Finite Element Analysis of Disbondment in Thermoplastics Composite Pipe (TCP) Fitness for Surface (FFS)

MUHAMMAD FAKHRU RAZI BIN AZIZI

MECHANICAL ENGINEERING

UNIVERSITI TEKNOLOGI PETRONAS

JANUARY 2020

CERTIFICATION OF APPROVAL

Finite Element Analysis of Disbondment in Thermoplastics Composite Pipe (TCP) Fitness for Surface (FFS)

by

MUHAMMAD FAKHRU RAZI BIN AZIZI
22668

A project dissertation submitted to the

Mechanical Engineering Progranne
Univertsiti Teknologi PETRONAS
in partial fulfilment of the requirements for the Bachelor of Mechanical Engineering with Honours

Approved by,

ASSOC. PROF. Dr. ABDUL RAHIM BIN OTHMAN
Director
Research Management Centre
Research & Innovation
Universiti Teknologi PETRONAS

(AP DR Abdul Rahim Othman)

UNIVERSITI TEKNOLOGI PETRONAS
TRONOH, PERAK
JANUARY 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 except as specified in the references and acknowledgements, and that the original work contained herein have not been undertake or done by unspecified sources or persons

MUHAMMAD FAKHRU RAZI BIN AZIZI

ACKNOWLEDGEMENT

Thanks to the people who assist me throughout this journey. Your cooperation, help, guideline and encouragement given to me allow to complete my project.

My greatest gratitude to my supervisor AP DR Abdul Rahim Othman for all his support, knowledge, guidance and advices that motivate me to complete the Final Year Project as a partial fulfilment of the requirement for the Bachelor of Mechanical Engineering with Honours. Your words and the knowledge you share will benefit me in the future.

I am grateful to Mr Amirul Ariff and his team of research officers who has been helping me obtaining the data that I needed to complete this project. They also help me understand and explain to me the theory behind the theme for this project. Their contribution is a great help for me to complete this project.

My thanks to my family for their emotional support and always cheering me to do my best in the project.

Finally, to my dearest friends who involved directly or indirectly, helping me to complete my FYP project.

Thank you,

Muhammad Fakhru Razi bin Azizi

ABSTRACT

In this work, we study the stress distribution analysis and thermal distributin analysis on TCP composite with failure mode which is cathodic disbondment using finite element model from ANSYS Workbench Software. Mesh sensitivity study in this model has concluded to 12 number of mesh until it produces insignificant difference in stress value as the number of mesh is more sensitive. Mesh sensitivity analysis can be performed by simulating models with different number of mesh and element sizes under the condition applied on the model. The variation of the meshes are done to get accepted level tolerance that can be found out from grid independence test. This is done by varying the mesh size from coarse to fine and checking the output result for each mesh.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1
INTRODUCTION

1.1 Background of Study

Thermoplastic composite pipe (TCP) is a multi-layer composite which consists of fibre reinforced tapes in which the encapsulation is a thermoplastic resin. In thermoplastic composite commonly consist of three layers arrangement which is liner layer, fibre reinforced and outer layer. Both of the liner and outer layer of thermoplastic composite pipe is made from similar material which is polyethylene but with different composition which are PE80 and PE100. The inner layer of TCP is made from high density polyethylene (PE) meanwhile the outer layer is made from low density polyethylene material. On the other hand, the middle part of TCP is glass fibre reinforced polyethylene (GRP). Thus, possible defects and failure on those three layers of TCP will be discussed for better understanding in defects that may occurs during the manufacturing process or service of TCP. The defects occur on the thermoplastic composite pipe is Cathodic Disbondment between PE and GRP. When in operation, pipes are subjected to combined mechanical and thermal loading, a 3D finite element analysis (FEA) model is used to study stress state in a section of TCP under several subjected conditions

1.2 Problem Statement

- Insufficient research papers study in predicting material availability before failure due to disbondment analysis
- To performed thermal distribution analysis based on operation temperature under disbondment condition
- To obtain stress distribution analysis under disbondment condition

1.3 Objectives

This study is performed to achieve the following objectives:

- To determine cathodic disbondment or disbonding analysis on TCP by using finite element analysis (ANSYS Workbench)
- To performed thermal distribution analysis based on operation temperature under disbondment condition
- To obtain stress distribution analysis under disbondment condition

1.4 Scopes of Study

This research is limited to:

- Material: Thermoplastics Composite Pipe (Layers of PE80, Glass reinforced Polymer and PE 100)
- Failure mode on cathodic disbondment
- Conditioned on onshore/offshore/buried pipeline for environmental surrounding
- The pipe is subjected to the flow of fluid in a full stream pipeline

Chapter 2
LITERATURE REVIEW

2.1 CATHODIC DISBONDMENT

Degradation of the FBE layer and subsequent disbondment of the coatings is assumed to be caused by a mechanically assisted cathodic delamination cycle, due to the simultaneous action of chemical degradation and residual stresses. Coatings with a higher curing level of epoxy powders were revealed to exhibit better performance against cathodic degradation cycle. Furthermore, surface pretreatment of the steel substratum will boost the coating performance considerably [1].

The primary cause in the earlier stage of cathodic disbonding may have been the reduction and dissolution of the interfacial oxide. Oxygen and water that moved through the coating significantly impacted cathodic disbonding. So, disbonding decreased significantly if the movement of oxygen and water through the PE coating was fully hedged. [2]

There was no connection between success of a CD and strength of the dry bond. In the presence of 1 Molar Sodium Hydroxide NaOH, improved wet adhesion strength of changed materials implicitly indicates that CD efficiency could also be enhanced by improving wet adhesion strength. Measurement of the contact angle also appears to play a role in the creation of a stronger bond between polyethylene also steel in dry conditions (except for talc-filled PE). The improved CD performance may also be due to a high adsorption ability of talc, which could adsorb species of low molecular weight produced during thermal degradation that occurs during the coating process, thereby improving the between polyethylene and steel in dry conditions. The improved CD cohesive film strength and adhesion at the interface of steel and polyethylene. [3]

2.2 FINITE ELEMENT IN TCP

Finite element modeling allowed the evaluation of stresses produced during coating application and comparison of the effect of pipeline thickness on internal stresses. At the edges of the tubing, the stresses are concentrated near the interfaces, in the adhesive layer. The stress values are very similar, but in the area of the edge effect regardless of the thickness of the coating. Epoxy / steel interface also experiences major stresses over the entire length of the tubing, from several MPa. This value is close to the residual adhesion value of epoxy after moist ageing that may clarify the disbondment found on three-layer coatings.[4]

3D FE model capable of predicting stress in the TCP segment under combined strain, axial tension and thermal gradient was used to investigate the distribution of through-thickness failure for TCP operation. Different combinations of pressure and thermrmal gradient were tested for failure distributions based on von Misses criterion for isotropic liners and interactive Tsai-Hill for fibre-reinforced laminate. Increasing the internal temperature under low pressures induces a dramatic increase in the fault coefficient of the inner liner. Failure estimates will fluctuate less if the TCP riser works under higher pressures with increasing thermal gradients over its service life. This will give the designer a wider range of internal temperatures for fluids. Increasing tension results in a uniform increase in failure coefficient through laminate plies. [5]

The finite element test is used to analyze the distribution of stress in buried gas pipe that is exposed to thermo-mechanical loads and stress concentrations due to geometry changes. Dividing this analysis into two parts. In the first case, stress values for the socket joint of the buried PE pipe were determined so that the stress can be reduced to levels below the allowable values point by applying the proper pipe joints. The effects of thermo-mechanical loads on the stress distribution in buried pipes repaired with patch are well investigated for the second part. Performed using Software ANSYS. Based on the results, maximum stresses of Von Misses occur in the middle of the inner surface of the socket, while maximum values of the above stresses occur in the socket where the inner surface of the socket joins the pipe outer surface. The maximum values of the above stresses are well below the permissible stresses in both pipe and socket, and therefore the introduced socket joint can be used under the described working condition.[6]

When the orthotropic pipe undergoes a pressure greater than 10MPa, the first epoxy-fiberglass plies fail, but this does not mean that the pipe fails entirely. The pipe fails at a burst pressure of 27.17MPa on all the plies and this stage is called functional failure as the composite layer is cracked at 5.17MPa nominal pressure. Composite layer is the material that withstands the highest stresses with a value of 62MPa for the tension on the hoop. The FPF pressure for the open-end or pure internal pressure condition was shown to be the highest, this is because the axial stresses are the lowest relative to the others under these conditions. In addition, the maximum strain and maximum stress criterion tend to overestimate the FPF pressure because they do not take into account the relationship between axial and hoop stresses.[7]

2.3 THERMOPLASTIC COMPOSITE PIPE (TCP)

More advantages in fatigue, impact resistance (toughness), chemical resistance when using thermoplastic.[8] In addition, pipe parts are likely to be welded together, using standard metal connectors or by combining end connectors. Onshore and offshore flow lines, downhole coiled tubing, interference lines and offshore risers are applications for this pipe system, in particular for high pressure and deepwater.

The pipe is more flexible than steel, and lighter. The pipe can be transported and installed from smaller vessels. Besides, this pipe is easier to handle [9]

Excellent strength characteristics, high corrosion and erosion resistance are provided by Fiber reinforced composites pipes.[10] Alternatively, by adjusting the winding angle, the manufacturer can change the strength and rigidity characteristics to design various pipes depending on the different working conditions or specifications. These advantages are highly effective for energy-generation applications.

The matrix of polyethylene provides high melting viscosity and impedes complete impregnation. However, it was found likely that good quality pipes and high failure pressures represent the full strength of the reinforcement. The non-linear strain response of PE pipes can be demonstrated using modified laminate theory to explain the non-linearity of the matrix and the fiber angle variance of the defects. [11]

Polyethylene matrix deliver high melt viscosity hindered full impregnation. However, good quality pipes and attain high failure pressures was found probable to wind reflect the full strength of the reinforcement. The non-linear strain response of PE pipes can be shown using laminate theory modified to take the justification of the non-linearity of the matrix and the variation in fibre angle that occurs as the pipe defects. [11]

Chapter 3
METHODOLOGY

3.1 PROJECT METHODOLOGY

Based on the literature review, the researcher had outlined methods for this project in order to achieve all of the objectives aforementioned. The premise of this project is simple – to investigate the cathodic disbondment in thermoplastics composite pipe (TCP) under several subjected conditions. This project flowchart is as displayed as Figure 1. Method verification was being done to verify the method used in this study following the specification and standards. This is to ensure that errors can be avoided or as minimal it can be and also to produce reliable results. The pipe configuration and specification is modelled and using the exact same boundary condition. Performing the analysis using ANSYS Workbench and validate the data obtained.

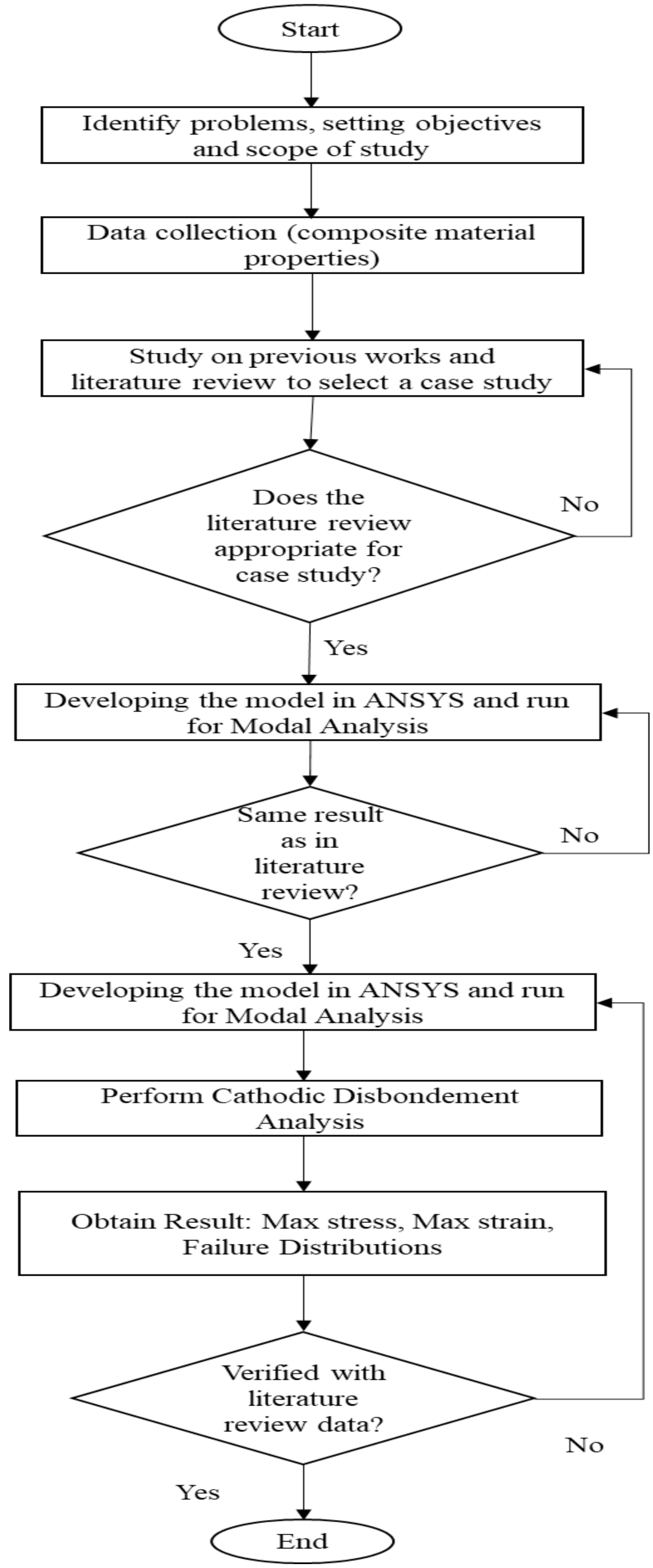


Figure 1: Methodology Flowchart

3.2 FLOWCHART ELABORATION

3.2.1 Thermoplastic composite polymer

The material that have been chosen are High Density Polyethylene 100 (PE100), High Density Polyethylene 80(PE 80) and Glass Reinforced Polymer (GRP). These materials will be analysed on their material properties and behaviors when subjected to cathodic disbondment. These materials have different characteristics as they are matured differently in a mixture of respective matrix and resin configuration. It is crucial to study and understand different composition of materials exhibit different materials properties. Understanding on material failure mechanism and focusing on cathodic disbondment that occur on those materials are important before run the project. So, study and research have been done on these materials that are guided by literature review and finding on discussions cited in this report. The material and mechanical properties are needed upon these materials are density, Poison's ratio, specific heat capacity, Young Modulus and others.

Modelling the materials is the method in analyzing the material structure and reaction when subjected to certain condition. Modelling method that have been done is disbondment on two different materials which are PE80 with GRP and PE100 with GRP. During the modelling part, Catia Software is used to model the pipe. After finish the modelling, the file will be uploaded and imported in (.igs) document into ANSYS Workbench Software. In finite element analysis, the material is modelled to obtain mechanical properties value. A model with correct modelling structure will be guided by comparing the mechanical properties value obtained with literature review findings. Gonzalez (2016) has published the values of physical properties of HDPE which are Young's Modulus, Poison's Ratio and Shear Modulus. Same modelling condition can be emulated to prove the quality of modelling designing in finite element analysis.

3.2.2 Numerical Method: Finite Element Analysis (FEA)

Modelling on material will be done on Catia Software and will be uploaded and imported in (.igs) format document into finite element analysis software, ANSYS Workbench which is reliable and has been established to perform computational fluid dynamics. ANSYS Workbench is introduced to the student to understand the function and operation keys and also for familiarization and tutorial on the exercises. The objective of utilizing ANSYS Workbench software is to generate thermal stress distribution and stress analysis on cathodic disbondment of the material when the model is synthesized by using identified mechanical properties. The finite element analysis is a numerical technique to solve problems of partial differential equations and can be formulated as functional minimization. In terms of nodal values of a physical field which is sought for solving problems. Discretized finite element problem with unknown nodal values is transformed from continuous physical problems. For a linear problem a system of linear algebraic equations should be solved. Values inside finite elements can be recovered using nodal values. [12]

3.2.3 Structure Modelling

For individual structure, the mechanical properties and dimesions of the pipelines are as according to the Table below:

Table 1: Mechanical properties of HDPE

Material	Inner diameter (mm)	Outer diameter (mm)	Poison's Ratio	Young Modulus (Mpa)	Density (Kg/m^3)	Specific Heat, cp (j/kg-k)	Thermal Conductivity (w/m-k)
PE80	59.47	64.77	0.4	1085	957	1850	0.45
PE 100	50.88	55.78	0.4	1340	960	1880	0.44

Table 2: Mechanical Properties for GRP material (Young Modulus)

Material	Inner diameter	Outer diameter	Young Modulus	Young Modulus	Young Modulus
			x-direction	y-direction	z-direction
			(Mpa)	(Mpa)	(Mpa)
GRP	55.78	59.79	35000	9000	9000

Table 3: Mechanical Properties for GRP material (Poison's Ratio)

Material	Inner diameter	Outer diameter	Poison's Ratio XY	Poison's Ratio YZ	Poison's Ratio XZ
GRP	56.18	59.39	0.28	0.4	0.28

Table 4: Mechanical Properties for GRP material (Shear Modulus)

Material	Inner diameter	Outer diameter	Shear Modulus XY (Mpa)	Shear Modulus YZ (Mpa)	Shear Modulus XZ (Mpa)
GRP	55.78	59.79	4700	3500	4700

The value of the property for GRP material is different in every direction because it is considered as anisotropic where the mechanical properties is direction-dependence.

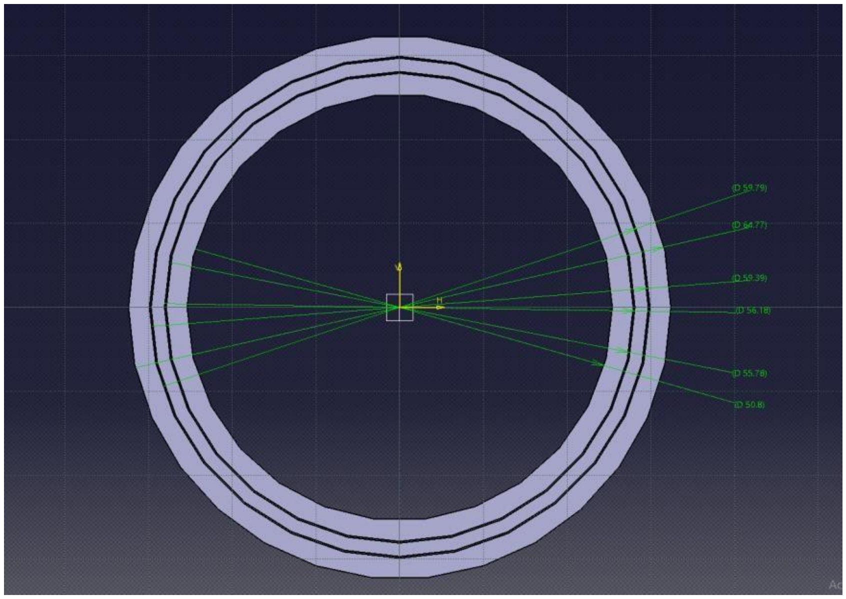


Figure 2: Dimension of composite pipe

Figure 3: Dimension of composite pipe close-up

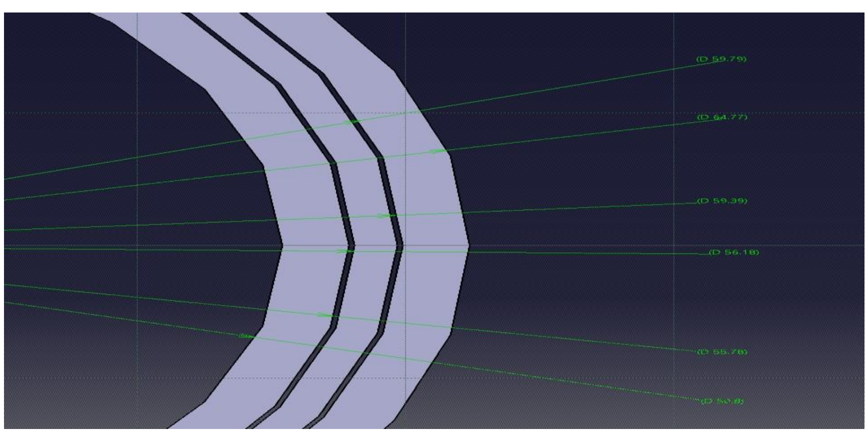
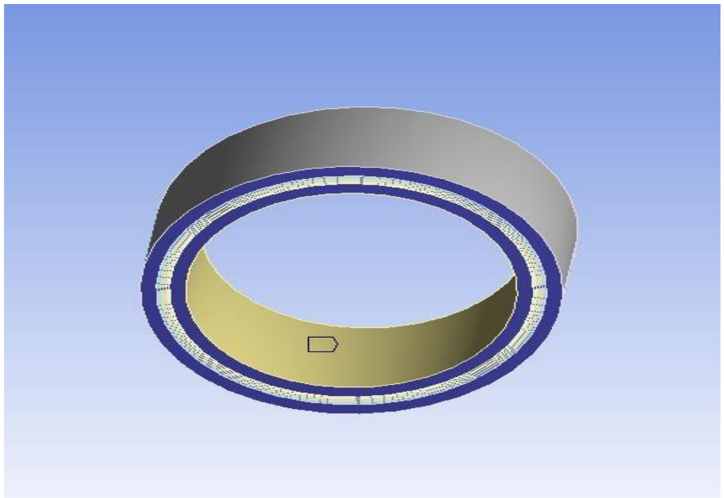


Table 5: Disbondment dimension

Failure	Dimension (mm)
Disbondment between PE100 and GRP	0.4
Disbondment between PE80 and GRP	0.4

3.3 STRESS DISTRIBUTION ANALYSIS WITH BOUNDARY CONDITION OF THE MODEL

The boundary condition was considered which is fixed support. This condition generally applies to pipes subjected to very elastic support, in which the axial displacement in the end is restricted in the normal directions. The model is analysed in term of force reaction that occur on radial and axial direction inside the inner diameter of the composite material



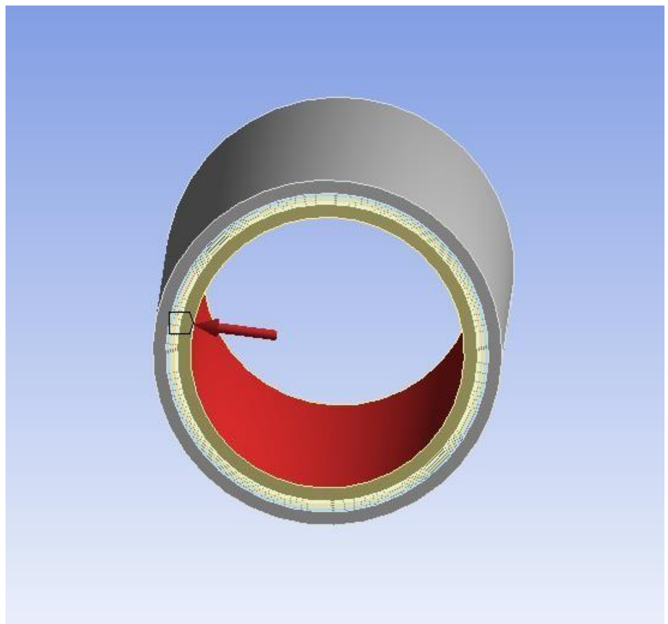
Details of "Fixed Support"

Scope	
Scoping Method	Geometry Selection
Geometry	17 Faces
Definition	
Type	Fixed Support
Suppressed	No

Figure 4: Details of fixed support

3.4 STRESS DISTRIBUTION ANALYSIS WITH INTERNAL PRESSURE APPLIED TO THE MODEL

The internal pressure value applied in the composite pipe is 5.17Mpa. These pressure are known as a nominal pressure or operating pressure. This pressure also is applied to imitate high operational flow of stream of pipelens on offshore industries.



Details of "Pressure"

Scope	
Scoping Method	Geometry Selection
Geometry	2 Faces
Definition	
Type	Pressure
Define By	Normal To
Applied By	Surface Effect
Magnitude	5.17e+006 Pa (ramped)
Suppressed	No

Figure 5: Details of pressure

3.5 THERMAL ANALYSIS WITH MECHANICAL AND PHYSICAL PROPERTIES

For thermal analysis, Fluent is used to conduct all the simulation and given parameter and effort to obtain the data. Usually the flow inside the pipe is in multiphase flow which flow mixture more than 2 fluid components which are gas and liquid. For the physics of the model, the assumption applied are multiphase.

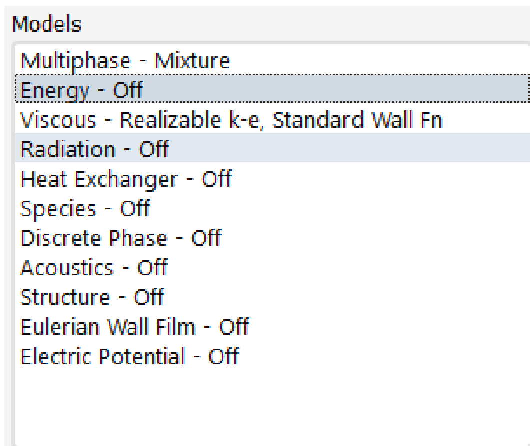


Figure 6: Models assumption

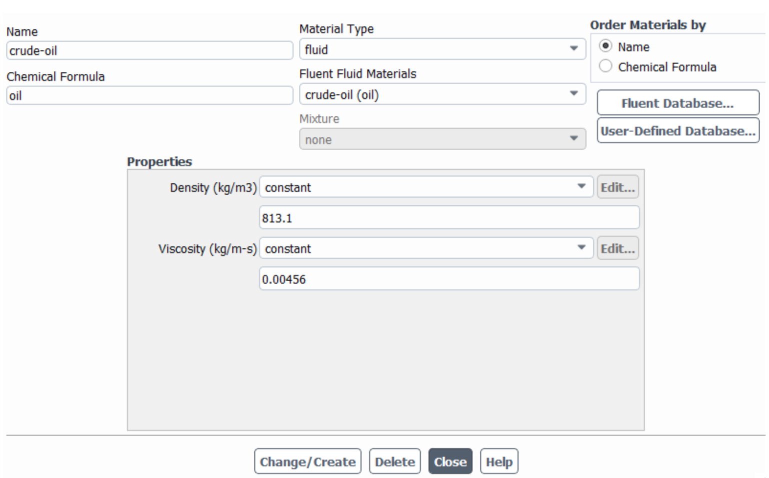


Figure 7: Crude oil properties

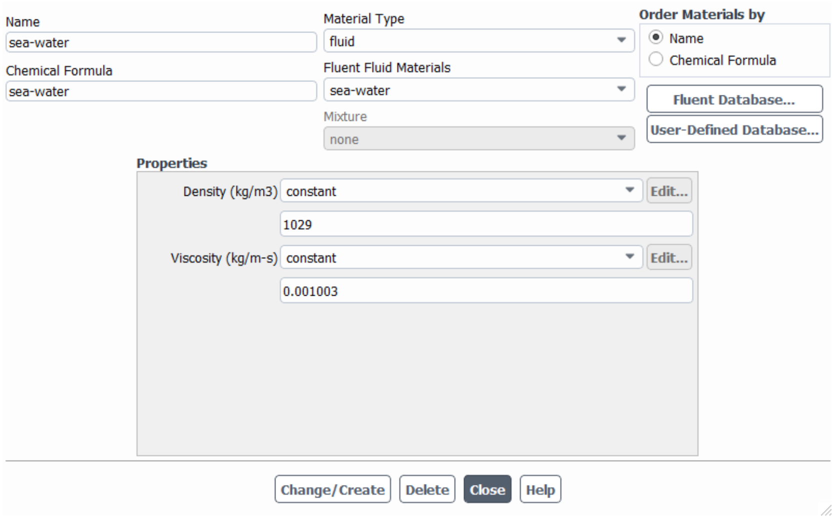


Figure 8: Sea water properties

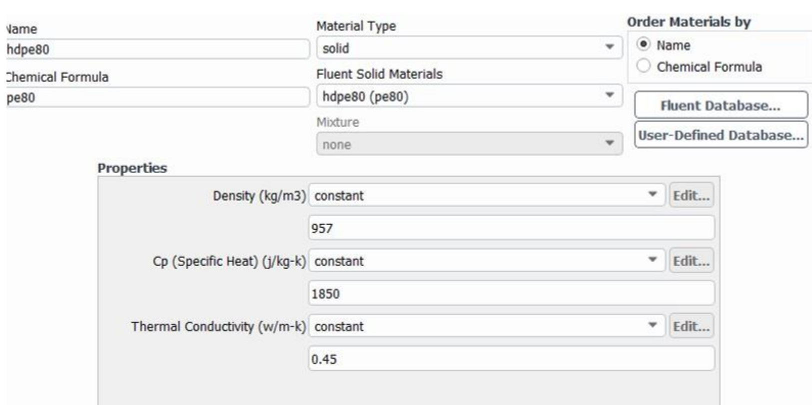


Figure 9: PE80 properties

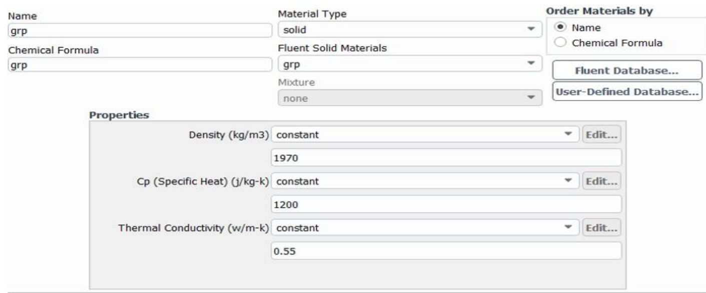


Figure 10: GRP properties

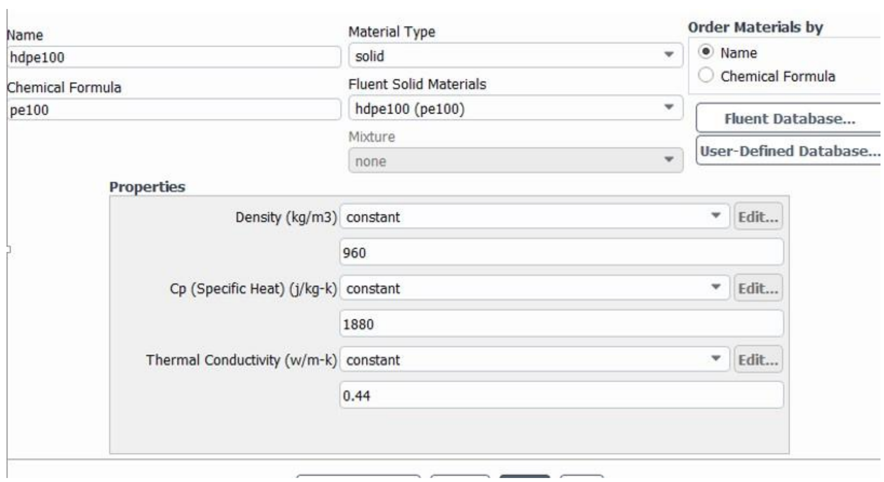


Figure 11: PE100 properties

3.6 THERMAL ANALYSIS WITH BOUNDARY CONDITIONS

In order to obtain results, the assumption need to be adapted to the model of the project as it is a portion of operating pipeline in full stream operation. The boundary condition is to be considered in this model. At inlet of the pipe the temperature is 303K[13] while the temperature at the outlet is 295K due to heat loss.

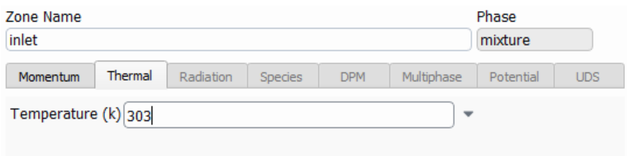


Figure 12: Input temperature

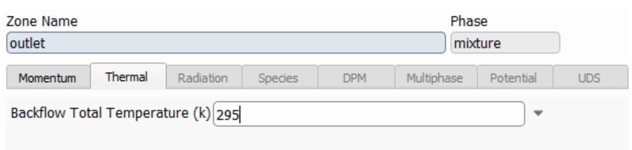


Figure 13: Output temperature

CHAPTER 4
RESULTS AND DISCUSSIONS

4.1 STRESS DISTRIBUTION ANALYSIS

The internal pressure value applied in the composite pipe is 5.17Mpa. These pressure are known as a nominal pressure or operating pressure. This pressure also is applied to imitate high operational flow of stream of pipelens on offshore industries.

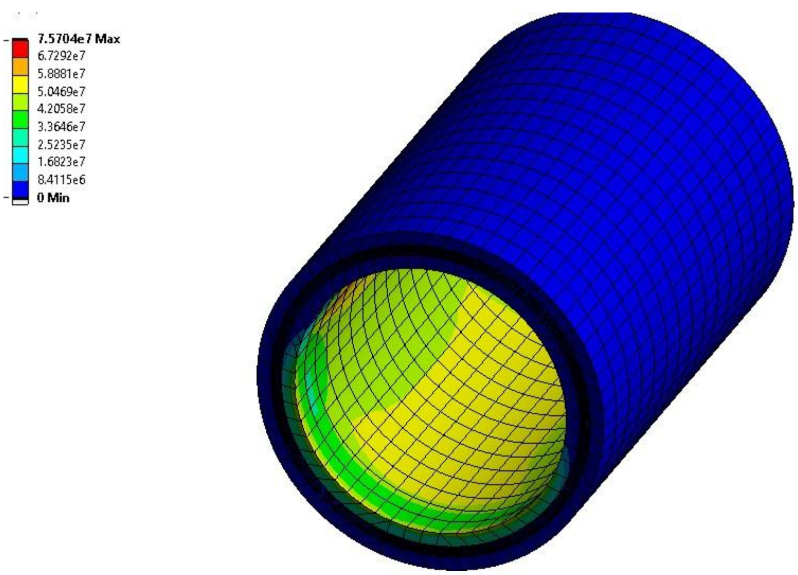


Figure 14: Von Misses stress analysis for the model

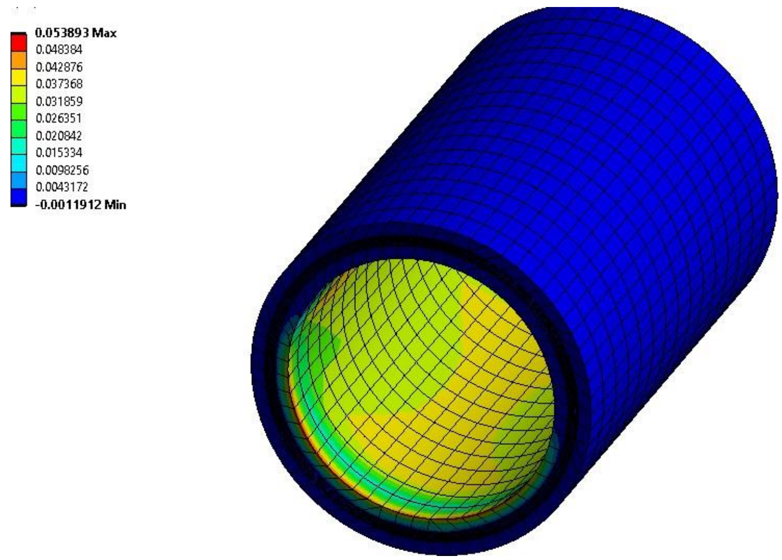


Figure 15: Strain analysis of the model

For the composite pipe, the model has shown significantly impact when subjected to the pressure value. The inner surface of the model is found to be in higher stress compared to the outer surface. High stress in the inner surface was concluded to be the result of high compressive force compared to outer surface applied. The main max stress obtained on GRP generate max stress of 61.4Mpa while from student simulation, the main max stress obtained is 75.7Mpa. The percentage error obtained from max stress occurred is 22%. Although the pipe is in cathodic disbondment the pipeline still can be operated because the max stress obtained is not exceeding the 192Mpa yield stress of the pipe which will not exceeding the plastic deformation.

4.2 MESH SENSITIVITY STUDY

In finite element study, mesh sensitivity will be the one of the contributing factors in obtaining precise result of model analysis. A CFD solution can never be trusted unless by checking whether the result depends on the grid or not. Mesh sensitivity analysis can be performed by simulating models with different number of mesh and element sizes under the condition applied on the model. The output result of a coarser mesh and finer mesh can neither be the same. So, variation of the meshes are done to get accepted level tolerance that can be found out from grid independence test. This is done by varying the mesh size from coarse to fine and checking the output result for each mesh. When varying the mesh does not effect the result much when the test is stop and select that minimum mesh size for the final output solution.

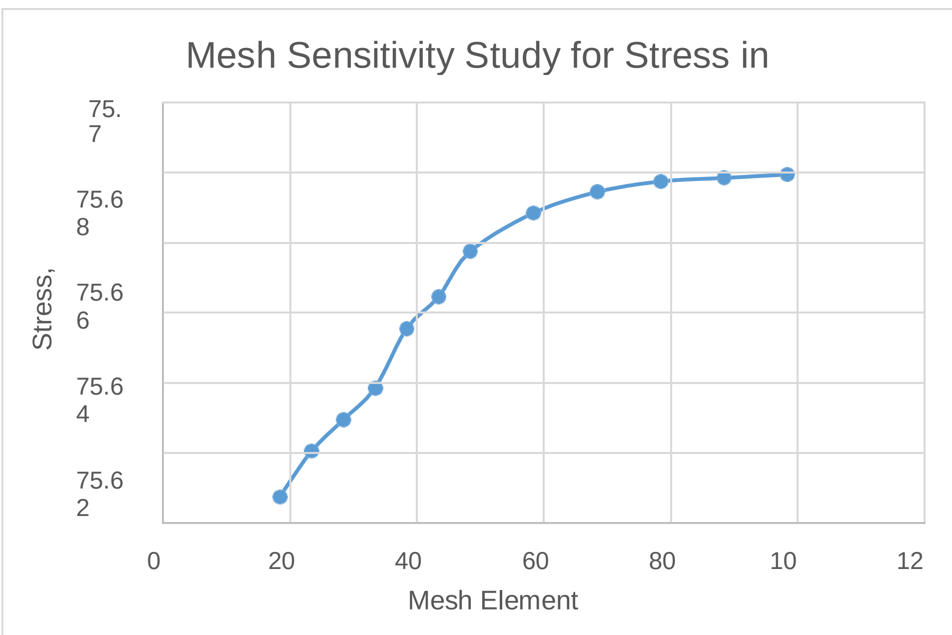


Figure 16: Mesh sensitivity study graph

Table 6: Mesh sensitivity analysis

Mesh Element Number	Von Misses Stress(MPa)	Element number
20	75.589	5466
25	75.602	6106
30	75.611	6726
35	75.62	7366
40	75.637	8085
45	75.646	8766
50	75.659	11846
60	75.67	18062
70	75.576	20586
80	75.679	23066
90	75.68	25482
100	75.681	27726

Based on Fig.(20), result obtained is effective by using mesh number 80. Stress value will change in incrementally when the number of mesh is increase. The result in Stress (Mpa) is increasing rapidly until it reached mesh number 80. From mesh number 80 the stress increasing by low percentage. As finer mesh number is applied on the model, more computational time is required to obtain the result. Mesh size is one of the most crucial element that need to be focused on to obtain good solution in FE analysis. Speed of calculation will be also affected due to unnecessary mesh sensitivity on the model

4.3 THERMAL DISTRIBUTION ANALYSIS

In this analysis, thermal analysis distribution on cathodic disbondment pipe has been done to study the characteristics and the behaviours of the 3 layers pipe. The boundary condition is to be considered in this model. At inlet of the pipe the temperature is 303K[13] while the temperature at the outlet is 295K due to heat loss.

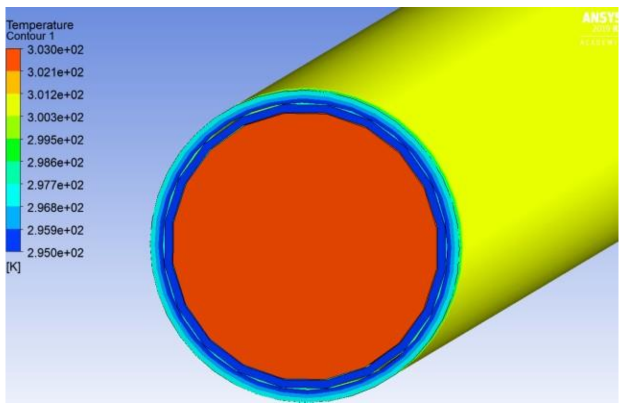


Figure 17: Thermal distribution on the model

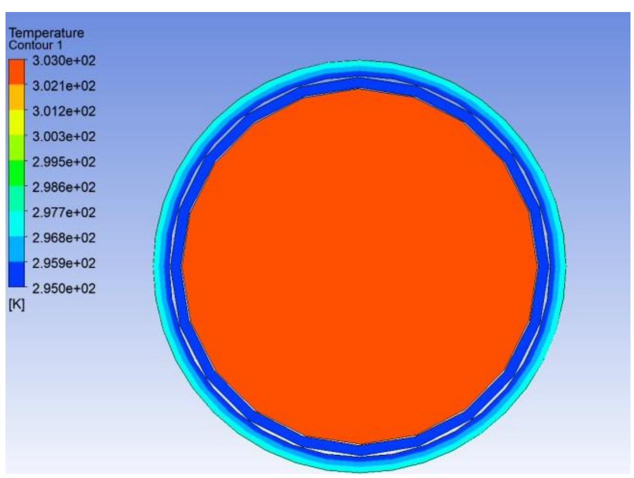


Figure 18: Thermal analysis on partition of the model

In Fig.20 and Fig.21, an observation of high temperature which is 303K was indicated at the center of the pipe due to the flow of the fluid. The temperature is then distributed to the layer of the first composite pipe from the inner which is PE100, the temperature is 295K. The temperature distribution stop at the disbondment area between PE100 and GRP. This is due to when disbond occur between two different materials, these two materials are not contact with each other. Thus, based on understanding on heat transfer, there will be no conduction process occurred between two different layer of materials which are PE100 and GRP but there will be slightly convection process occurred. An observation at low temperature which is 295K was indicated at the outer layer of the pipe which is PE80. The temperature is then distributed to the middle layer which is GRP although there is disbondment between these two different materials. Due to insufficient reference, there is no further discussion on this part.

CHAPTER 5

CONCLUSION AND RECOMMENDATIONS

5.1 CONCLUSION

This line of work has successfully accomplished all the objectives of this project which are to perform finite element study on stress distribution analysis and thermal distribution analysis on a thermoplastic composite material model thus characterize its mechanical properties. Individual material study and stress analysis of the combined composite has been done to observe the characteristic of each material under certain load. Input from individual stress analysis study has proven that the stress value obtained from the model will not exceed yield strength under cathodic disbondment condition. The analysis has identified on max stress which is 75.7Mpa will not form plastic deformation. Mesh sensitivity study in this model has concluded to 12 number of mesh until it produces insignificant difference in stress value as the number of mesh is more sensitive. Mesh sensitivity analysis can be performed by simulating models with different number of mesh and element sizes under the condition applied on the model. The variation of the meshes are done to get accepted level tolerance that can be found out from grid independence test. This is done by varying the mesh size from coarse to fine and checking the output result for each mesh. Finally, thermal analysis is also been performed to study the heat distribution on the composite material under cathodic disbondment condition.

5.2 RECOMMENDATION

The analysis on stress distribution of TCP model can be improved by accomplishing an experimental result under cathodic disbondment condition and compares with simulation study. This can improve the precision of the model and result of the simulation. This study on stress distribution analysis and thermal distribution analsysis on TCP composite under cathodic disbondment condition will contribute in providing mpre data and knowledge to achieve further analysis so that the establishment of an overall study of the composite can be applied in industry and help to improve the performance in the real life application. Due to unforseen circumstancces specifucally covid 19 outbreak the candidates did not have the accessibility facilities and the utilities required in order to gain and obtain more accurate and precise result.

REFERENCES

[1] M. Rahsepar, M. Asgharzadeh, M. J. Hadianfard, and S. A. Jenabali Jahromi, “Failure Analysis of Disbondment of Three-Layer Polyethylene Coatings from the Surface of Buried Steel Pipelines,” *J. Fail. Anal. Prev.*, vol. 15, no. 5, pp. 604–611, 2015.

[2] T. Kamimura and H. Kishikawa, “Mechanism of Cathodic Disbonding of Three-Layer Polyethylene-Coated Steel Pipe,” *Corrosion*, vol. 54, no. 12, pp. 979–987, 1998.

[3] D. Roy, G. P. Simon, and M. Forsyth, “Improved cathodic disbondment performance of polyethylene blends,” *Polym. Eng. Sci.*, vol. 42, no. 4, pp. 781–789, 2002.

[4] Y. Joliff, L. Belec, and E. Aragon, “Influence of the thickness of pipeline coating on internal stresses during the manufacturing process by finite element analysis,” *Comput. Mater. Sci.*, vol. 68, pp. 342–349, 2013.

[5] J. C. Hastie, M. Kashtalyan, and I. A. Guz, “Failure analysis of thermoplastic composite pipe (TCP) under combined pressure, tension and thermal gradient for an offshore riser application,” *Int. J. Press. Vessel. Pip.*, vol. 178, no. August, p. 103998, 2019.

[6] R. Khademi-Zahedi and P. Alimouri, “Finite element analysis to the effect of thermo-mechanical loads on stress distribution in buried polyethylene gas pipes jointed by electrofusion sockets, repaired by PE patches,” *Energies*, vol. 11, no. 10, 2018.

[7] O. A. González-Estrada, J. S. León, and A. Pertuz, “Influence of the

boundary condition on the first ply failure and stress distribution on a multilayer composite pipe by the finite element method," *J. Phys. Conf. Ser.*, vol. 1159, no. 1, 2019.

J. L. C. G. De Kanter and J. Leijten, "Thermoplastic composite pipe: Analysis and testing of a novel pipe system for oil & gas," *ICCM Int. Conf. Compos. Mater.*, 2009.

[8] H. T. and U. K. V. Ahmed Arabi Hassen, "Non-Destructive Investigation of Thermoplastic Reinforced Composites Ahmed," vol. 2, no. i, pp. 617–638, 2010.

[9] T. A. Sebaey, "Design of oil and gas composite pipes for energy production," *Energy Procedia*, vol. 162, pp. 146–155, 2019.

[10] A. G. Gibson and A. Fahrer, *Reinforced Thermoplastic Tubes for Pressure Applications*, vol. 2. Conference Design Consultants, 2000.

[11] M. IZUMI and M. SONOHARA, "Introduction to the Feature," *Igaku Toshokan*, vol. 55, no. 3, pp. 211–211, 2008.

[12] F. A. Idrus, M. D. Chong, N. S. Abd Rahim, M. Mohd Basri, and J. Musel, "Physicochemical Parameters of Surface Seawater in Malaysia Exclusive Economic Zones Off the Coast of Sarawak," *Borneo J. Resour. Sci. Technol.*, vol. 7, no. 1, pp. 1–10, 2017.