Fatigue Studies On Carbon Steel Piping Welded Materials and Its Connections

Ridzuan bin Mat Isa and Khalil Abdul Razak.

Faculty of Chemical Engineering, Universiti Teknologi MARA

***Abstract*— in this study, the fatigue characteristics in pipe is investigated to increase the knowledge of the fatigue crack growth in pipe. The material used in this study is carbon steel. This study does a simulation of the fatigue crack growth of a created model. This simulation uses a software called ABAQUS, this software uses extended finite element analysis as a platform to simulate the changes happen to the model when in different condition.**

Keywords*— *fatigue, finite element method, Paris law, Abaqus software

I. INTRODUCTION

In oil and gas industry, the usage of pipes and piping system is very basic. There are various processes in oil and gas industry which the application of piping system is very crucial such as crude oil drilling, production, processing, storage and transportation of crude oil. It is very important for the pipes and all its components to function properly in order to achieve maximum efficiency in any processes involve piping system. Pipes comes in different size and compositions. Examples of pipes used in oil and gas industry are stainless steel and carbon steel. Each type of steel have their own size and characteristics depending on the condition of the site. The selection of pipes is also depends on the type of materials to be transported by the pipeline.

During the handling of the pipes, it is frequently exposed to stress and strain that lead to fatigue loading. Continuous exposure of excessive stress to the pipe will lead to fatigue crack growth which will cause mechanical failure or crack. Under normal operating condition, the failure of piping component can happen even though the stress is well below the allowable stress limit. The failure shows that the conventional stress analysis cannot guarantee the integrity of the components under operational condition.

Fatigue is a mechanism that is frequently found in the piping system. The presence of fatigue may cause disadvantages to the integrity of the piping system and its components. Fatigue also can cause crack initiation in the pipe. This is because the some regions may experience high stress or area with the presence of undetected flaws occur to the weld or the components. It is important to make sure that crack initiation will not occur during the service life of the pipe and the equipment. In order to achieve that, the behavior of crack initiation under cyclic loading need to be understand. It will help the prediction of the life of the components.

Previously, the initiation of fatigue crack formation was studied by using small notched specimens. This study was done by evaluating the force of stress or strain at the top notch. The usage of Paris law has been applied to evaluate the fatigue crack growth by using constants derived from compact tension (CT) or Three Point Bend specimens.[1] To understand the fatigue mechanisms and the prediction of fatigue crack growth, various studies have been conducted involving the fatigue crack growth behavior of different types of materials. For the fatigue crack growth prediction, the usage of Paris law in very common. It is very important to be able to estimate the fatigue crack growth rate of a material because it will help the determination of the life of the material.[2]

During the life of a component, it is possible for the integrity of the material decrease over time. Thus, inspection is conducted to determine whether there are formation of crack or not. If crack is detected, then the crack growth is evaluated to determine the integrity of the material. Analytical method can be used for the fatigue crack growth rate prediction which will help in decision making of repairing or replacement of the component.

The function of piping system is to transport any materials required into or out of a system whether it is in form of liquid or vapor. It is essential for the piping system to work properly to efficiently deliver the materials. But the problems lies on the fatigue loading of the pipe and its equipment. This is because fatigue loading is most common reason that lead to failures in the piping system. The failure that involve fatigue loading may occur under normal operating conditions. This may happen because of the presence of flaws in the material of the component that in undetected during inspections of the component.

The integrity of piping component that is subjected to fatigue loading can be analyse through the stress and strain analysis. Failure can also happen because of some mistakes during the selection of material and during the fabrication of the components. Some flaws may be undetected due to poor workmanship or inadequate sensitivity of Non Destructive Examination (NDE) instruments. [3]

Fatigue loading occurs when the pipes is exposed to excessive stress or strain. The definition of stress is the force per unit area of a material. It is an internal force that act in response to external forces. Strain is the deformation of an object due to external forces. It is also defined as extension per unit length.

In most applications, cylinders are most likely to experience cyclic stresses during their normal operating condition and large internal pressures which produce high tension hoop stresses along the inner surface of the cylinder. It may result in internal surface crack that will propagate due to cyclic loading. It is necessary to analyses the behavior of fatigue crack growth to ensure the integrity of the components. This is because after the crack reaches a critical size, the failures may be catastrophic.

II. METHODOLOGY

In oil and gas industry, the usage of pipe and pipeline system is very important as majority of the processes in oil and gas industry requires the usage of pipelines. Therefore, it is very crucial to make sure that the integrity of the pipe and all the component needed in pipeline system can withstand the abuse from environmental condition during its life. In order to achieve that, investigation on Fatigue Crack Growth (FCG) of piping components is required to determine the behavior of fatigue crack growth of pipe and its connections.

The study of fatigue crack growth has been performed using Extended Finite Element Analysis (X-FEM) where ABAQUS software was used. Abaqus is a simulation software usually used for finite element analysis and computer aided engineering, mostly

used in automotive, aerospace engineering and industrial product designing. In this thesis, the software used is ABAQUS version 6.14. In this simulation, the material used is carbon steel. Thus, the properties of carbon steel need to be inserted into the software.
There are a few module that is required to be completed to perform the simulation.

1. Part
This module is the step where the model of the structure is constructed. Model can be constructed using tools in the software. For example, model constructed using lines and circle.

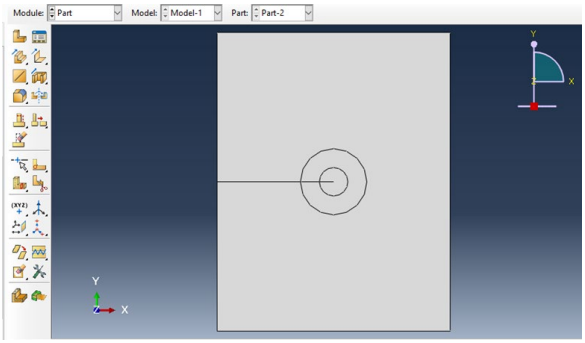

Figure 1: construction of model

In the modelling of structure, the model used is in two dimensional. Model is a rectangle carbon steel plate. A line sketched in the middle to represent the crack opening that will later be used in the simulation.

2. Property
In this module, the properties of the material is inserted. For example, Young's modulus and Poison's ratio.

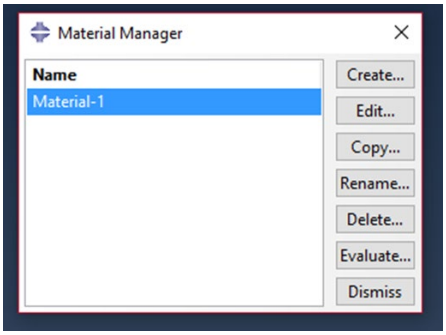

Figure 2: material properties

The Young's modulus and Poison's ratio of carbon steel are 200GPa and o.3 respectively.

3. Assembly
To assemble the parts (In case there are more than one part)

4. Step
To create a step which the model will undergo during the simulation.

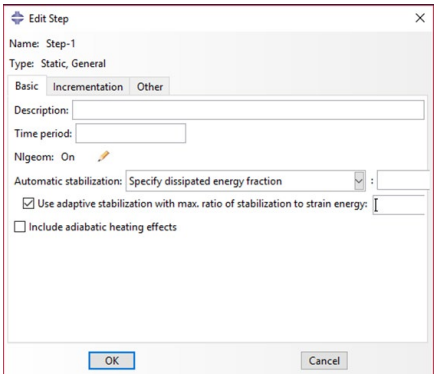

Figure 3: step creation

5. Interaction
To determine the interaction experienced by the model. In this case, the interaction is crack.

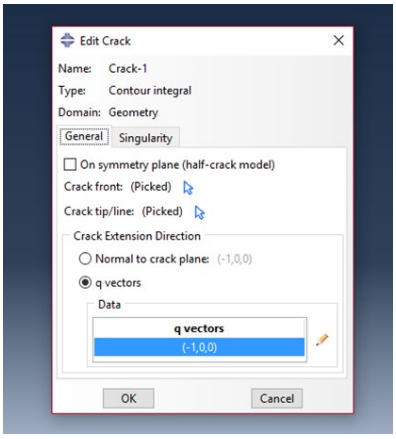

Figure 4: interaction specification

6. Load
Load is assigned to the model along with its magnitude.

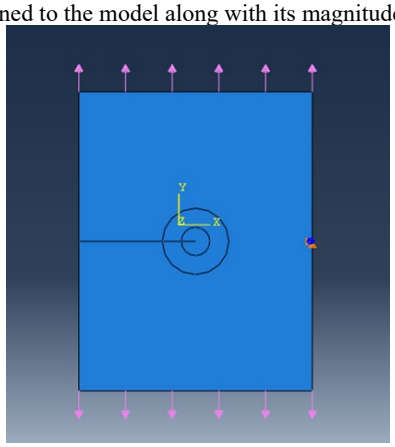

Figure 5: load application

The pressure applied to the model in outward direction to imitate tension force that act on the model.

7. Mesh

Create mesh of the model

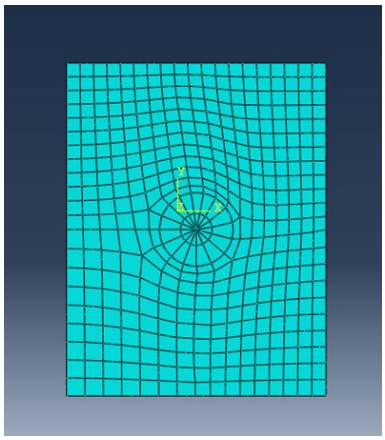

Figure 6: meshing of model

8. Job

Create job so that the simulation can be run.

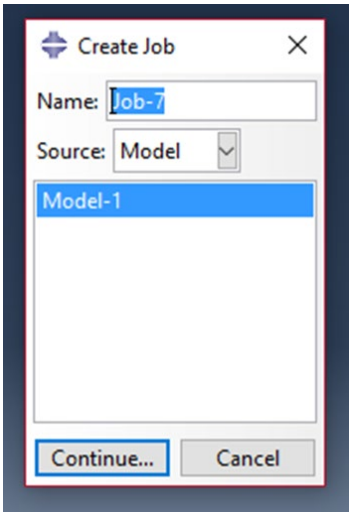

Figure 7: job creation

After all the module is complete, then the job can be submitted to start the simulation. After the simulation complete, the result can be viewed.

III. RESULTS AND DISCUSSION

The result was obtained by running the ABAQUS software to do the simulation of the fatigue crack growth. The simulation involve the application of a series of step toward the model that have been designed. The result that have been obtained is in form of image of the crack propagation of metal plate. The simulation have two part of study. The first one is study of stress over time while the second one is study of the effect of magnitude of force toward the stress.

Part 1: study of stress over time

In this study, the period of step application act as the manipulative variable. There were three value of period that have been used in the simulation which were 20, 40 and 60. All these simulation have the same value of pressure applied to the model which was 1000 psi.

For period of 20, the result obtained is;

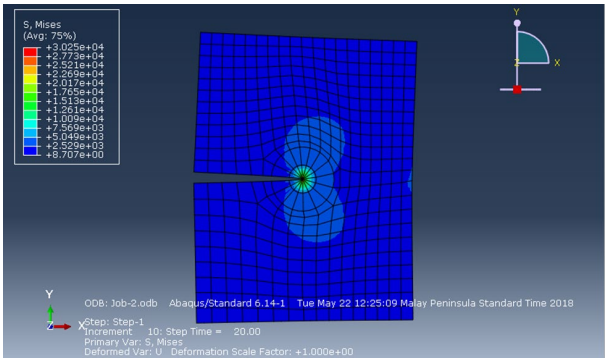

Figure 8: result (20)

For period of 40, the result obtained is;

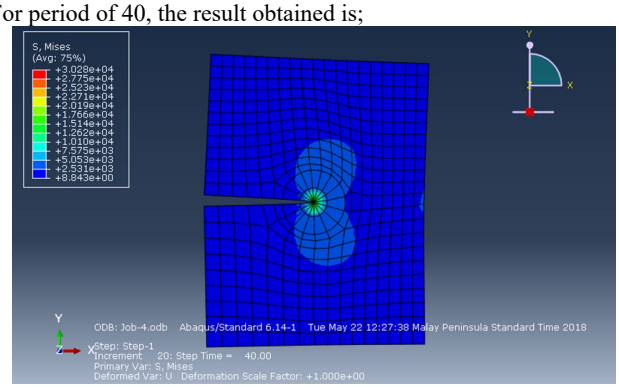

Figure 9: result (40)

For period of 60, the result obtained is;

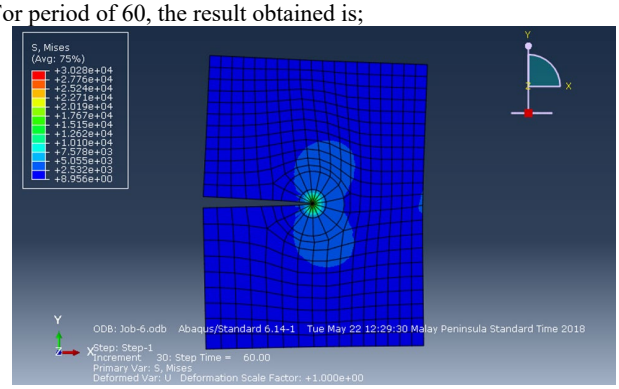

Figure 10: result (60)

Data summary,

Table 1: data summary part 1

Step time		
20	40	60
Mises stress (true stress)		
$3.025x10^4$	$3.028 x10^4$	$3.028 x10^4$
$2.773 x10^4$	$2.775 x10^4$	$2.776 x10^4$
$2.521 x10^4$	$2.523 x10^4$	$2.524 x10^4$
$2.269 x10^4$	$2.271 x10^4$	$2.271 x10^4$
$2.017 x10^4$	$2.019 x10^4$	$2.019 x10^4$
$1.765 x10^4$	$1.766 x10^4$	$1.767 x10^4$
$1.513 x10^4$	$1.514 x10^4$	$1.515 x10^4$
$1.261 x10^4$	$1.262 x10^4$	$1.262 x10^4$
$1.009 x10^4$	$1.010 x10^4$	$1.010 x10^4$
$7.569 x10^3$	$7.575 x10^3$	$7.578 x10^3$
$5.049 x10^3$	$5.053 x10^3$	$5.055 x10^3$
$2.529 x10^3$	$2.531 x10^3$	$2.532 x10^3$
8.707	8.843	8.956

From the result obtained, we can see that the image of crack opening of three sample does not differ much from one another. The only clear difference is that the value of the stress that the model experience. The value of stress is indicated by color as the box at the top left of the result image. The color varies according to the stress magnitude. The color start with blue where the stress is less and end with red color where the stress is the greatest.
Based on the result, there is only a slight increase in the magnitude of stress when the period of step increased. The maximum stress for period 20 is 3.025x104 while maximum stress for period 40 and 60 are 3.028 x104 and 3.028 x104 respectively. Not only the maximum stress that does not have significant increase. All other stress value also does not increase much.

Part 2: Effect of magnitude of force towards the stress

In this simulation, it is different from the first part where the magnitude of pressure is the manipulative variable while the period is constant. There were three value of pressure used in this simulation which were 1000, 2000 and 3000 psi. In real life, it will simulate what will happen if a structure experiences excessive force until the point of breakdown. Usually, excessive force will lead to structural deformation. In this case, simulation is done with the presence of crack opening to investigate the effect of pressure toward the structural integrity of the material.

1. 1000 psi

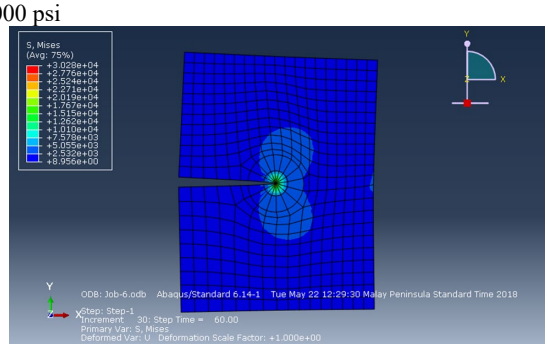

Figure 11: result (1000psi)

2. 2000 psi

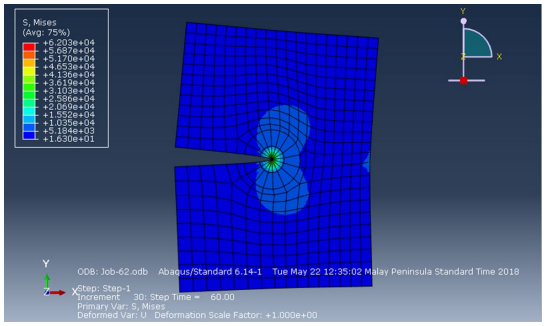

Figure 12: result (2000psi)

3. 3000 psi

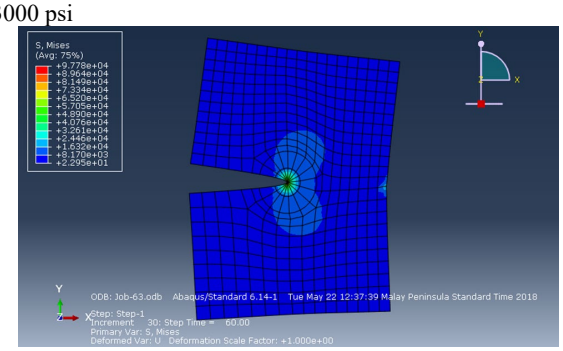

Figure 13: result (3000psi)

Data summary;

Table 2: data summary part 2

Pressure (psi)		
1000	2000	3000
Mises stress (true stress)		
$3.028 x10^4$	$6.203 x10^4$	$9.778 x10^4$
$2.776 x10^4$	$5.687 x10^4$	$8.964x10^4$
$2.524 x10^4$	$5.170 x10^4$	$8.149x10^4$
$2.271 x10^4$	$4.653 x10^4$	$7.334x10^4$
$2.019 x10^4$	$4.136 x10^4$	$6.520x10^4$
$1.767 x10^4$	$3.619x10^4$	$5.705x10^4$
$1.515 x10^4$	$3.103x10^4$	$4.890x10^4$
$1.262 x10^4$	$2.586x10^4$	$4.076x10^4$
$1.010 x10^4$	$2.069x10^4$	$3.261x10^4$
$7.578 x10^3$	$1.552x10^4$	$2.446x10^4$
$5.055 x10^3$	$1.035x10^4$	$1.632x10^4$
$2.532 x10^3$	$5.184x10^3$	$8.170x10^3$
8.956	$1.630x10^1$	$2.295x10^1$

From the results obtained, we can clearly detect the difference between the three samples. The crack opening of all tree samples have different angle. For the pressure of 1000 psi, the opening is quite small. Followed by pressure magnitude of 2000 psi, the opening is much larger than the first sample. The largest crack opening was obtained for the magnitude pressure of 3000 psi.

Based on the results, we can conclude that the crack opening is directly proportional to the magnitude of forces. Apart from that, the result also shows the increase in the value of stress experienced by the model. For pressure of 1000 psi, the maximum stress is 3.028 x104. It is clearly less than maximum stress for pressure 2000 psi and 3000 psi which are 6.203 x104 and 9.778 x104 respectively. It indicate that the higher the pressure, the higher the stress experienced by material.
From the result, we can also see that the stress magnitude varies according to the region.

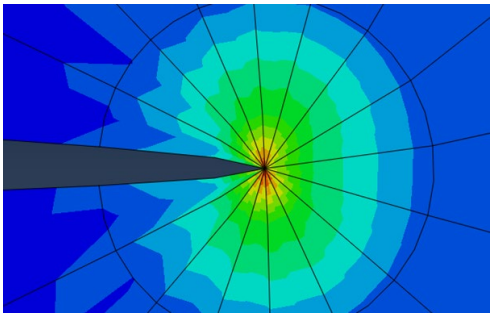

Figure 14: enlarged image of crack tip

Based on the enlarged image of model, a few observation have been made. Firstly, the area near the crack tip is the area that have the greatest stress compared to the other area. The distance from the crack tip definitely is a factor that effect the stress magnitude. Besides that, the result also shows that there are stress that experience by the right portion of the model. This is because of compression that happen due to model deformation.

IV. CONCLUSION

In conclusion, the objectives was achieved because we are able to study the effect of fatigue to the structure of the model. From the result and discussion, we can conclude that the stress increases with time and the stress is effected by the force applied where stress is directly proportional to the force applied to the model. The study of fatigue is very important in the industry. This is because fatigue is one of the main cause for mechanical damage for piping and its components. Stress is one of the main element that effect the formation of fatigue. The presence of stress in the structure will lead to formation damage. This problem then will cause inefficiency of the components of the systems. It is very important to understand the concept of fatigue and its effect to the system and component so that the problem can be solved before it even happens.

ACKNOWLEDGMENT

I would express my gratitude to Allah S.W.T for all His blessing and for providing me with good health and strength to finish this project. Apart from that, I also would like to grab this opportunity to thank my supervisor, Sir Khalil Abdul Razak, for the advice and guidance throughout the project course of Final Year Project 1 (FYP1) and Final Year Project 2 (FYP2).

References

[1] N. Engineering *et al.*, “Fatigue studies on carbon steel piping materials and components : Indian PHWRs FATIGUE STUDIES ON CARBON STEEL PIPING MATERIALS AND COMPONENTS : INDIAN PHWRS,” no. January 2015, 2008.

[2] N. Pugno, M. Ciavarella, P. Cornetti, and A. Carpinteri, “A generalized Paris’ law for fatigue crack growth,” *J. Mech. Phys. Solids*, vol. 54, no. 7, pp. 1333–1349, 2006.

[3] P. Arora, P. K. Singh, and P. Gan, “Fatigue Crack Growth Beha avior in Pipes and Elbows of Carb bon Steel and Sta ainless Steel Materials,” *Procedia Eng.*, vol. 55, pp. 703–709, 2013.