CRACK PREDICTION OF USING FINITE ELEMENT ANALYSIS

by

Muhammad Shahrul Izzwan Bin Salim
22646

Dissertation submitted in partial fulfilment of
the requirements for the
Bachelor of Mechanical Engineering with Honours

Jan 2020

Universiti Teknologi PETRONAS
32610 Seri Iskandar
Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

CRACK PREDICTION OF USING FINITE ELEMENT ANALYSIS

by

Muhammad Shahrul Izzwan bin Salim
22646

A project dissertation submitted to the
Mechanical Engineering Programme
Universiti Teknologi PETRONAS
In partial fulfilment of the requirement for the
Bachelor of Mechanical Engineering with Honours

Approved by,

____
(Dr. Tuan Mohammad Yusoff Shah)

UNIVERSITI TEKNOLOGI PETRONAS
BANDAR SERI ISKANDAR, PERAK
Jan 2020

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own as specified in the references and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.

MUHAMMAD SHAHRUL IZZWAN BIN SALIM

ABSTRACT

In current developing world, the use of simulation software has increased rapidly for this past few year. Many fields of study took advantage of simulation software including finance, medical and engineering. The development of simulation software is becoming more advance due to high demand from various industry. In fact, finite element analysis (FEA) are widely used in engineering industry to analyse the behaviour of designed structure element etc. This approximation method is promising a good and reliable results. Thus, the focus of this study is to establish a methodology for prediction of fatigue life (cycles) of a plate with hole in 3-dimensional structured model by using FEA where the data distribution from the FEA will be using for more advanced research. By using FEA, the specimen will be tested with different magnitude of uni-axial constant amplitude cyclic loadings. At the end of this project, the prediction of fatigue life analysis by using FEA model is obtained and compared with the experimental results. The relative percentage of error for these results are calculated and observed.

ACKNOWLEDGEMENTS

First of all, praises and thanks to the God, the Almighty, for His showers of blessings throughout my foundation and undergraduate year of studies. I would like to express my gratitude to my beloved Universiti Teknologi PETRONAS for the great past 5 years experiences. I would also like to thank everyone who had been helping me throughout this journey especially my lectures and friends.

My deep and sincere gratitude to my final year project supervisor, Dr. Tuan Mohammad Yusoff Shah for the great opportunity given to me to carry research under his supervision. I appreciate those time spent to guide me throughout this study period. Not to forget his assistances, Shaz and Ain who helped me a lot by providing some necessary information.

I am grateful for having parents who constantly loving, caring and pray for my success and tried their best to contribute to this project. Lastly, I would like to thank my friend, Najwa, who always gave her full commitment to help me in this study.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1

INTRODUCTION

1. Background of Study

The world's rising energy demand is driving the pipeline industry's growth in all countries. Based on statistical information, it is the safest and most economical method to transport gas and oil through pipelines [1]. Nearly 90% of the pipelines are made of steel, mainly carbon steel, with the remaining 10% of aluminium, fiberglass, composite, polyethylene and other types [2]. Higher world oil and gas demand is increasing the pipeline's capacity and operating pressure. It is becoming more important to provide higher strength pipeline material, more development of welding techniques and reliable detection of defects.

The oil and gas pipelines are permanently subjected to vibration emanating from different sources. The most important vibration sources are listed as below [3]:

1. Pressure pulsations at discrete frequencies. This kind of vibration is generated when loading is induced at the rotational speed of compressor.
2. The vibration of the structure.
3. Pressure fluctuations which is caused by turbulence in the flow or passing of the flow over the narrow or complex path.

Since the compressor's operating point changes based on gas demand, it can cause vibration in the pipelines in particular. These continuous vibrations in the critical region of the pipelines can lead to fatigue crack initiation and propagation. These types of vibrations induce various faults such as the initiation and propagation of longitudinal and circumferential cracks in the pipeline's critical areas such as the welding area in small bore connections.

To reduce the likelihood of such fatigue crack failure, a reliable method is needed to evaluate the pipeline's critical region. Finite element analysis (FEA) is a numerical method to solve the problem of engineering. Javadi, Tan and Zhang said the method of finite elements was widely used as a powerful tool in engineering problem analysis [4]. This statement is supported by Levin and Lieven [5] as they said that the application of FEA is widely used in the industry to model the behaviour of physical structures due to its high accuracy in the solution. FEA can use numerical methods to identify important parameters such as stress, heat, propagation of cracks and displacement. This method gives the industry an advantage because, instead of making models and conducting experiments, FEA will reduce costs by doing simulation.

Present research illustrates the technique of finite elements followed to estimate the structural element's fatigue life up to the initiation of crack and the evaluation of fatigue damage at crack launch. Crack initiation approach was used for assessing fatigue life and damage. Estimation of the fatigue life was rendered dependent on the criterion of Strain-life. Morrow's equation was used to measure the life of fatigue under a constant cyclic loading amplitude. Life of fatigue so calculated was used to assess life of fatigue under variable amplitude load. Continuum risk law for predicting cumulative damage under variable amplitude loading has been applied.

1.1 Problem Statement

According to Vipin W and Rashmi H , fatigue analysis through numerical simulation has been proved to be an effective method for fatigue life and damage prediction [6]. In fact, accurate fatigue life estimation is the most important element to ensure the structural integrity of the component throughout its intended operational life. Therefore, this study is conducted to establish the methodology for fatigue life prediction using FEA called ABAQUS. This methodology will cover from modelling phase up to fatigue life prediction. As the validation process, the result obtained from FEA through fatigue analysis are compared against existing experimental results from literature [6]. The purpose of validation is to demonstrate the effectiveness of this study to provide methodology for fatigue life prediction.

1.2 Objectives

There are several objectives that need to be achieved which are:

a) To establish the methodology for fatigue life prediction
b) To validate the results of maximum Von Mises Stress and fatigue life of FEA with experimental results in literature [6]

1.3 Scope of Study

While conducting this project, there are several scopes of study that need to be fulfilled which are:

a) This study will focus on Finite Element Analysis (FEA) modelling using ABAQUS and fe-safe
b) Mesh convergence analysis of the model

CHAPTER 2

LITERATURE REVIEW

2.1 Strain life approach for fatigue life estimation

For fatigue life estimation, strain based approach has been used for this study. Based on the experimental data on fatigue testing, fatigue behaviour of a material can be characterized by cyclic curves, plotted under constant amplitude, completely reversed straining with constant strain rate. Based on observation, failure initiates at local plastic zone, crack nucleates and grows to a critical size due to plastic straining in localized zones. Cyclic stress and stress data available in [6] conduct using Romberg Osgood relationship has been used for cyclic strain computation.

2.1.1 Cyclic stress strain computation

A material's stress strain behaviour under inelastic cyclic reversals is different from the strain obtained under monotonic elastic cyclic pressure. Cyclic stress stain behaviour is therefore important for accurate strain range and, in effect, accurate prediction of fatigue life using a localized strain-based method. The cyclic stress strain data obtained in [6] utilizing Romberg Osgood relationship equation below:

$$\Delta\varepsilon eq = \Delta\varepsilon^{e} eq + \Delta\varepsilon^{p} eq = \frac{\Delta\sigma \mathrm{eq}}{E} + 2\left(\frac{\Delta\sigma \mathrm{eq}}{2K'}\right)^{\frac{1}{n'}} \qquad (1)$$

Where, $\Delta\varepsilon eq$ and $\Delta\sigma eq$ are the equivalent range local stress and strain, E is Young’s Modulus, *K’* is cyclic hardening coefficient, *n’* is cyclic hardening exponent, and $\Delta\varepsilon^{e} eq$ and $\Delta\varepsilon^{p} eq$ are mean equivalent elastic and plastic strain gauge.

2.1.2 Fatigue Model

From the strain life curve, Morrow modified the baseline of the curve to account for the effect of mean stress is chosen for carrying out the fatigue analysis using FEA. Fatigue strength coefficient in the elastic component has been altered by Morrow for better accurate estimation. Morrow's fatigue model equation:

$$\frac{\Delta\varepsilon eq}{2} = \frac{\sigma' f - \sigma m}{E}(2Nf)^b + \varepsilon' f(2Nf)^c \tag{2}$$

Where, $\Delta\varepsilon eq$ is equivalent strain range, c is fatigue ductility exponent, $\varepsilon' f$ is fatigue ductility coefficient, b is fatigue strength exponent, $\sigma' f$ is fatigue strength coefficient and σm is local mean stress.

2.1.3 Cumulative Damage Model

Fatigue Life estimated for constant amplitude loading have been further used to compute the fatigue life of same structural element under variable amplitude loading. Cumulative damage law established by M.A. Miner and known as Miner's Rule has been used to predict the fatigue life under variable amplitude cyclic loadings [6]. Miner's rule accurately predicts the cumulative fatigue damage up to crack initiation phase due to slip band formations, micro cracks and dislocation. This law states that the damage fraction (D) at given constant stress level is equal to the number of applied cycles (n_i) at given stress level divided by the fatigue life (Nf) at that same stress level. The equation:

$$D = \sum_{i=1}^{K} \frac{n_i}{N_f} \tag{3}$$

Where, n_i is actual cycle count, N_f is average no of cycles to failure, K is stress level, D is the fraction of life consumed by exposure to various load cycles.

2.2 Fatigue analysis using finite element method

Three phases of fatigue analysis have been carried out using

1. Static stress analysis to determine max strain range under given cyclic loading.
2. Estimating the fatigue life.
3. Establishing damage contours.

2.2.1 Static stress analysis to determine max strain range under given cyclic loading

The full stress value is obtained through the use of commercially available ABAQUS tools to perform static analysis. The stress contours have defined region corresponding to the highest stress of where crack is likely to start. Elasto-plastic material model was used to carry out the static stress analysis to capture the stresses for load range. With the aid of Romberg-Osgood eq, the maximum stress value so obtained was used to find the strain range. (1)

2.2.2 Estimating the fatigue life

This is the second step in the study of fatigue. Strain based approach was used to estimate the fatigue life. For an accurate estimate of the fatigue life, the criterion of tomorrow which deals with the mean stress effect was applied. Results of the strain range obtained from the first step using the Romberg-Osgood equation were used to estimate the cycles to crack initiation.

2.2.3 Establishing fatigue damage contours

The accumulated damage from fatigue was estimated using a model of continuum damage. In the individual load cycle, continuum damage has been summed up in this damage model to measure the total damage at the end of the fatigue cycles. This continuum model considers the rate at which damage occurs not to be linear, but to be related to the damage already accumulated from the previous load cycles. An incremental damage procedure was used to measure the amount of loading block repetitions up to the initiation of a crack. An incremental damage procedure measures the block load no resulting in a damage fraction of 0.1. Following this damage parameters are modified as defined in eq. (4) the process for each increase of 0.1 damage fraction has been repeated until the Miners damage fraction is 1.AT at the end of the analysis a damage contour has been developed which can be be used for the crack growth analysis using suitable progressive damage models.

$$\Delta D = \frac{(1-D_i)^{P_i}}{(P_i+1)N_{fi}} \tag{4}$$

Where, ΔD is the damage for the cycles in current damage increment, D_i is the damage current accumulated, P_i is the current damage rate parameter, N_{fi} is the endurance of cycle. P_i, for a cycle is defined by the relationship in eq. (5)

$$P_i = 2.55(\sigma_{max}\varepsilon_a)^{-0.8} \tag{5}$$

2.3 Rainflow Counting Method

Rainflow counting can be used for analysis of fatigue data. This method is able to reduce a spectrum of varying stress into an equivalent set of simple stress reversals. This method succeeds extracted the smaller interruption cycles from a sequence, which indicates the material memory effect seen with stress-strain hysteresis cycles. A case study that has been conducted by [7] was utilising rainflow counting method for its research. The rainflow counting of the stress-time history of the mentioned study is shown in Figure 1 is performed using the developed rainflow algorithm. The stress PSD data shown in Figure 2 are used to calculate fatigue life using other fatigue theories in the same study and expected to have similar fatigue life result because it used the same stress history. Result shown in Table 1 are taken from [7] . It is observed that Dirlik method gives the closest result to that Rainflow counting. Therefore, these approaches are proven to predict fatigue life with better accuracy.

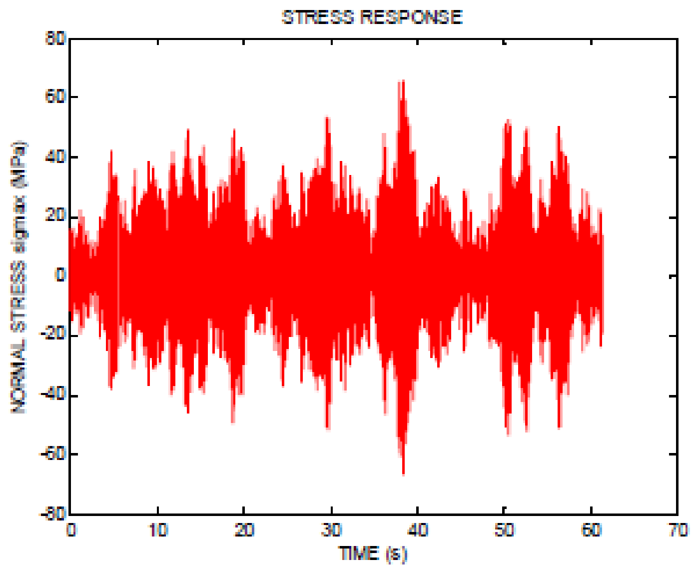


FIGURE 1. Stress Data for 0.001g2/Hz White Noise PSD Input at the Critical Location

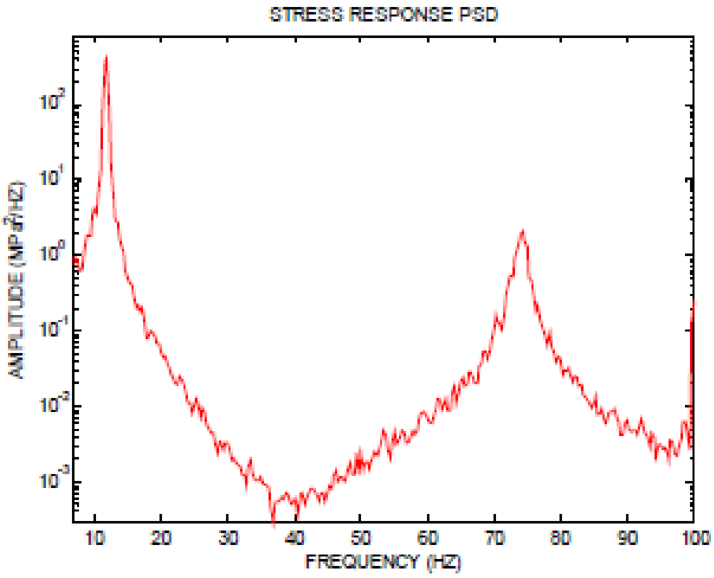


FIGURE 2. Stress PSD Data for 0.001g2/Hz White Noise PSD Input at the Critical Location

TABLE 1. Fatigue Life Result Calculated in Time and Frequency Domains [7]

FATIGUE THEORIES	FATIGUE LIFE (s)	FATIGUE LIFE (h)
FREQUENCY DOMAIN	-	-
Narrow-band	4.14E+09	1.15E+06
Wirching	6.89E+09	1.91E+06
Tunna	9.95E+09	2.76E+06
Hancock	1.72E+07	4.78E+03
Kam and Dover	1.93E+07	5.37E+03
Steinberg	7.08E+06	1.97E+03
Dirlik	1.40E+10	3.89E+06
TIME DOMAIN	-	-
Rainflow Counting	**1.43E+10**	**3.97E+06**

CHAPTER 3

METHODOLOGY

3.1 Problem Analysis

Figure 1 shows the flowchart of research methodology used in executing this project. Based on the flowchart, the first step is to analyse the problem. The main objective of this phase is to identify the importance parameter of this project. Since the targeted output is already identified, which is the fatigue life prediction, the input parameters need to be determined before proceeding to the next stages. The input parameters must have a relation with the output parameter to ensure the data generated is on the right path.

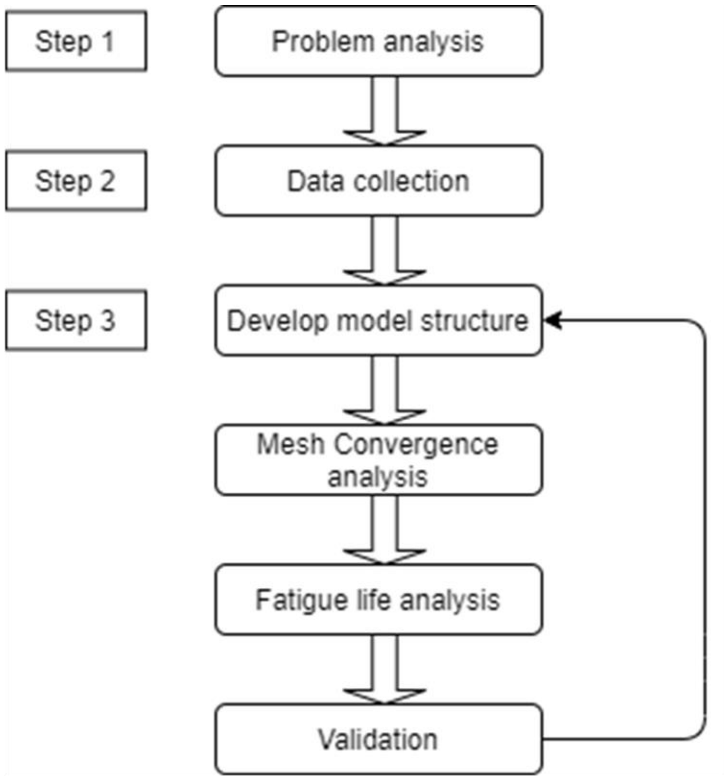


FIGURE 3. Flowchart of Research Methodology

Choosing the right dimension for the model structure is crucial for fatigue life analysis. Therefore, geometrical details of specimen from literature [6] is used to model the structure. Fatigue life analysis is conducted for medium strength steel 100 mm long x 25.6 mm wide x 7.68 mm thick plate with hole of diameter 12.8 mm at the centre of the plate. The plate geometry is shown in Figure 2.

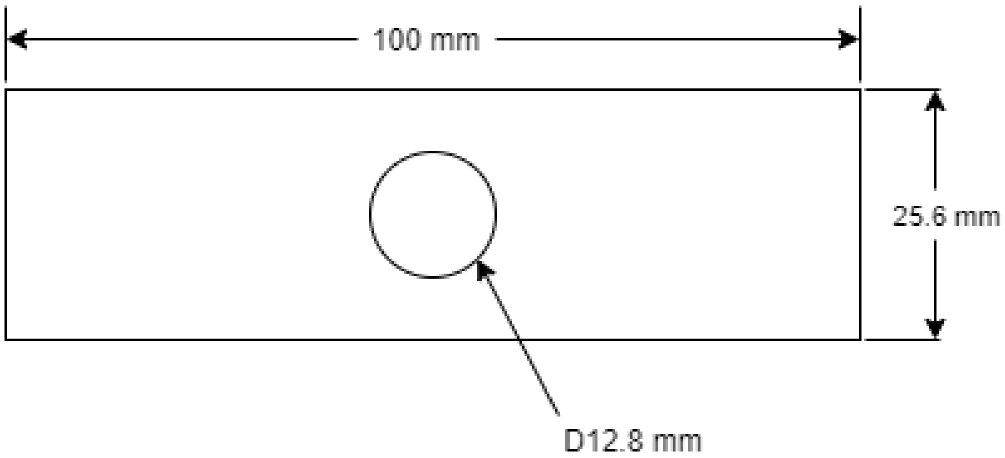


FIGURE 4. Geometrical Details of the Model

3.2 Data Collection

The second step in this project is to collect data regarding the model structure properties. A material used for the model structure is medium strength steel (SAE 130 – has quite similar properties available in fe-safe). Mechanical and cyclic properties of medium strength steel used during analysis have been tabulated in Table 1.

TABLE 2. Properties of Model Structure

Properties	Notation	Values
Modulus of Elasticity (MPa)	E	206900
Poisson's ratio	v	0.32
Yield Stress (MPa)	σ_y	648.3
Ultimate Stress (MPa)	σ_u	786.2
Fatigue Ductility coefficient	ε'_f	1.142
Fatigue Ductility exponent	c	-0.67
Fatigue Strength coefficient (MPa)	σ'_f	1165.6
Fatigue Strength exponent	b	-0.081
Cyclic strength coefficient	k'	1062.1
Cyclic strain hardening exponent	n'	0.123

3.3 Develop Model Structure

The plate with hole at the centre is modelled using three dimensional deformable solid elements. Several analyses have been conducted for various uniaxial constant amplitude cyclic loadings. The loads from literature [6] have been applied along the length direction of the model structure. Loads are shown in the Table 2. The detail of the model structure and mesh details are shown in Figure 3 (a) & (b).

TABLE 3. Load Cases

S.N	Load (kN)
1	62.25
2	56.29
3	53.89
4	47.39
5	40.18
7	31.14
8	25.27
9	22.02
10	20.92

FIGURE 5. FEA Model Structure (a), Mesh Details (b)

The complete model structure has been meshed with C3D8R (8-node linear brick) elements available in ABAQUS software. The mesh global size has been finalised based on the convergence analysis carried out before proceeding for the full analysis of the model structure. ABAQUS has been widely used in many fields such as scientific research and engineering applications. For instance, it has been used to study dynamic crack propagation and mechanical behaviours of composites [7]. However, convergence difficulties are familiar issues while carrying out damage and fracture analysis in ABAQUS/Standard [7]. There are several method of convergence analysis. Manually control global mesh seed approach has been conducted to choose proper mesh size for the model structure. The method basically is trial and error where reducing the mesh seed to increase the number of elements per area of the model structure.

The number of elements and max von mises stress of each mesh seed were recorded to create a convergence plot. The further increase in mesh density stops when the Max Von Mises Stress (Y-axis) showed significantly low in value increased when the number of element increased. This showed that the solution has been converged properly. Based on the Table 3 and Figure 4, the percentage error was 0.051% for the no of elements of 1492216. However, this study used mesh size of 0.3 mm with 861224 no of element to reduce computational time for the analysis with error should be between 0.08% to 0.05%.

TABLE 4. Percentage error of Mesh Convergence Analysis

Mesh size (mm)	Num of elements	Max Von Mises Stress (MPa)	Percentage Error (%)
5.5	105	728.6	-
5	224	708.6	-0.027
2	2884	947.5	0.337
1	23856	1139	0.202
0.5	178845	1236	0.085
0.25	1492216	1299	0.051

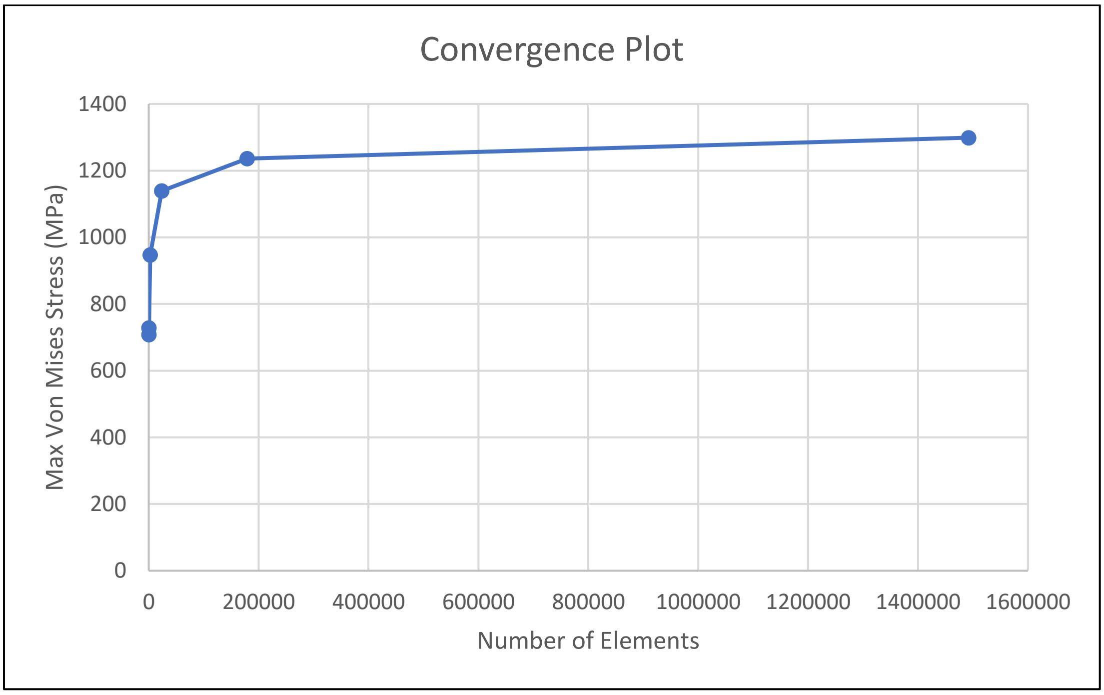


FIGURE 6. Convergence Plot

3.4 Fatigue Analysis using fe-safe

Fatigue is most likely to occur with cyclic loading is induced. However, fatigue is difficult to predict, as it is not visible, and it happens abruptly. Typically, fatigue consists of three stages which are crack initiation, crack propagation and fracture as shown in the Figure 5 [8].

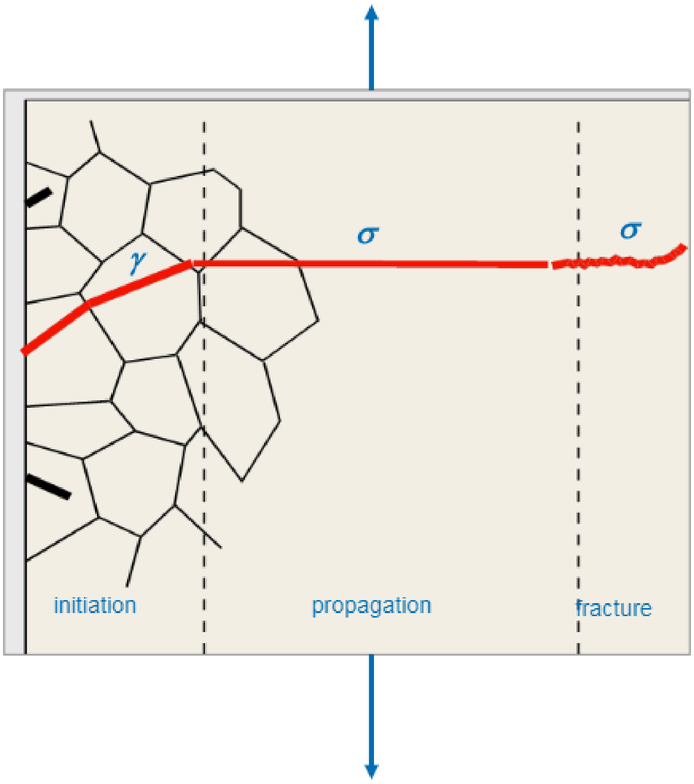


FIGURE 7. Fatigue Phases

For this study, a plate 100 mm x 25.6 mm x 7.68 mm with a hole at the centre (D=12.8) were put under several static uniaxial loads in Table 2. The respectful example loads, and BCs of the model structure can be seen in Figure 6.

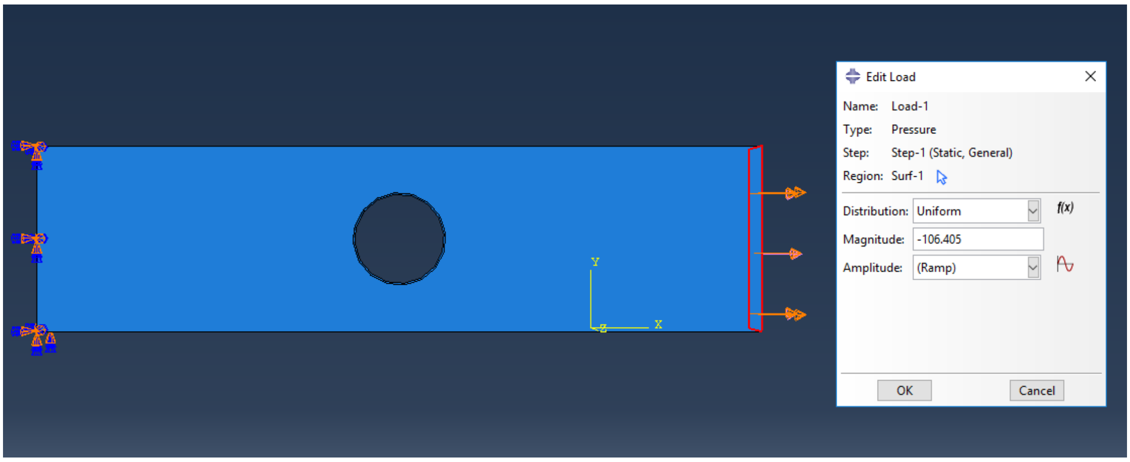


FIGURE 8. Load and Boundary Condition (BC) Details

When the ABAQUS job for the linear elastic model solution is complete, the *.odb file was used as input into fe-safe for further fatigue life prediction. In the fe-safe, the load history applied in the FEA model need to be couple with a sinusoidal signal to produce a fully reversing load cycle. After loading signal is generated, the material SAE 130 was assigned and algorithm that used for fatigue life prediction was selected. According to [8], the Brown Miller strain based algorithm has the highest accuracy within fe-safe for assessing ductile metals. Therefore, Brown Miller: Morrow algorithm has been used in this study for assessing fatigue life (no of cycles to crack initiation). The details of calculation involved for the solutions are already included in the literature section of this paper. The procedures that were described above, are shown in the following figures:

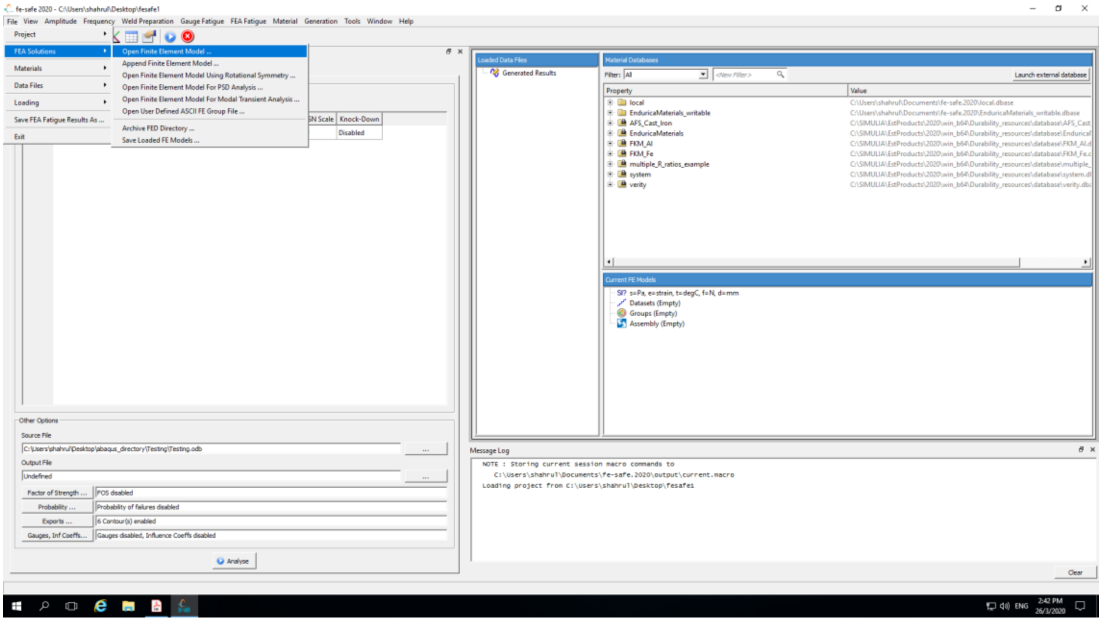


FIGURE 9. Import *.odb FEA results

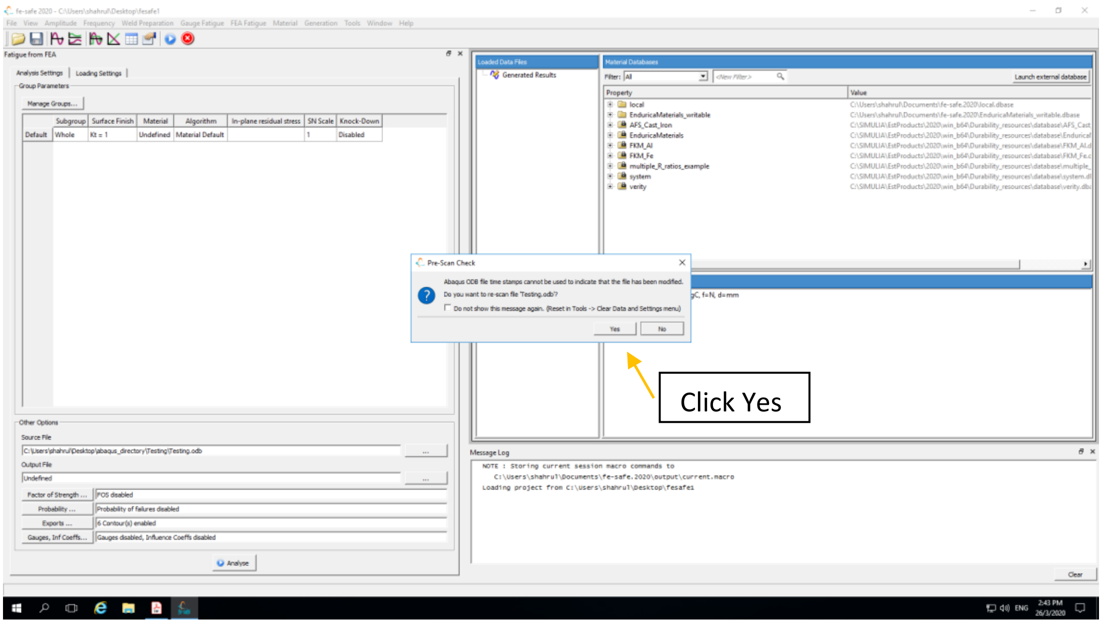


FIGURE 10. Fe-safe prompt user for Pre-Scan Check

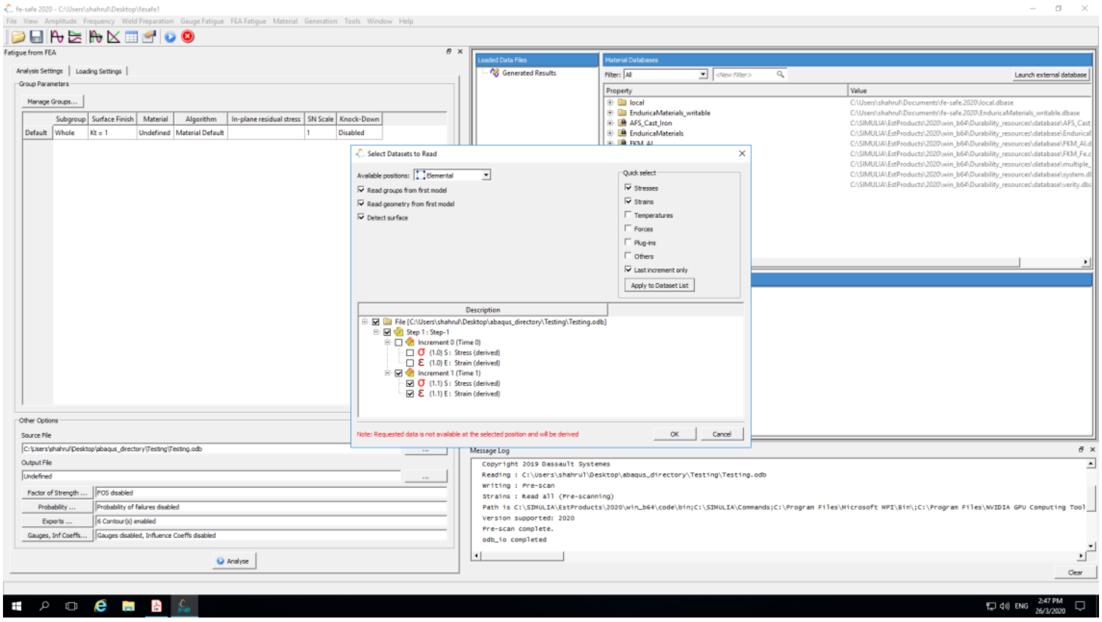


FIGURE 11. Selecting the Datasets

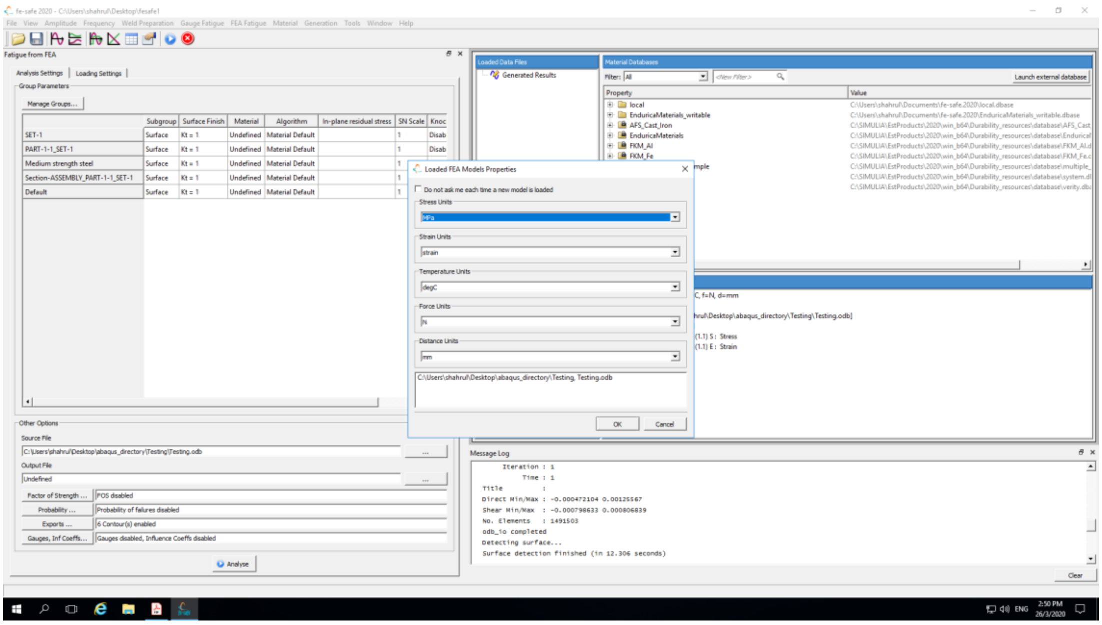


FIGURE 12. Selecting proper Properties Units

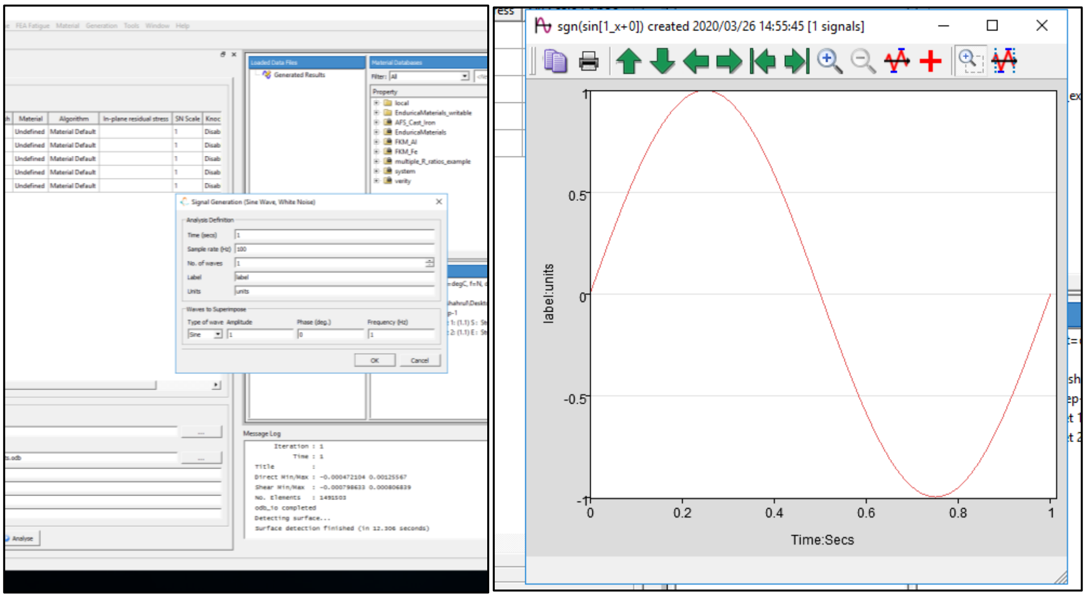


FIGURE 13. Generate Loading Signal

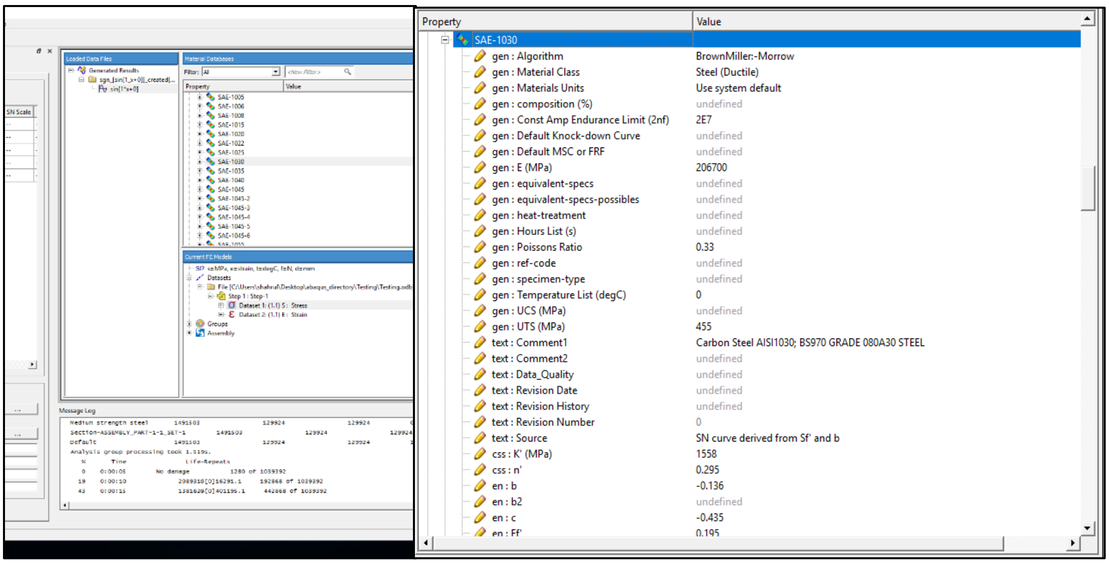


FIGURE 14. Selecting material of the Model Structure

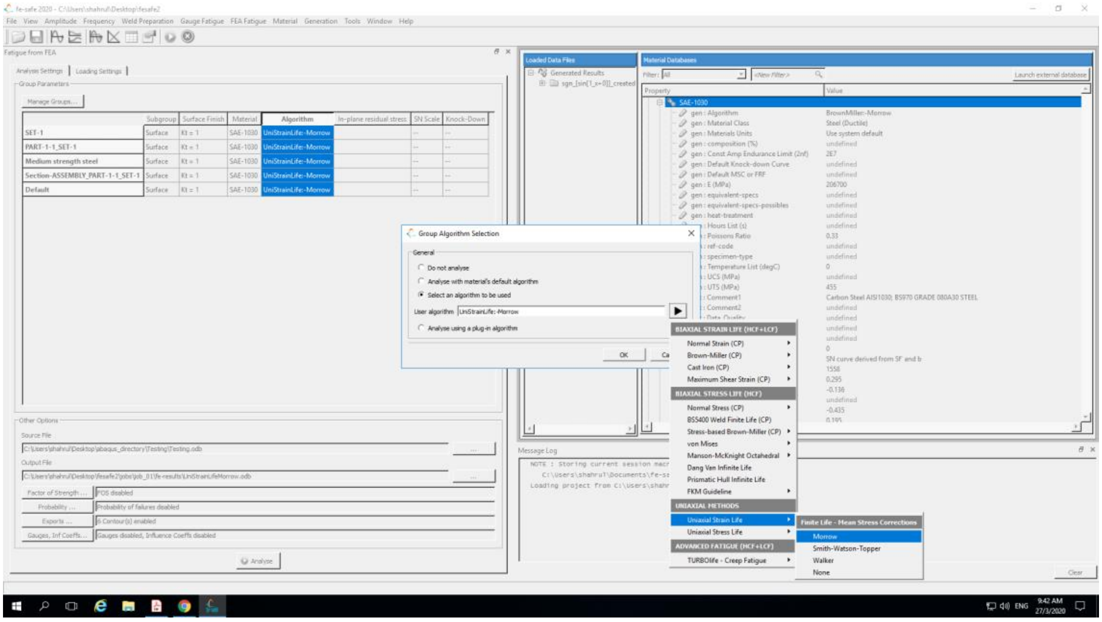


FIGURE 15. Algorithm Selection Tab

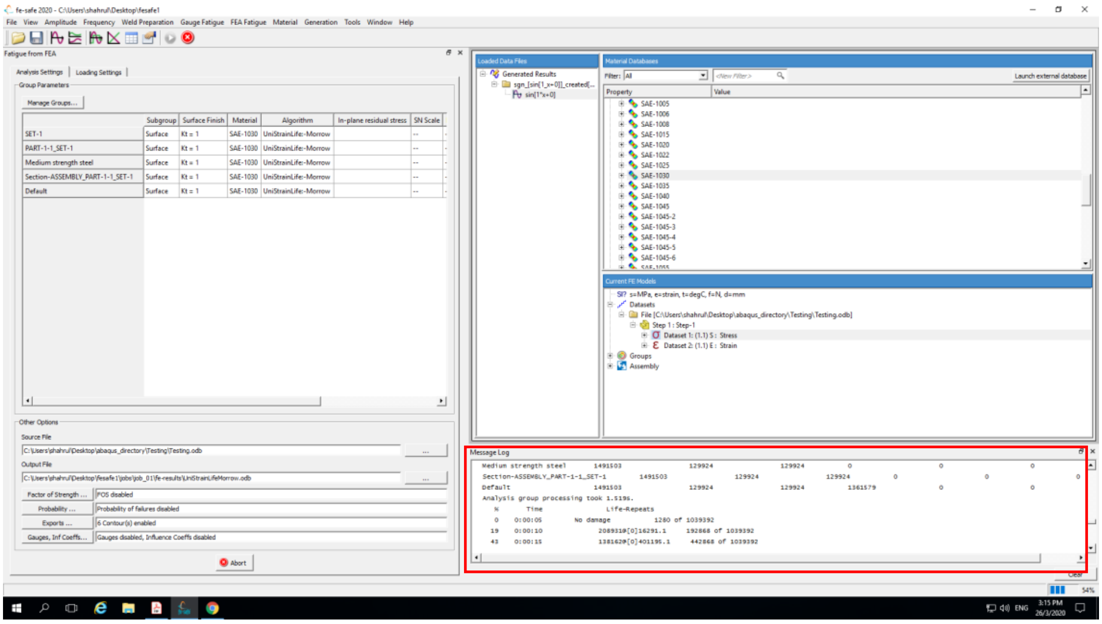


FIGURE 16. Analysis in Process

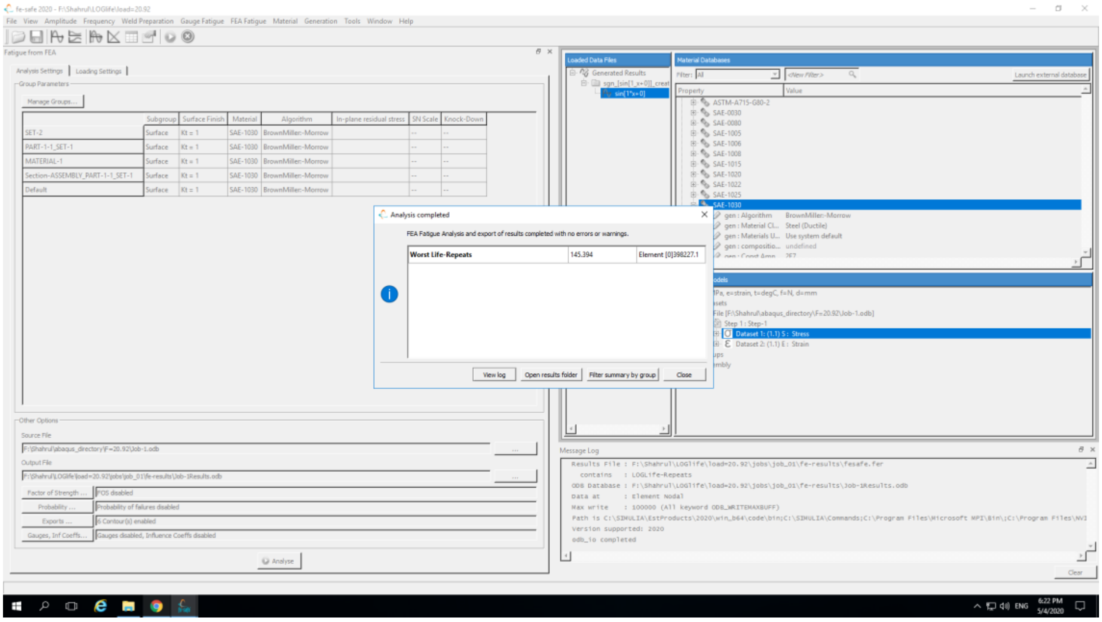


FIGURE 17. Worst Life-Repeats (no of cycles to crack initiation) for Load=20.92 kN

CHAPTER 4

RESULTS AND DISCUSSIONS

4.1 Static Stress Analysis Results

Based on the constant amplitude loads in Table 2, several static stress analysis have been conducted and the maximum Von Mises stress for each load have been recorded through ABAQUS software. These stresses obtained are compared against values available in literature [7]. These values are observed and discussed. The model structure stress contour for load=53.89kN is shown in the Figure 16. The other load results value obtained from FEA are tabulated in the Table 4.

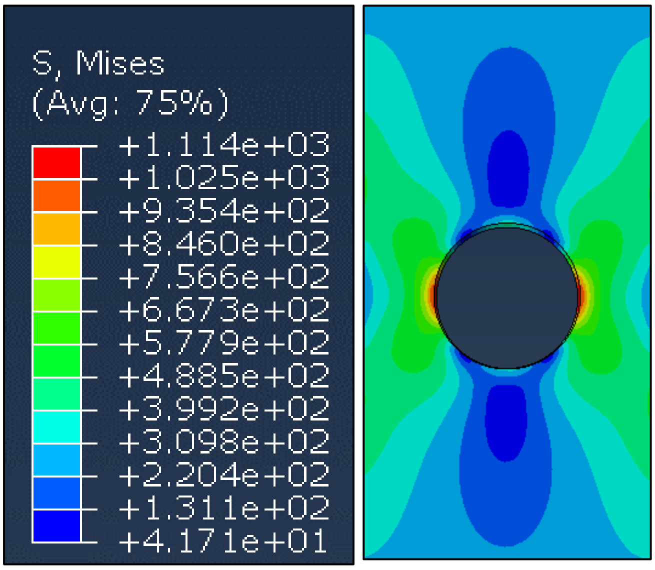


FIGURE 18. Stress Contours for load=53.89kN

4.2 Fatigue Life Predictions for Constant Amplitude Loadings

Fatigue life (no of cycles to crack initiation) obtained through fatigue analysis using fe-safe and its comparison against previous experimental result from literature [7] has been tabulated in Table 4. From the stress contours for all the load cases, the location of crack initiation most likely to occur at the highest stress level in the vicinity of hole, shown in the Table 5. The red zone of the stress contour which showed the highest level of stress indicates the crack initiation location. As mention earlier, the data from fatigue life analysis related to crack initiation can be further used as a basis for more advanced research.

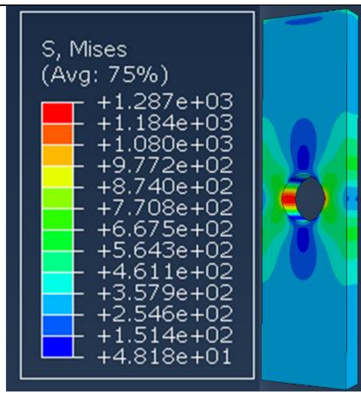
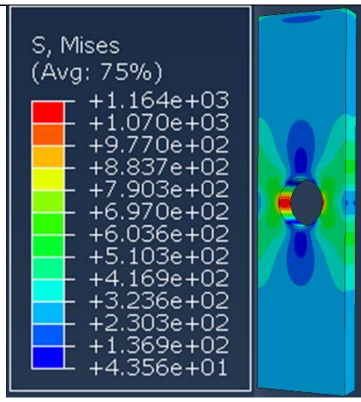
However, the results obtained from FEA were slightly different from literature [1]. From the methodology flowchart figure, the step 3 were repeat as there is error in validation process. All properties of the model structure have been validated again and step 3 were repeated several times to increase accuracy. However, the results were remained unchanged as there might be problems that need to be investigated due to differences in results.

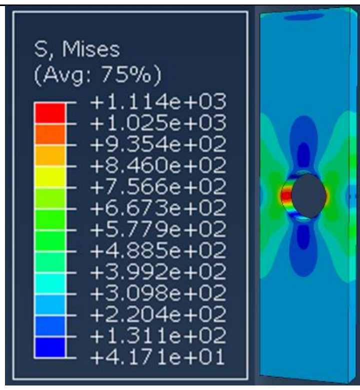
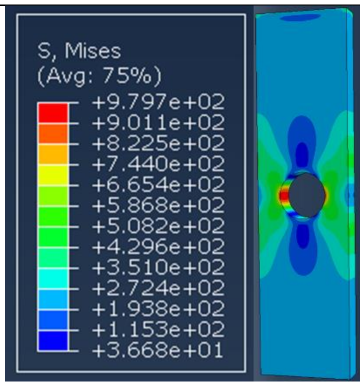
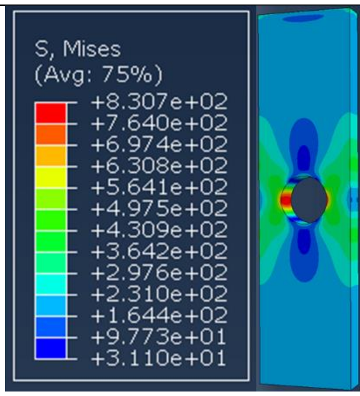
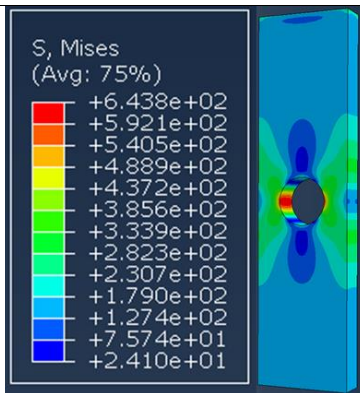
TABLE 5. No of cycle to crack initiation

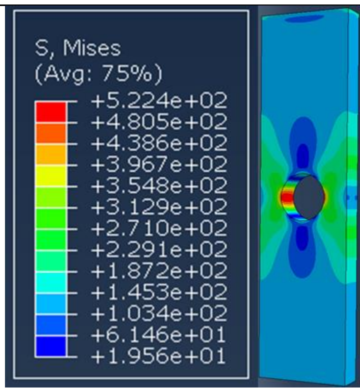
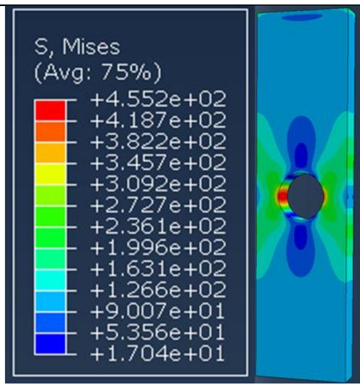
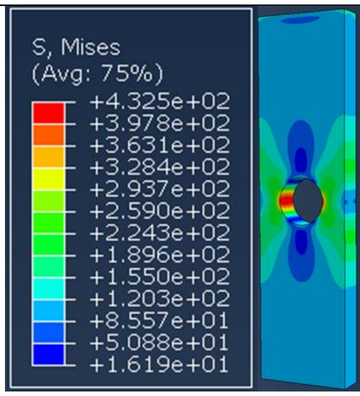
Loads (kN)	Max Von Mises Stress (MPa) Literature [7]	Max Von Mises Stress (Mpa) FEA	Percentage of Error (%) for Max Von Mises Stress	Fatigue Life (no of cycles) by Experiment [7]	Fatigue Life (no of cycles) by FEA
62.25	736.7	1287	42.8	68	145
56.29	681.4	1164	41.5	190	213
53.89	661.8	1114	40.6	265	251
47.39	612.6	979.7	37.5	1250	411
40.18	563.9	830.7	32.1	2400	779

31.14	502	643.8	22.0	11500	2134
25.27	448.7	522.4	14.1	55400	4984
22.02	409.2	455.2	10.1	160780	8836
20.92	394.6	432.5	8.8	188000	10969

TABLE 6. Fatigue life and Crack Initiation Location of this study

No	Load (kN)	Fatigue life	Location of Crack Initiate
1	62.25	145	
2	56.29	213	

3	53.89	251	
4	47.39	411	
5	40.18	779	
6	31.14	2134	

7	25.27	4984	
8	22.02	8836	
9	20.92	10969	

CHAPTER 5

CONCLUSIONS AND RECOMMENDATIONS

5.1 Conclusions

As a conclusion, the main two objectives of this project are achievable. The first objective is to establish the methodology for fatigue life prediction. The model structure is constructed using an established dimension and model properties in the previous studies. Hence, the data is proven. Fatigue life analysis using strain based approach is used in this study for better accuracy of fatigue life prediction. As for the second objective, the obtained results from FEA is compared to the data from previous studies. The comparison of the data is unreliable because the percentage of error is not constant for each load's cases. Some of the error are exceeding 40 percent. The methodology has been repeated several times and still unable to solve. However, the error in the data obtained can be reduced with a further investigation by identifying the other approaches of fatigue analysis prediction through previous studies that available.

5.2 Recommendations

There are several recommendations to improve this project in near future. Fatigue analysis for the 3-dimentional model are too complex for the solver to compute because it involved more element in the structure which take longer time for the solution. Therefore, this study should focus more on finding suitable specimen for 2-dimentional model with available experimental data provided by previous studies and thus the desired results could be improved.

REFERENCES

[1] Y.F. Cheng, Stress Corrosion Cracking of Pipelines, John Wiley & Sons, First Edition 2013.

[2] Canadian Energy Pipeline Association (2007) Stress Corrosion Cracking: Recommended Practices, 2nd ed., CEPA, Calgary, Alberta, Canada.

[3] M. Wastling, M. Kroon, R. Andrews, T. Miles, Development of vibration assessment and screening methods for attachments to pipeline systems, The American Society of Mechanical Engineering, 2009.

[4] A. A. Javadi and T. P. Tan, 'Neural network for constitutive modelling in finite element analysis', p. 10.

[5] R. I. Levin and N. A. J. Lieven, 'DYNAMIC FINITE ELEMENT MODEL UPDATING USING NEURAL NETWORKS', Journal of Sound and Vibration, vol. 210, no. 5, pp. 593–607, Mar. 1998.

[6] V. Wagare and R. Hundekari, 'Fatigue life and damage prediction of plate with central hole using finite element method', vol. 3, p. 5, 2015.

[7] Y. Eldoğan and E. Cigeroglu, 'Vibration Fatigue Analysis of a Cantilever Beam Using Different Fatigue Theories', in Topics in Modal Analysis, Volume 7, R. Allemang, J. De Clerck, C. Niezrecki, and A. Wicks, Eds. New York, NY: Springer New York, 2014, pp. 471–478.

[8] H. W. Wang, H. W. Ji, Y. Sun, and H. Miao, 'Discussion on Convergence Issues in ABAQUS/Standard while Carrying Out Damage and Fracture Analysis', AMR, vol. 189–193, pp. 2247–2250, Feb. 2011, doi: 10.4028/www.scientific.net/AMR.189-193.2247.

[9] N. Mavrodontis, 'Fatigue Analysis with Fe-safe', Simuleon FEA Blog, Jan. 2019, Available: https://info.simuleon.com/blog/fatigue-analysis-with-fe-safe

APPENDICES

APPENDIX A: Gantt Chart

FYP	Detail / Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14
1	Selection of project title														
	Writing literature review														
	Familiarisation with FEA software														
	Identify input and output parameters														
	Analyse the data														
2	Learning the fatigue analysis approach														
	Modelling the the specimen structure														
	Conducting fatigue analysis with FEA														
	Analyse the output data														
	Compare the data														
	Conclusion														

FYP	Detail / Week	-1	1	2	3	4	5	6	7	8	9	10	11	12	13	14
1	Selection of Project Topic															
	Preliminary research work															
	Submission of progress assessment 1 (*SV*)															
	Proposal defence															

	Submission of interim draft report															
	Submission of progress assessment 2 (*SV*)															
	Submission of interim report															
	Project work continues															
	Submission of progress assessment 1 (*SV*)															
	Submission of draft dissertation															
2	Submission of dissertation (soft bound)															
	Viva															
	Submission of progress assessment 2 (*SV*)															
	Submission of project dissertation (hard bound)															