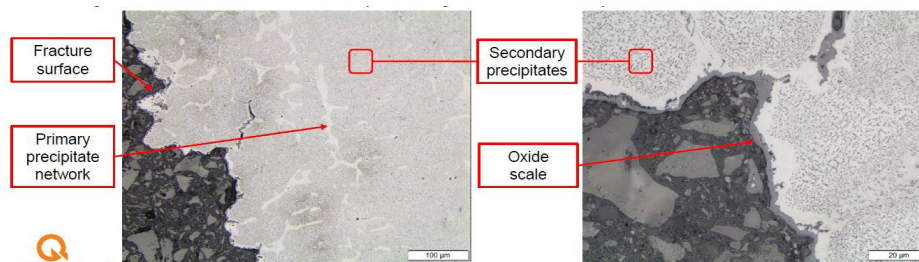


Figure 34.3: Microstructure at the end of the crack

CONCLUSION

From the investigation, it is concluded, the most likely cause of failure was an excessive shock loading that caused the tube to fail in brittle manner. There was also some evidence that movement of tube within the roof penetration arrangement may have been causing thermal cyclic and mechanical bending load in the top part of tubes. Fatigue due to this loading may also have contributed to the failure.

REFERENCES

American Petroleum Institute. 2011. API Recommended Practice 571, Second Edition, April 2011.
Damage Mechanism Affecting Fixed Equipment in the Refining Industry.
Quest Integrity: Alice Young and Nic Goodman; *Analysis of a Failed Furnace Tube*, 2021.

P35

Study on Chemical State Model of Reservice Chemical Equipment

Z. Nan, D. Wenjie, W. Qiangjing, S. Xinglong and L. Tongman

Ning Xia University

Tel: 13895316321 Email: 1473991063@qq.com

ABSTRACT

In this paper, the finite element method is used to predict the fatigue life of chemical equipment. Under the impact of pressure load spectrum and thermal load spectrum, rain flow counting and stress amplitude correction of pressure load spectrum were carried out, and combined with S-N curve, the fatigue life of chemical equipment was analysed. The fatigue life and maximum damage of chemical equipment under stress are obtained.

Keywords: life prediction, fatigue damage, stress impact, finite element

INTRODUCTION

With the rapid development of chemical industry, the operation and maintenance of process industry equipment presents the characteristics of large task, increasing frequency year by year, difficult detection and prediction, and high safety requirements. However, in traditional operation and maintenance, maintenance replaces operation and maintenance for chemical production units that are static or equipped with a small number of transmission mechanisms, and rarely has predictive operation and maintenance. The consequences are extremely serious. Often, chemical equipment cannot work normally without reaching its rated life. If the fault cannot be found in time or parts can be replaced. It will cause great loss to social property and personal property.

Collins proposed the concept of equivalent stress and equivalent total strain amplitude, and the life can be predicted according to the n-S curve of low cycle fatigue obtained under the equivalent strain spoke and uniaxial stress state. Pedersen et al. used notch stress method to reanalyse fatigue data of 767 specimens and put forward suggestions for revision of relevant design methods in IIW specification. Scholars David.G.Wilder et al. studied the fatigue strength and stability of structures under impact loading. The strength analysis and structure optimisation of micro gas turbine are carried out.

At present, the research of chemical equipment condition prediction is not enough. The application of finite element technology to chemical equipment safety condition prediction has high academic value, which can strongly support the development of theory and method of fault diagnosis of process equipment. And reliability prediction and safety early warning are of great industrial value.

The typical chemical equipment filter bag is selected as the analysis object, the analysis process is mainly divided into two parts, namely finite element calculation and fatigue life analysis. The finite element calculation includes Fluent module and Mechanical module.



Figure 35.1: Flow chart of filter bag stress analysis

A single filter bag was used for analysis and the model was simplified to a tubular structure. The fluid domain is meshed. In the actual operation of the dust collector, the speed is not a constant. It changes over time and is written in UDF for speed. Iterative calculation is performed after the relevant Settings are completed.

The filter bag material is glass fiber, and the pressure load of the fluid part is imported to the structure part for simulation calculation. According to Palmgren-Miner theory, and with the help of finite element method for analysis. The current fatigue life analysis generally refers to the fatigue analysis based on the finite element analysis results, that is, the finite element analysis results (generally stress and strain results) as the main input file of fatigue analysis, select a fatigue analysis model, calculate the fatigue life distribution of parts, and its flow chart is as follows:

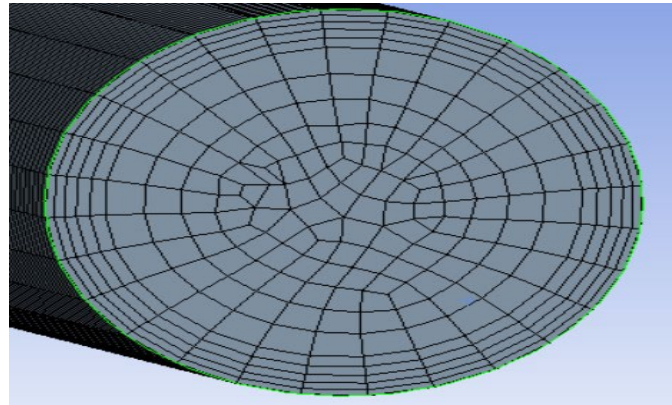


Figure 35.2: Filter bag structure and fluid mesh division

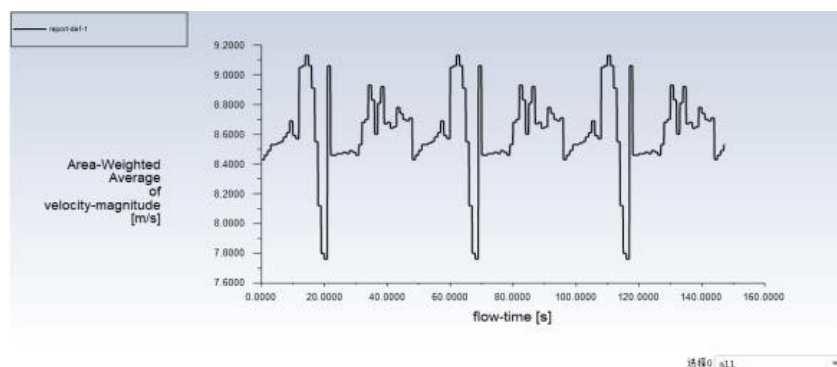


Figure 35.3: UDF writes random velocity entry

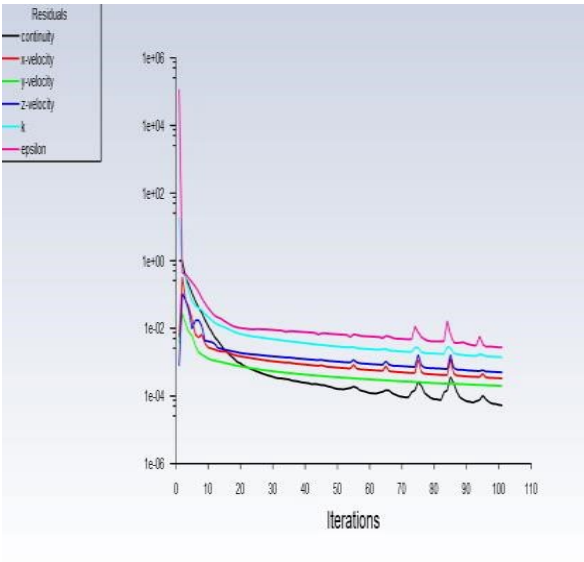


Figure 35.4: Fluent performs iterative calculation

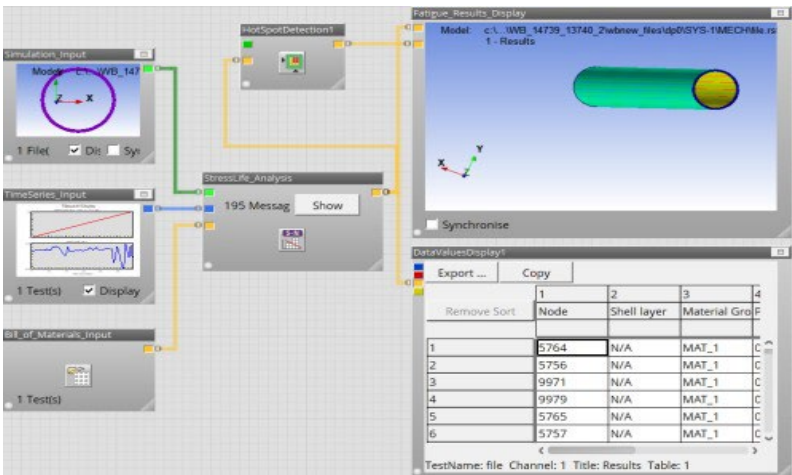


Figure 35.5: Life prediction analysis structure diagram

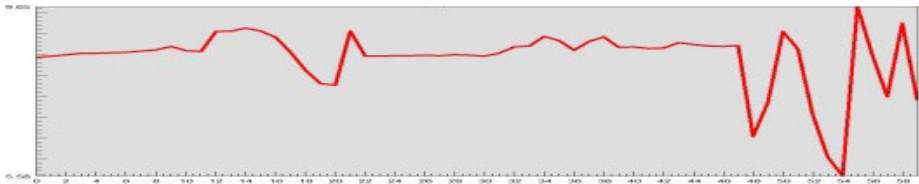


Figure 35.6: Measured random fatigue load spectrum

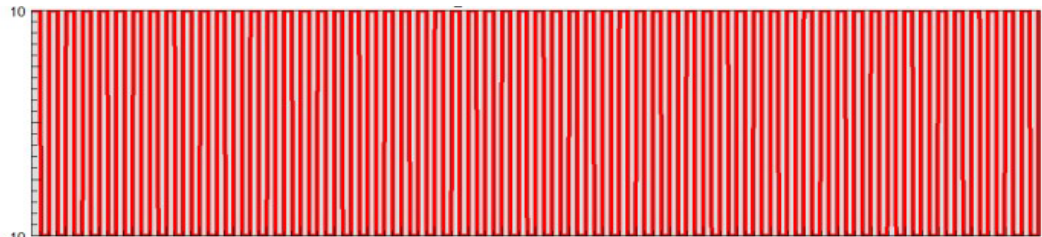


Figure 35.7: Sinusoidal fatigue load spectrum

The fatigue life curve of glass fiber was obtained by referring to relevant information, which was input into fatigue analysis software through material editing function, and s-N curve was made.

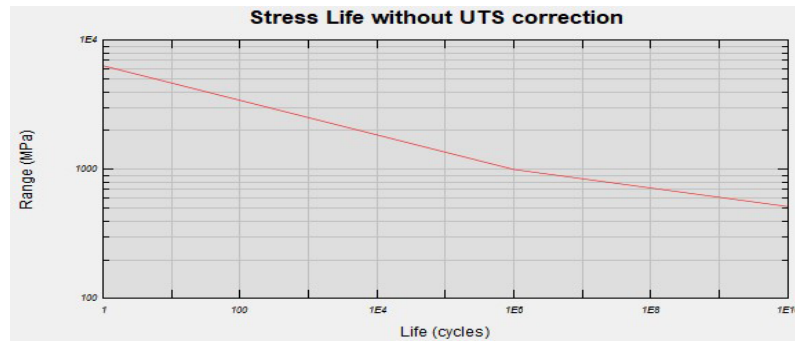


Figure 35.8: Fatigue curve of material

After the input conditions are completed, the fatigue life prediction is carried out after the analysis method is selected in the S-N analysis model and the stress amplitude is modified:

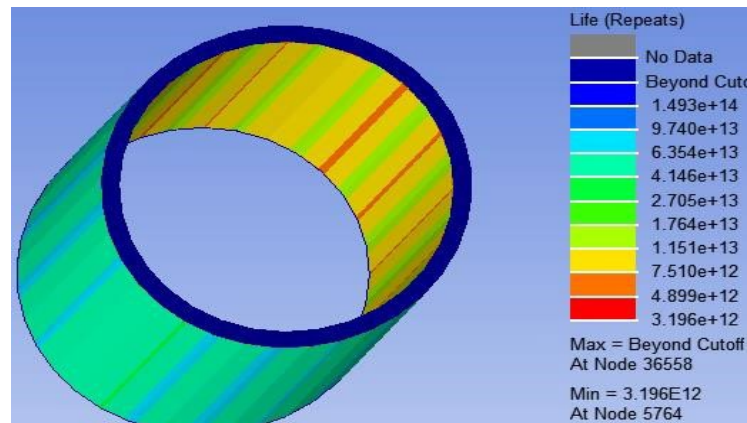


Figure 35.9: Fatigue life of filter bag under random load pressure impact

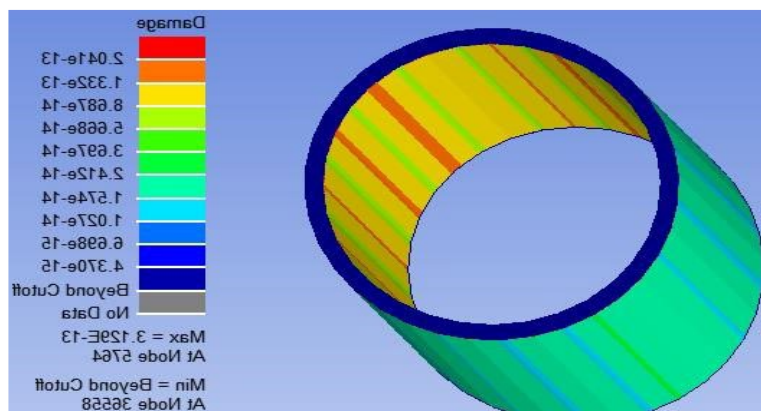


Figure 35.10: Maximum damage of filter bag under regular load press

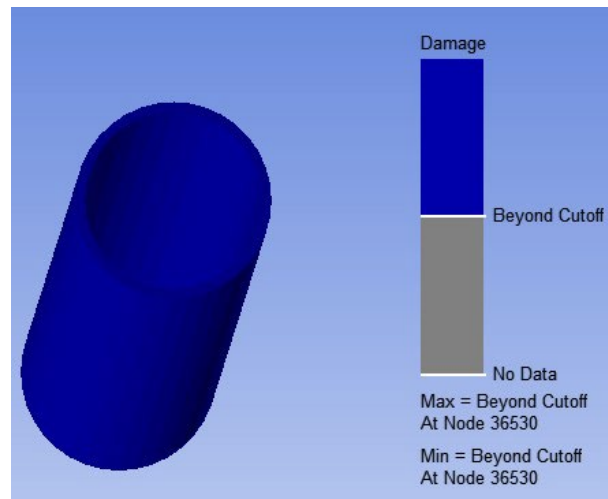


Figure 35.11: Fatigue life of filter bag under random load pressure impact

RESULTS AND DISCUSSION

The results show that the minimum fatigue life of the filter bag is 3.196×10^{12} times and the maximum damage is 2.041×10^{-13} under the pressure random load. Under the action of sinusoidal fatigue load, the life of filter bag does not change obviously, which is often called infinite life.

CONCLUSION

In view of the common load (pressure shock and thermal shock) in actual production, the fatigue life of reservice chemical equipment is predicted according to two kinds of pressure load spectrum and thermal load spectrum with equal amplitude. According to the predicted results, the life of the filter bag does not change obviously under the action of sinusoidal fatigue load, which is often called infinite life. The damage effect of random load spectrum on chemical equipment is stronger. The significance of this research lies in those parts should be replaced or repaired as soon as possible before the equipment is damaged to minimize the loss.

REFERENCES

- Collins J A. Failure of Materials in Mechanical Design [C]. Analysis Prediction Prevention. New York:J Wiley, 1981.629-633.
- David G. Wilder, Assen R. Aleksiev, Marianne L. Magnusson, Malcolm H. Pope, Kevin F. Spratt, Vijay K. Goel. Muscular Response to Sudden Load: A Tool to Evaluate Fatigue and Rehabilitation [J].
- Fisher J W. Fatigue and fracture in steel bridges: Case studies[M]. John Wiley Sons: 1984.
- Li Jianhua, WANG Yuhua. Research progress on comprehensive Energy consumption analysis of steam pyrolysis process
- Pedersen M M, Mouritsen O Ø, Hansen M R, et al. Re-analysis of fatigue data for welded joints using the notch stress approach[J]. *International journal of fatigue*, 2010,32(10):1620-1626.
- Tavernelli J F, Coffin L F. Experimental Support for Generalized Equation Predicting Low Cycle Fatigue. Transactions of the ASME[J], *Journal of Basic Engineering*, 1952, 84(4): 533~537.

P36

Investigation and Analysis of Pipe Support Weld Cracks

M. Z. Kamardin¹, M. A. Ton Alias¹, A. H. Mohamad¹, A. R. Barol², and R. Yahaya²

¹*PETRONAS Group Technical Solutions (GTS), Menara 3, Menara Berkembar PETRONAS,
50088, Kuala Lumpur Malaysia*

²*PETRONAS Refinery and Petrochemical Corporation – Utilities and Facilities Sdn. Bhd (PRPC-
UF),*

81600, Pengerang, Johor, Malaysia

Tel: +60327836491 Email: zaid_kamardin@petronas.com.my

ABSTRACT

High pressure boiler feed water (HPBFW) pipe was found failed due to cracks at fillet weld between pipe support saddle pad to pipe body. Microscopy examination and fractography analysis shows the cracks initiated and propagated along the heat affected zone (HAZ) of the weld. Microhardness testing indicates higher hardness on the HAZ of the weld while Scanning Electron Microscopy (SEM) of the fracture surface shows quasi-cleavage features which are tell-tale signs of hydrogen embrittlement (HE). From site measurement of the supporting structures and finite element analysis (FEA), it was found that section of the pipe that failed was subjected to high downward and bending load leading to the crack propagation. Based on all the data collected, it is concluded that the pipe failed due to HE as a result of poor welding execution during construction and contributed by increase in secondary load due to pipe supporting structure settlements. The result of the failure analysis enables the team to identify the root causes of the failure and develop prevention and corrective actions accordingly.

Keywords: Failure Investigation, Root Cause Analysis, Hydrogen Embrittlement, Delayed Cracking, Underbead Cracking

INTRODUCTION

The main function of the facility in Pengerang, Johor is to supply utilities including high pressure boiler feed water (HPBFW) to neighbouring refinery and petrochemical plants.

After approximately 5 years in commercial operations, a leak was identified at a 20-inch HPBFW pipe located on a pipe bridge during routine Operation Structured Round by Field Operator. Site inspection revealed longitudinal and circumferential cracks along the toe of a pipe support saddle pad to pipe fillet weld at 6 o'clock position. The cracks propagated in relatively straight manner with no branching (Figure 36.1). The failed section was removed and sent to laboratory for analysis.



Figure 36.1: Longitudinal and Circumferential Cracks at Saddle Pad to Pipe Fillet Weld

RESULTS AND DISCUSSION

Microscopy examination and fractography analysis of the failed section reveals that both primary longitudinal and secondary circumferential cracks propagated along the fillet weld heat affected zone (HAZ) and into the pipe body (i.e., from outer to inner surface) (Figure 36.2).

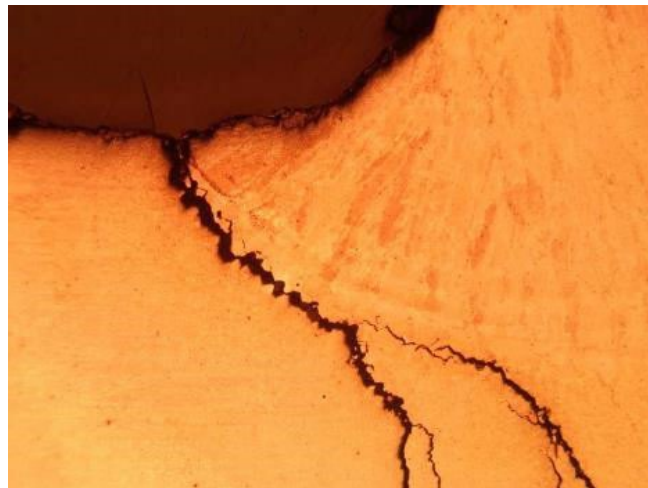


Figure 36.2: Microscopy Image Shows Crack Initiated at the Weld Toe and Propagated along the HAZ

The fracture surface exhibits intergranular and quasicleavage features (Figure 36.3). These indications are consistent with cracking due to HE. In addition, striation marks were observed on the fracture surface, indicates contribution of fatigue to the crack propagation.

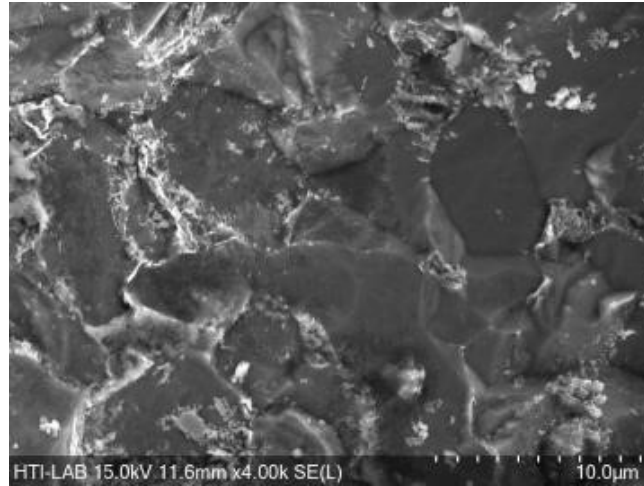


Figure 36.3: Scanning Electron Microscopy (SEM)) of the Fracture Surface Shows Quasi-Cleavage Features

Microhardness testing detected higher hardness on the HAZ of the weld, measuring up to 271 HV, which indicates presence of hard microstructure and residual stress that can contribute to the crack initiation and propagation

Finite Element Analysis (FEA) was conducted to understand loads acting on the weld. When subjected to downward force, high stress concentration point is observed at the weld toe, instead of at the weld root, which is consistent with the observed crack initiation point (Figure 36.4). Additionally, based on FEA, due to design and restraint of the pipe support saddle, the tangential stress is higher than the axial stress increasing the tendency of crack to propagate longitudinally.

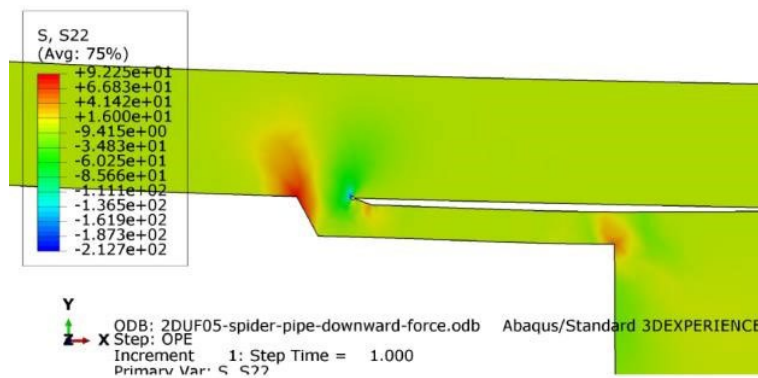


Figure 36.4: FEA Result when the Weld is Subjected to Downward Forces

Site inspection and assessment of the pipe bridge structure by Theodolite survey revealed vertical downward deflection of more than 70mm at the failed location. This is the main factor that increased the loads acting on the pipe. The investigation also noted frequent fluctuation of flow rate between 50 to 200m³/hour which could result in low frequency vibration and cyclic stress on the piping (Figure 36.5).

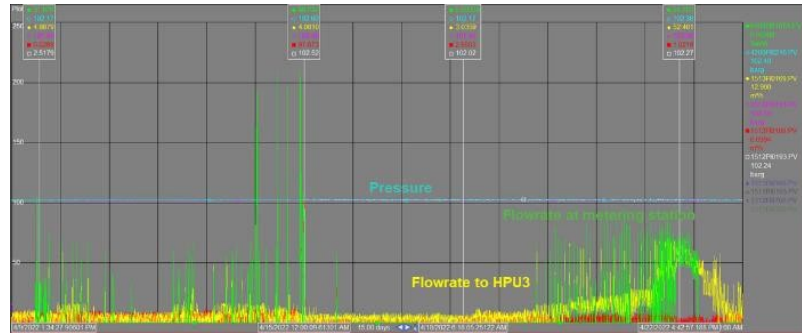


Figure 36.5: HPBFW Flow Rate Data Indicates Frequent Changes and Fluctuation

CONCLUSION

Based on all the evidence and data collected and analysed, the failure was caused by HE. The crack was initiated at hard microstructure and hydrogen-embrittled area of the fillet weld and propagated in longitudinal and circumferential manner as a result of increased in secondary load due to pipe structure deformation and low cycle fatigue due to intermittent change of the flow rate.

HE, also known as cold cracking, delayed cracking, underbead cracking and hydrogen cracking in various standards and literature, requires combination of three factors to cause failure which are [2, 3]:

- i. Hydrogen generated by the welding process
 - ii. Hard and brittle microstructure
 - iii. Tensile stress acting on the welded joint
- Failure investigation conducted confirms the presence of all three factors at the failed location.

Moving forward, the facility plans to extend the inspection and testing to other potentially susceptible weld locations. In addition, soil settlement and pipe support survey are to be conducted to identify other locations that may experience high load and stress.

REFERENCES

- The Welding Institute (TWI), *Fracture Mechanics Assessment Of Hydrogen Effects On Steel*
<https://www.twi-global.com/technical-knowledge/published-papers/fracture-mechanics-techniques-for-assessing-the-effects-of-hydrogen-on-steel-properties?msclkid=9c6c4944d0d411ecb7033a6acad55922>
- American Petroleum Institute. 2011. API Recommended Practice 571, Second Edition, April 2011. *Damage Mechanism Affecting Fixed Equipment in the Refining Industry*. Washington: API.
- The Welding Institute (TWI), *Defects - Hydrogen Cracks In Steels - Prevention and Best Practice*
<https://www.twi-global.com/technical-knowledge/job-knowledge/defects-hydrogen-cracks-in-steels-prevention-and-best-practice-046>

P37

Combating Repetitive Failures at Sour Gas Concentration Unit Overhead System

M. H. Hasan, R. Jaafar and N. Ali

*Materials, Corrosion and Inspection, Level 9, Tower 3, Kuala Lumpur City Centre (KLCC),
Persiaran KLCC, 50088 Wilayah Persekutuan Kuala Lumpur, Malaysia*

*Tel: +60327836794 Email: hawari@petronas.com.my, rohana_jaafar@petronas.com.my,
nurjaimi_ali@petronas.com.my*

ABSTRACT

SGCU De-ethanizer Overhead Condenser experienced frequent severe corrosion with short run length of two to three years. Periodic inspection on water boot Overhead Receiver/Separator also revealed significant metal loss, indicating similar problem encountered by the whole overhead system. Root Cause Failure Analysis (RCFA) was conducted using Event Tree Diagram to identify the final root causes. Several types of credible damages were identified and evaluated to arrive at the most probable root causes. Erosion Corrosion, Hydrochloric (HCl) Acid Corrosion, Sour Water Corrosion, Sulfuric (H₂SO₄) Acid Corrosion and Wet H₂S Cracking were among the possible corrosion threat that might contribute to the leak incidents. The failed sample analysis revealed Hydrogen Grooving and “Fan-Shaped” morphology. Computational Fluid Dynamics (CFD) reported very low flow velocity, thus eliminating erosion corrosion as the governing degradation mechanism. RCFA concluded that significant corrosion was caused by Sulfuric Acid Corrosion attack.

Keywords: Sour Gas Concentration Unit, Overhead Corrosion, Refinery Corrosion, Sulfuric Acid Corrosion, Hydrogen Grooving.

INTRODUCTION

A refinery facility kept experiencing repetitive failures due to accelerated thinning at Saturated Gas Concentration Unit (SGCU) De-ethanizer overhead system. Overhead condenser specifically, suffers severe corrosion with shortest run length in the range of two (2) to three (3) years requiring periodic replacement. Periodic inspection at SGCU overhead receiver/separator water boot also revealed continuous metal loss exceeding 6mm design corrosion allowance. SGCU unit consists of feed gas compression section, the caustic treating section and fractionation section namely a de-ethanizer and a C3/C4 splitter designed to process the feed stream from Crude Distillation Unit (CDU), Condensate Fractionation Unit (CFU) and Catalytic Reforming Unit (CRU) to produce treated LPG, C3 for refrigerant and C4 for gasoline blending compound. The light hydrocarbon liquid coming from crude stabilizer, condensate stabilizer and CRU are collected in Gas Concentration Compressor Scrubber to strip C2 and lighter components before compressed inside the compressor to intended pressure. The gas is then heated up in the De-ethanizer Feed Exchanger before routed into column for separation. The De-ethanizer Overhead Vapor exit at column top and routed to De-ethanizer Overhead Receiver via the De-ethanizer Overhead Condenser. The C2 and lighter gas are then sent to Fuel Gas System, while the overhead receiver liquid is pumped up to no. 1 tray of the De-ethanizer as total reflux. On the other hand, the De-ethanizer Bottom Liquid is sent to the C3/C4 Splitter Column to separate C3 from C4.

The objective of this study is to identify the root cause of the repetitive failure by performing process reliability and integrity assessment, besides recommending feasible as well as cost effective solution to address the issues. This paper discusses the study conducted covering only the overhead of SGCU.

RESULTS AND DISCUSSION

Root Cause Analysis for De-ethanizer Overhead Condenser Towards determining the governing degradation mechanism leading to repetitive failures of condenser tubes, root cause analysis has identified five (5) possible degradation mechanisms for tubes external, namely Erosion-Corrosion, Hydrochloric (HCl) Acid Corrosion, Sulfuric Acid (H_2SO_4) Corrosion, Sour Water Corrosion (Acidic), Under Deposit Corrosion (UDC). This RCA also does not bias in looking only at the external cause, but also possible from internal mechanism such as Cooling Water Corrosion (CWC).

Based on the fault tree analysis (FTA) study, it can be strongly concluded that the cause of significant corrosion on De-ethanizer Overhead Condenser tubes external is caused by Sulfuric Acid Corrosion Attack. However, the extent of attack could not be predicted due to insufficient information and monitoring data. HCl Acid Corrosion attack is also possible, but the severity depends on the concentration of chloride presence in the system. Again, due to the limited sampling data, it is highly required for operation to establish the trending to firmly predict the extent of corrosion caused by this degradation mechanism.

Erosion corrosion, on the other hand, as highlighted by laboratory analysis to be the tube failure root cause, is not the governing degradation mechanism due to very low flow velocity observed in shell side. This mechanism, however, is possible at other turbulence flow locations such as elbows, tees, and pumps discharges. Likewise, Sour Water Corrosion (Acidic), UDC and Cooling Water Corrosion also are not the cause of repetitive failure due to unmatched corrosion rate, no significant deposit presence and the attack was from external instead of internal, respectively.

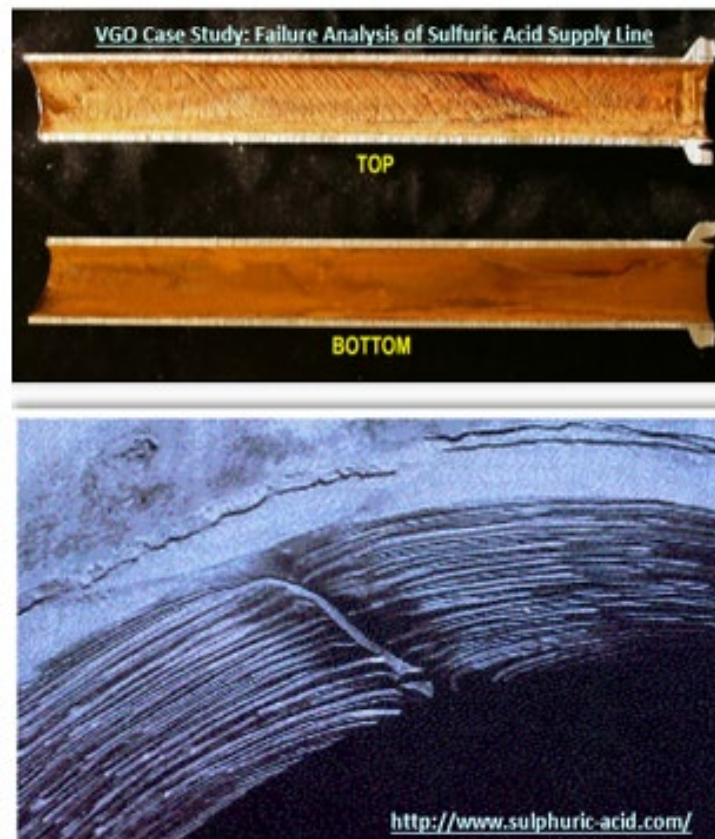


Figure 37.1: Typical Morphology of Sulfuric Acid Corrosion Attack

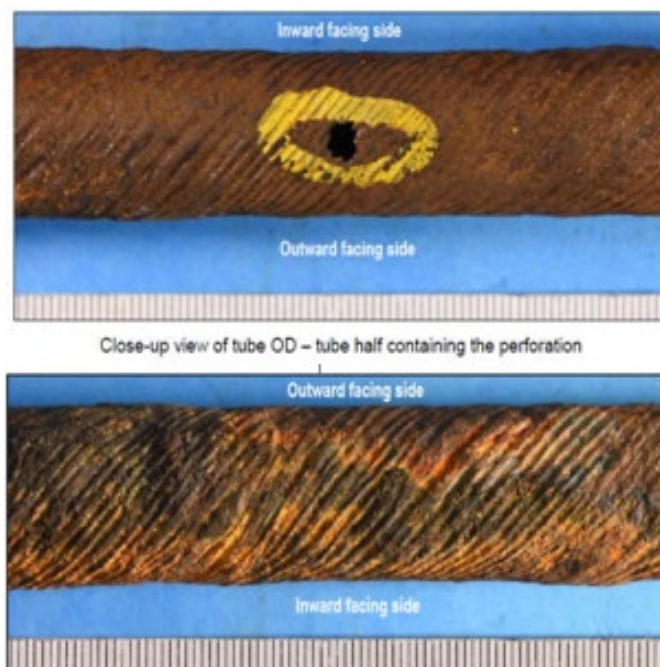


Figure 37.2: De-ethanizer Overhead Condenser Tube Failure Close Up View and Morphology

CONCLUSION

As the result of repetitive failures observed at De-ethanizer overhead systems, detail analysis has been carried out to determine the root cause of the issues. The study concluded that the governing cause of repetitive.

De-ethanizer Overhead Condenser tubes failure was due to the H_2SO_4 Acid Corrosion attack as the result of Caustic Treater relocation to elsewhere which deviates from original design philosophy to remove high H_2S concentrated. Material of construction at this De-ethanizer overhead also was not designed for sour service which could lead to Wet H_2S Damage since the calculated H_2S partial pressure already exceeding integrity limit. Summary of findings are as follows:

- De-ethanizer overhead system is exposed to several damage mechanisms namely Erosion Corrosion, HCl Acid Corrosion, H_2SO_4 Acid Corrosion, Sour Water Corrosion and Wet H_2S Damage. Relocation of Caustic Treater to downstream facilities led to presence of Very High H_2S of maximum 5,000ppm.
The reaction of H_2S and moisture in the overhead system leads to H_2SO_4 acid formation in liquid and FeS scale on the metal surface. Under Low Flow condition of less than 1m/s observed in condenser Shell Side (Tube external), mechanism of hydrogen grooving is expected to occur where movement of hydrogen bubbles on metal surface removes protective FeS scale and leads to localised corrosion. Morphology on tube surface is observed to be very similar to H_2SO_4 Acid Corrosion attack due to Hydrogen Grooving.
- Significant High corrosion rate of 1.35mm/year recorded at Overhead Receiver water boot indicating severe corrosion attack, typically in presence of acidic condition, either HCl Acid corrosion or H_2SO_4 Acid corrosion or combination of both mechanisms. In this case, it is obvious that the H_2SO_4 Acid is dominant over the HCl Acid taking into account the amount of chloride versus the H_2S contents as well as the expected predicted corrosion rates as discussed extensively in this report. Actual pH was not able to sample or determine due to unsuccessful attempts, however, pH 6 was reported from laboratory analysis on failed condenser tube sample.

- In addition to above, as the result of very high H₂S presence, the overhead system is also exposed to risk of Wet H₂S Damage in which it is best monitor via specific inspection program targeting areas starting from the condenser and downstream facilities where free water could be present.

ACKNOWLEDGEMENT

The authors would like to express gratitude and thanks to the following contributed team members, Marina Mosir, Megat Hilmi Ruzaini Megat Nuruddin and Maryam Ismail, PETRONAS.

LIST OF EDITORS



Associate Professor Ts. Dr. Nasrul Azuan Alang, known among his colleagues as Dr. Rambo, is an Associate Professor and researcher at the Faculty of Mechanical and Automotive Engineering Technology, Universiti Malaysia Pahang Al-Sultan Abdullah (UMPSA), Malaysia. He served as the Editor-in-Chief of the *Journal of Mechanical Engineering and Sciences (JMES)* from 2020 to 2025. Born in Perak, Malaysia, he is a registered Graduate Engineer with the Board of Engineers Malaysia (BEM) and the Malaysian Board of Technologists (MBOT). Dr. Nasrul obtained his Bachelor of Engineering (Mechanical Engineering) with First Class Honours from Universiti Teknologi Malaysia (UTM) in 2007.

He subsequently earned a Master of Engineering degree from Universiti Kebangsaan Malaysia (UKM) and a PhD in Mechanical Engineering from Imperial College London, United Kingdom. Throughout his career, he has served as a guest editor for several Scopus-indexed journals and as a technical committee member for more than six international conferences. Dr. Nasrul has authored two teaching modules in the areas of materials engineering and mechanics of materials. His research activities have resulted in the publication of more than 50 scientific papers in journals and conference proceedings, primarily focusing on mechanics of materials, creep behaviour, and finite element modelling. His contributions have been recognised through several prestigious awards, including the *Anugerah Cendekia Bitara* (2018), *Anugerah Perkhidmatan Cemerlang* (2020), and *Anugerah Editor Terbaik* (2023).



Dr. Aizat Alias is a Senior Lecturer at the Faculty of Civil Engineering Technology, Universiti Malaysia Pahang Al-Sultan Abdullah (UMPSA), Malaysia. He obtained his Bachelor's and Master's degrees from Universiti Teknologi Malaysia (UTM) and earned his PhD from Imperial College London, United Kingdom. He began his professional career in May 2009 as a Civil Engineer with the Public Works Department Malaysia (JKR) before joining academia in September 2010 as a Lecturer at UMPSA. He was promoted to Senior Lecturer in 2019. Dr. Aizat is actively involved in research and scholarly publication and has authored and co-authored more than 11 scientific papers in journals and conference proceedings.



Dr. Mohd Shamil Shaari holds a PhD in Mechanical and Materials Engineering from Universiti Kebangsaan Malaysia (UKM). He is currently a Senior Lecturer at the Faculty of Mechanical and Automotive Engineering Technology, Universiti Malaysia Pahang Al-Sultan Abdullah (UMPSA), Malaysia. He serves as the Editor-in-Chief of *Recent Trends in Mechanical and Automotive Engineering Technology (RTMAET)* and as the Managing Editor of the *Journal of Mechanical Engineering and Sciences (JMES)*. His research interests focus on finite element analysis, fatigue and fracture mechanics, and additive manufacturing. His recent

work emphasises the design, optimisation, and performance evaluation of topology-optimised and lattice-based structures for biomedical applications.



Associate Professor Dr. Salvinder Singh Karam Singh is an Associate Professor in the Department of Mechanical and Manufacturing Engineering, Universiti Kebangsaan Malaysia (UKM), Selangor, Malaysia. He received his PhD in Mechanical Engineering from Universiti Kebangsaan Malaysia. His research interests include fatigue life assessment, fracture mechanics, reliability assessment, and probabilistic engineering approaches. Throughout his academic career, he has published more than 500 scholarly articles and has achieved an H-index of 15. His publications have appeared in reputable international publishers, including Elsevier and Springer. In addition to his strong teaching and research record, he has contributed significantly to educational innovation, particularly in teaching methodologies and curriculum development.

RECENT TRENDS IN MECHANICAL AND AUTOMOTIVE ENGINEERING TECHNOLOGY

**THE 6TH SYMPOSIUM ON DAMAGE
MECHANISM IN MATERIALS AND STRUCTURES
(SDMMS 2022)**

The Proceedings of the 6th Symposium on Damage Mechanism in Materials and Structures (SDMMS2022) present a collection of 37 research articles highlighting recent advancements in durability, fatigue, creep, fracture mechanics, failure analysis, structural integrity, and material behaviour. Organised by Universiti Malaysia Pahang Al-Sultan Abdullah, Universiti Kebangsaan Malaysia, and co-organised by Ningxia University, China, the symposium was held on 16–17 August 2022 at Swiss-Garden Resort, Kuantan, Pahang.

This publication brings together innovative experimental, analytical, and numerical studies from researchers and industry experts locally and internationally. Serving as a platform for knowledge exchange and collaboration, the proceedings provide valuable insights and references for researchers, engineers, practitioners, and postgraduate students in the field of mechanical and automotive engineering technology.

This book serves as a valuable reference for researchers, engineers, postgraduate students, and practitioners interested in the latest developments in mechanical and automotive engineering technology.

